Geophysical Research Abstracts
Vol. 15, EGU2013-5893, 2013
EGU General Assembly 2013

Towards CFD modeling of turbulent pipeline material transportation

Amir Shahirpour, Nicoleta Herzog, and Cristoph Egbers
Brandenburg University of Technology Cottbus, Department of Aerodynamics and Fluid Mechanics, Cottbus, Germany (amir.shahirpour@tu-cottbus.de)

Safe and financially efficient pipeline transportation of carbon dioxide is a critical issue in the developing field of the CCS Technology. In this part of the process, carbon dioxide is transported via pipes with diameter of 1.5 m and entry pressure of 150 *bar*, with Reynolds number of 10^7 and viscosity of $8\times10^{(-5)}$ *Pa.s* as dense fluid [1]. Presence of large and small scale structures in the pipeline, high Reynolds numbers at which CO_2 should be transferred, and 3 dimensional turbulence caused by local geometrical modifications, increase the importance of simulation of turbulent material transport through the individual components of the CO_2 chain process.

In this study, incompressible turbulent channel flow and pipe flow have been modeled using OpenFoam, an open source CFD software. In the first step, simulation of a turbulent channel flow has been considered using LES for shear Reynolds number of 395. A simple geometry has been chosen with cyclic fluid inlet and outlet boundary conditions to simulate a fully developed flow. The mesh is gradually refined towards the wall to provide values close enough to the wall for the wall coordinate (y^+). Grid resolution study has been conducted for One-Equation model. The accuracy of the results is analyzed with respect to the grid smoothness in order to reach an optimized resolution for carrying out the next simulations. Furthermore, three LES models, One-Equation, Smagorinsky and Dynamic Smagorinsky are applied for the grid resolution of ($60 \times 100 \times 80$) in (x, y, z) directions. The results are then validated with reference to the DNS carried out by Moser et al.[2] for the similar geometry using logarithmic velocity profile (U^+) and Reynolds stress tensor components. In the second step the similar flow is modeled using Reynolds averaged method. Several RANS models, like K-epsilon and Launder-Reece-Rodi are applied and validated against DNS and LES results in a similar fashion.

In the most recent step, it has been intended to generate an optimized LES solver to model turbulent pipe flow for larger Reynolds numbers. The validations are carried out using experiments conducted in Cottbus Large Pipe Test Facility at BTU as a reference [3]. In the mentioned experimental research, evolution of statistical pipe flow quantities, such as turbulence intensity, skewness and flatness are investigated to clarify the development length needed to achieve fully developed turbulence. These observations take place in a relatively large pipe test facility with an inner pipe diameter of $Di = 0.19\ m$ and a total length of $L = 27\ m$ where a bulk Reynolds number of 8.5×10^5 can be reached.

1. CO_2 pipeline Infrastructure: *An analysis of global challenges and opportunities*, Final Report For International Energy Agency of Greenhouse Gas Program (2010)

2. J. Kim, P. Moin, R. Moser, *Turbulence statistics in fully developed channel flow at low Reynolds number,* J.Fluid Mech. 177, 133-166, (1987)

3. F. Zimmer, E.-S. Zanoun and C. Egbers, *A study on the influence of triggering pipe flow regarding mean and higher order statistics,* Journal of Physics: Conference Series 318 (2011) 032039