Proceedings of
The 5th National Congress on Fluids Engineering
March 26-28. 2008, Jeju, Korea

A Parametric Study for the Design of Gas-Liquid Centrifugal Separator

Suryakant Nagdewe*1, S. J. Lee*, H. D. Kim*2, D. S. Kim**, and K. M. Kwak**

기체-액체 원심분리기의 설계를 위한 매개변수 연구

Suryakant Nagdewe[*1]·오성진[*]·김희동[*2]·김동선[**]·곽경민[**]

Key Words : *Centrifugal Separator, Centrifugal force, Swirl Vane, Tangential Velocity, Separation Space*

Abstract

A gas-liquid centrifugal separator is widely used in industry because of its simple geometry and little maintenance. Also, these separators have considerable advantages over filters, scrubbers or precipitators in term of compact design, low pressure drop and higher capacity. A gas-liquid centrifugal separator is a device that utilizes centrifugal force and low pressure to separate liquid from gas by density difference. Design parameters such as length of separation space, swirl vane exit angle, inlet to outlet pipe diameter ratio, models for separation efficiency and low pressure drop as a function of physical dimension are not available in literature. In present study, length of separation space (from vane to gas exit opening) has been studied using CFD. The 3-D Navier-Stokes equations are numerically solved using a fully implicit finite volume scheme. Based upon the obtained solutions, tangential velocities, centrifugal forces, vortices and total pressure losses are analyzed to find the best design parameters.

1. Introduction

A gas-liquid centrifugal separator is widely used in industry because of its simple geometry and little maintenance. Also, these separators have considerable advantages over filters, scrubbers or precipitators in term of compact design, low pressure drop and higher capacity. In spite of wide use, neither dedicated performance models for swirl tubes, nor detailed information about the flow pointing to the best model assumptions, has been published. Present study deals with the parametric study for the design of gas-liquid centrifugal separator. Two-phase simulation is not considered in this work.

2. Numerical Procedure

A separator geometry with diameter of D=539.8mm has been selected for the present study as shown in Fig. 1. Inlet and outlet pipe diameter is kept same as 0.63D with 0.91D and 0.93D as a length respectively. Diffuser prior to swirl vane has a length of 0.94D and separation space that is the distance from vane to outlet considered as Lv/D equal to 0, 0.37, 0.47 and 1.11. Computational domain is discretized into small cells for numerical calculations. Structured grid is generated inside centrifugal separator with 0.49, 0.52 and 0.6 million of grid points.

A three-dimensional Reynold's Averaged Navier-Stokes equations are solved using fully implicit finite volume scheme. For turbulent flow in separator, key success of CFD lies in a accurate description of turbulent behaviour. Two-equation k-epsilon RNG turbulence model is used to simulate swirling turbulent flow. Standard wall function is selected. Second order upwind scheme has been used to interpolate the conservative variables on the surface of control volume.

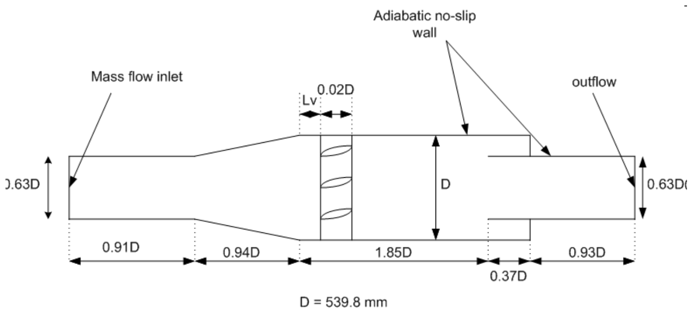
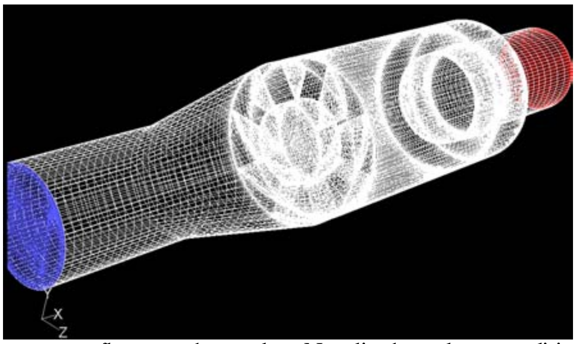


Fig. 1 Computational domain and grid for the present centrifugal separator

At inlet, mass flow inlet boundary condition is specified,

* Researcher ANU, surya@anuis.andong.ac.kr
* Professor ANU, kimhd@andong..ac.kr
** FMTRC, Daejoo Co



whereas outflow at the outlet. No-slip boundary condition is specified at all the wall boundary of separator.

3 Results and discussions

Numerical simulation is carried out with pressure 835 kPa, temperature 279.15 K and mass flow rate at 58.304 kg/s. Grid independent study has been carried out with three different grids as mention earlier. Pressure drop and velocity distribution suggest to select a grid size of 0.42 million grid points. Four types of grid has been generated for different swirl vane positions. Figure 2 shows the tangential velocity distribution along the separator.

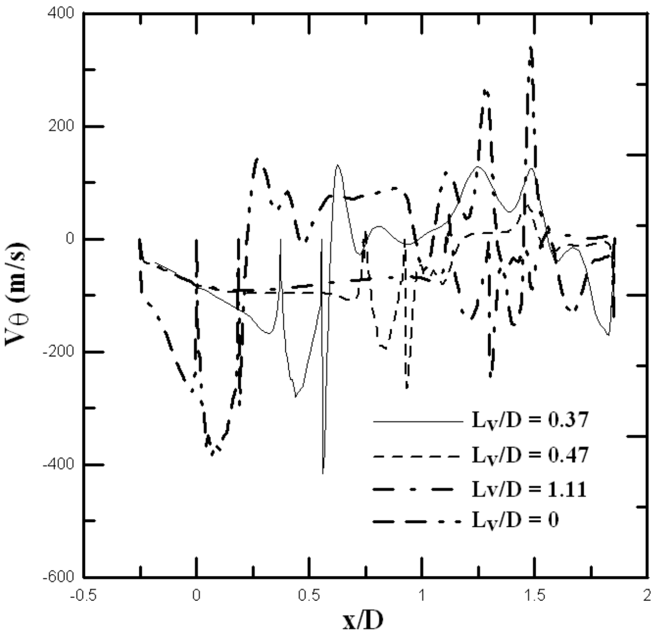


Fig. 2 Tangential velocity distribution along the separator at y/D = 0.9 and z = 0

Fig. 2 shows that there is a negative velocity just after the swirl vane in all the position. This is the region of inward flow. As the vane position is moving toward outlet pipe, magnitude of inward flow is reducing as seen in Fig. 2. Maximum outward tangential velocity is predicted by the vane position at Lv/D equal to 1.11. We have observed that tangential velocity decrease slightly as we move further (Fig. 3). The velocity flow profile show that at the center or close to axis, velocity is very low. Fig. 3 clearly shows the inward flow after the vane. Very low velocity is observed at the core of separator. Figure 4 shows the tangential velocity vector along the length of separator. This shows the very complex flow inside the separator. There are many vortices observed in flowfield.

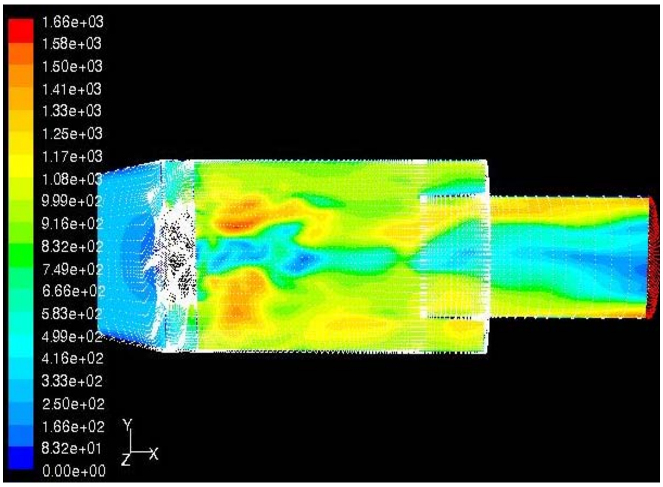


Fig. 3 Velocity contours along the separator at surface z = 0

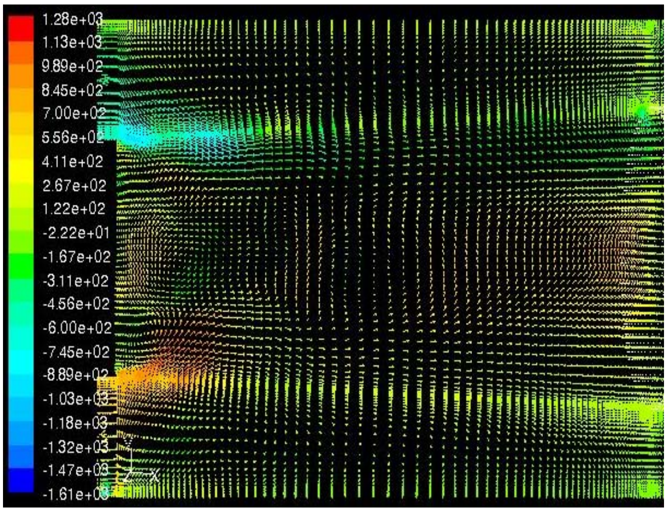


Fig. 4 Tangential velocity vector along the separator at surface z = 0

4 Conclusions

The 3-D Navier-Stokes equations are numerically solved using a fully implicit finite volume scheme. Based upon the obtained solutions, tangential velocities, centrifugal forces, vortices and total pressure losses are analyzed to find the best design parameters. The observed results have shown that the separation space (non-dimensional distance) should be in the order of 3 for the best separation efficiency.