Penerbit Akademia Baru

Journal of Advanced Research in Applied Mechanics
ISSN (online): 2289-7895 | Vol. 5, No.1. Pages 8-14, 2015

Numerical Simulation of Splitting Devices in Horizontal Pipeline

N. Mat Isa[*,a], *A. F. Ab Rahman*[b] *and A. Sadikin*[c]

Center for Energy and Industrial Environment Studies (CEIES)
Faculty of Mechanical Engineering and Manufacturing
Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia
[a,*]*sikin@uthm.edu.my*, [b]*ahmadfattah.afai@gmail.com*, [c]*azmah@uthm.edu.my*

Abstract – *This research was carried out to determine the suitable splitting device for horizontal pipeline in a pulverised coal power plant. In this research, six splitting devices have been designed and only one device achieves the main objective which is to improve fluid flow to be a homogeneous flow in the pipe. The procedure of choosing the best device is from the velocity profile of the fluid after moving through the splitting device. From the data, 'standard k-epsilon' is the best method for the simulation of this research. The geometry of the device in this research is followed as the geometry used by another researcher for analysis through experimental approach. All analysis in this research is using software ANSYS 15.0.*

Keywords: Splitting device, CFD, homogeneous.

1.0 INTRODUCTION

Nowadays, there are many designs of splitting devices had been developed. A splitting device is used to split the substance from one input or source to the several output. In an industrial field, the splitting device in pneumatic concept line is already applied. This device is also known as a pneumatic conveying splitting device.

Generally the problems occur in the conveying unit is the consistency of the flow and the flow become roped flow [1]. It might because of some reason, such bending effect, interaction between particles and fluid, effects of walls in conveying systems and others[2]. These factors must be avoided in order to improve the pneumatic conveying system [3]. A new design of splitting devices of pneumatic conveying in horizontal pipeline will purpose to overcome the roping flow.

1.1 Power Plant

Electricity is one of the important sources in our daily life. It lights houses, buildings, streets, provides domestic and industrial heat, and powers most equipment used in homes, offices and machinery in factories. Without electricity, people will be difficult to do their routine. The World Energy Outlook 2012 and Electricity Information 2012 from the OECD's International Energy Agency (IEA) present that from 2000 to 2010 total world primary energy demand grew by 26%. The electricity demand is almost increasing. Starting from year 2000, the electricity consumption is drastically increasing at the most regions (World Nuclear Association, 2013).

1.2 Coal Power Plant

Coals are a combustible, sedimentary, organic rock, which is composed mainly of carbon, hydrogen and oxygen. The rank of coal depends on volatile matter, fixed carbon, inherent moisture, and oxygen, although no one parameter defines rank. Typically, coal rank increases as the amount of volatile matter decreases and the amount of fixed carbon increases.

The coal-fired power plants fall into five categories which are stoker-fires, pulverized coal, cyclone-fired, fluidized-bed combustion and coal gasification. In the United States, pulverized coal power plant is dominantly used for electric energy generation [4]. This research is focusing on pulverized coal power plant.

Pulverized-coal combustion units are predominantly used in coal-based power plant. For example, in the United Stated, roughly 92% of the total coal consumed by the power plant is burned on this combustion system [4].

1.3 Splitting Device

The splitting device is used to ensure that the flow of powdered coal in pneumatic conveying system is homogenous. In pneumatic conveying the roped flow may occur along the pipeline. It needs to be avoided in order to increase the conveying system efficiency.

The splitting device is used to break the roped flow, so that that flow can be homogenous [5,6]. By using a pneumatic concept, the powdered coal will flow through it and the flow that in roping condition will be separated by the roped splitter.

1.4 Computational Fluid Dynamic (CFD)

The term CFD generally refers to the analysis of systems using fluid flow, heat transfer and associated phenomena such as chemical reactions. It is also known as computer based simulation. By using CFD, a computational model that represents a system or device to be studied can be developed. Then, the fluid flow has to be applied to the virtual prototype. Finally, CFD software will show the result a prediction of the fluid dynamics [7].

2.0 METHODOLOGY

As a simulation based research, all of the complete system and the component need to be designed or modelled. The design process had been carried out by using the software of “SolidWorks 2013”. All design is saved as an.IGS file. This type of file then will be imported into “Design Modelling” in the software of “ANSYS 15.0” for analysis process using CFD.

Six devices are designed for this research. The materials for these devices are made from perspex and the dimensions are followed as the size of the device that carried through experimental approach. These devices are shown in figure 1.

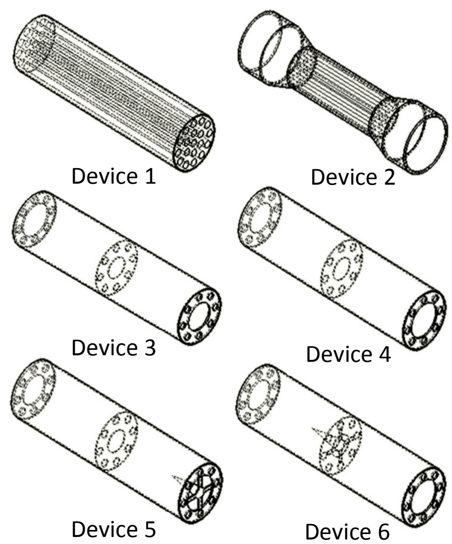


Figure 1: View of Splitting Devices in Wireframe

The analysis for this research is done by using a simulation approach which through the CFD. The parameter that related to the research will be set in the software. The method in CFD is divided into three stages which are pre-processing, solver and post processing.

2.1 Pre-Processing

The first step for this stage is inserted geometry. The model will be drawn in the Design Modeller (DM) window, followed the geometry of fluid that want to be analysed. Then, the model will be meshed as shown in figure 2. The numbers of cells will influence the result and the time taken for analysis process. Then, all face is required to give name, so that the boundary condition can be set up. The last step in this stage is set up. Model that being selected for this research are k-ε Std and k-ω.

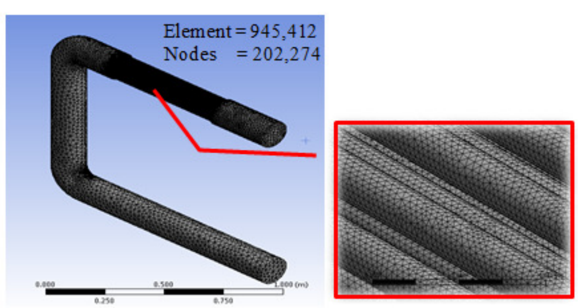


Figure 2: Meshing

2.2 Solver

The solution method used for this research is SIMPLE as shown in figure 3. The solution for all devices is set about 1000 iterations.

Journal of Advanced Research in Applied Mechanics
ISSN (online): 2289-7895 | Vol. 5, No.1. Pages 8-14, 2015

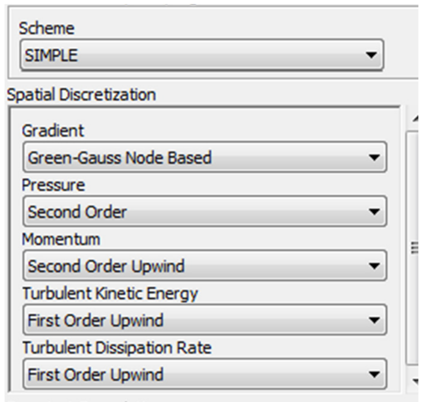


Figure 3: Solution Method

2.3 Post-Processing

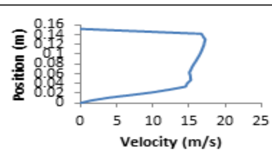
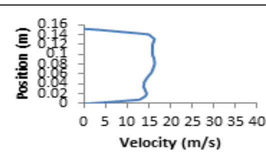
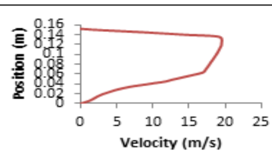
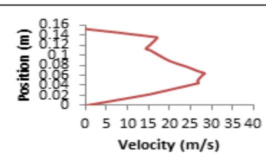
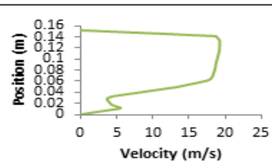
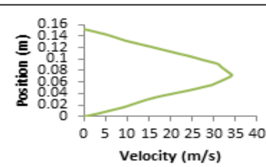
In this stage, the streamline, contour and vector can be created to see the behaviour flow at any point.

3.0 RESULTS AND DISCUSSION

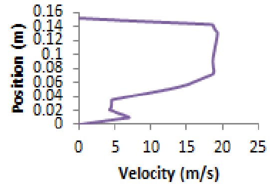
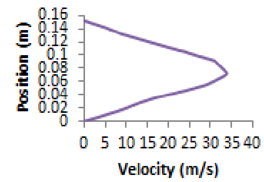
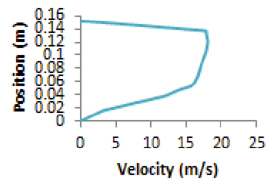
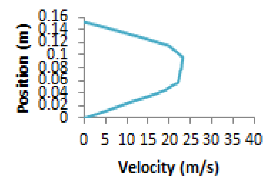
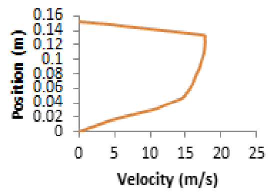
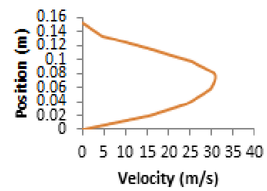
3.1 Graph Analysis

One of the ways to study about velocity profile in the conveying system is by using graph analysis. The data for graph analysis are taken at before and after the splitting device. The result of this analysis is used for identifying which device is the most suitable to be used. The data for this graph analysis are taken perpendicular to the flow direction of fluid as shown in table 1.

Table 1: Velocity Profile at Device Inlet and Outlet

	Velocity Profile at Device Inlet	Velocity Profile at Device Outlet
Device 1		
Device 2		
Device 3		

Journal of Advanced Research in Applied Mechanics
ISSN (online): 2289-7895 | Vol. 5, No.1. Pages 8-14, 2015

Device 4		
Device 5		
Device 6		

The data of the analysis are plotted for position against velocity. The main location to be analysis is at a flow after splitting devices. All of the graph will compile into one as figure 4. Each of the splitting devices will result different velocity profile which depends on its design and concept.

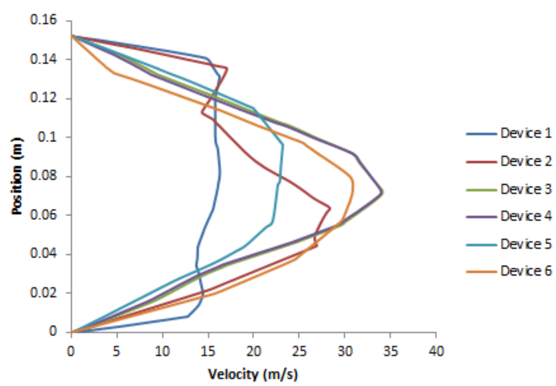


Figure 4: Graph of Position against Velocity for Different Device at Outlet Reading Location

The second date is taken parallel to the flow direction which is located at after splitting devices as shown in figure 5. The data for this graph is to show the velocity profile along 10cm after entering splitting devices. From the overall result, the best design is the device 1, which is homogeneous and remains constant as inlet velocity.

Penerbit Akademia Baru
Journal of Advanced Research in Applied Mechanics
ISSN (online): 2289-7895 | Vol. 5, No.1. Pages 8-14, 2015

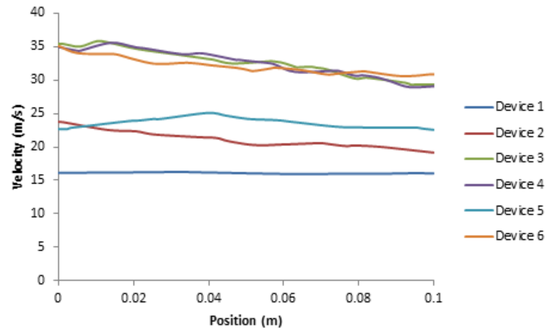


Figure 5: Graph of Velocity against Position for Different Device Parallel with Fluid Flow

3.2 Validation with Experiment

The devices that involved in validation with the experiment are device 3 and 4. The validation is carried out to identify the effectiveness CFD in analysis about splitting device compared to the experiment. The analysis using an experimental approach had been carried out by another researcher. The validation is done between data from experiments, using k-ε std and using k-ω method as shown in table 2. From the percentages differences, k-ε std is the lowest.

Table 2: Percentage Difference for Validation Data

		k-ω	k-ε	Experiment
Device 3	Average Velocity (m/s)	5.30	4.97	4.34
	Percentage Different (%)	**22.34**	**14.71**	**0**
Device 4	Average Velocity (m/s)	4.48	4.22	4.30
	Percentage Different (%)	**4.16**	**1.77**	**0**

3.3 Turbulent Flow Verification

The turbulent model is chosen for this research. That is because the flow in the conveying system is in turbulent flow. To strengthen this statement, Reynolds number needs to be calculated by using the formulae:

$$Re = \frac{\rho v D}{\mu}$$

Where,

$$\rho_{air} = 1.225\ kg/m^3,\ \mu_{air\ @\ 27.5°C} = 1.86\ x\ 10^{-5} kg/ms,\ D = 6inch\ @\ 0.1524m$$

Therefore, Reynolds number at the inlet:

$v = 15m/s$

$Re = \frac{(1.225)(15)(0.1524)}{1.86 \, x \, 10^{-5}}$ $= \underline{\underline{150{,}556.45}}$ (turbulent flow)

4.0 CONCLUSSION

From the research that had been done for the effectiveness of the splitting device for conveying system in order to changing roping flow to the homogenous, the result shows that different splitting device and splitting concept may result different velocity profile. The simulation approach can be a good method for analysis in order to reduce the analysis cost. Besides that, sometimes the result might be a too different with a real data. It may cause by surrounding or the source of error that can exist in reality. From the six splitting device that's been designed in this research, the best splitting device is device 1. The velocity profile for device 1 has shown a constant homogenous. In other word, this device had approached the research purpose.

REFERENCES

[1] V. Lijo, H. D. Kim, T. Setoguchi, Analysis of Choked Viscous Flows through a Constant Area Duct, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 224 (2010) 1151-1162.

[2] Dr. H. Schneider, Dr. Th. Frank, DI K. Pachler, Dr. K. Bernard, A Numerical Study of the Gas-Particle Flow in Pipework and Flow Splitting Devices of Coal-Fired Power Plant, 10th Workshop on Two-Phase Prediction, Martin-Luther Universitat Halle-Wittenberg, Germany (2002).

[3] David Mills, Pneumatic Conveying Design Guide, Second Edition. Elsevier (2004) Butterworth-Heinemann, UK.

[4] Nicholas P. Cheremisinoff, Clean Electricity Through Advanced Coal Technologies, 1st Edition Handbook of Pollution Prevention and Cleaner Production (2012).

[5] N. Mat Isa, I. Malek, S. Sadikin.S.M. Basharie.N. Asmuin, Experimental study of splitting device for horizontal pipeline, Applied Mechanics and Materials 660 (2014) 714-718.

[6] J. T. Robert, Splitting Techniques in Vertical Pneumatic Conveying, University of Nottingham, UK, Degree of Doctor Philosophy's Thesis (2008).

[7] Chandana Ratnayake, A Comprehensive Scaling Up Technique for Pneumatic Transport Systems, The Norwegian University of Sceince and Technology, Norway Degree of Doctor Philosophy's Thesis (2005).