CFD simulation formation of air-core in hydrocyclone

Farhadinasab, F *
Department of Chemical Engineering, Amirkabir University of Technology, Tehran, Iran (email: Farhadinasab@aut.ac.ir)

Dabir, B
Department of Chemical Engineering, Amirkabir University of Technology, Tehran, Iran (email: drbdabir@aut.ac.ir)

ABSTRACT

Hydrocyclone is a key unit operation in mineral processing industry. Hydrocyclones are designed to separate at a particular particle size where particles much larger than the cut size migrate rapidly to the wall and move downward and out into the underflow. Hydrocyclone simulation using Computational Fluid Dynamics (CFD) techniques is gaining popularity in process design. Air-core dimension is key to predicting the mass split between the underflow and overflow. FLUENT^{TM} 6.3 code was used to compute the dynamics of the flow. In this work influence of turbulence model of Reynolds Stress Model (RSM) and Large Eddy simulation (LES) on diameter of air-core was studied. In this way, modeling of hydrocyclone with air-core using multiphase VOF model was done. In the present simulation, structured meshed has been used to mesh generation of the geometry of hydrocyclone. To overcome meshing obstacle, geometry was divided to sub-domains. The simulation results are verified by comparing predicted and measured air-core in the hydrocyclone. Obtained results were in good agreement with literature data and showed that LES model predicted air-core more accurate than RSM model.

Keywords: CFD, Hydrocyclone, Air-core, Fluent, VOF model, RSM, LES.

INTRODUCTI ON

The first time hydrocyclone was used after Second World War by Dutch State Mines Company. Hydrocyclone is simple device with various applications across a range of industries, mineral, chemical and pharmaceutical due to several advantages, such as ease of operation, high capacity, less maintenance, less space requirement etc.

The flow behavior in hydrocyclones is quite complex, and this has led designers to rely on empirical equations for predicting performance. Computational fluid dynamics (CFD) provides a means of predicting velocity profiles under a wide range of design and operating conditions. The numerical treatment of the Navier–Stokes equations, the basis of any CFD technique, crept into analysis of the cyclone in the early 1980s.

Generally, two ways of air input can be distinguished. In dilute flow separation air is sucked through the spigot to the centre of the hydrocyclone. However, also feed suspension always contains a certain amount of air in a dispersed or diluted form. So, the effect of both amount of air causes a continuous air flow from underflow to the overflow. The formation of air-core is illustrated in Fig. 1.

The geometrical configuration of the hydrocyclone analyzed in this study is presented in Fig. 2, and is identical to the hydrocyclone used in the Hsieh's study.

Literature review

The CFD approach is so complex that specifying the form and location of the air/water interface becomes a very difficult problem. Numerical modeling of the air-core for hydrocyclones has been dealt with in a number of ways. The usual simplification is that the air-core is modeled as a fixed cylindrical surface with a slip condition with the help of theoretical models in order to estimate the air-core diameter [2-7]. And another way is CFD approach for predicating air-core diameter and shape.

The first CFD studies on hydrocyclone was done Hsieh [1], which used a mixing length eddy viscosity turbulence model and used validating result from his data that Hsieh investigated hydrocyclones, with further publications by co-workers from the University of Utah [8-11]. Hsieh measured the velocities and turbulence parameters in a 75mm glass cyclone with a water feed and operating with an air core using laser Doppler anemometry (LDA) [1].