Computational Fluid Dynamics Analysis of External Recirculation Flow at Updraft Gasifier Using Ejector

Fajri Vidiana[a], Adi Surjosatyo[b] and Yulianto Sulistyo Nugroho[c]

[abc]*Department of Mechanical Engineering, Faculty of Engineering, University of Indonesia, Indonesia*
Email: adi.surjosatyo@ui.ac.id

Abstract. Gasification process at updraft gasifier produces the amount of more tar than the other gasifier type. To reduce tar at updraft gasifier, it is carried out recirculation of pirolysis gas into combustion zone. Ejector is an equipment used to entrain the secondary fluid flow by moving momentum and energy from high velocity of primary flow (jet). Research carries out the simulation of isothermal 3D using Computational Fluid Dynamics (CFD) to obtain the maximum of recirculation flow at updraft gasifier using ejector. The result of simulation shows that change of nozzle diameter gives great influence at change of total flow for resirculation compared with the change of diameter and length of mixing area and nozzle exit position (NXP). The maximum of resirculation flow 81 lpm at nozzle diameter 0.75 cm.

Keywords : CFD, maximum flow, external recirculation, ejector, updraft gasifier
PACS: 88.20.mr

INTRODUCTION

Gasification Process at Updraft gasifier had much advantages for simple operation, high carbón convertion, higher efficiency, more stable continuity, wáter content of fuel > 60 %, flexible shapes of biomass size and easier in scale up compared by *downdraft gasifier* [1], but type updraft gasifier had high content of tar.

In an attempt reduces tar at updraft gasifier, we expanded recirculation of pirolisis gas into combustión area and gas release from reduction area at updraft gasifier. One of methods used to recirculate gas by using ejector. The application of ejector had many advantages such as: reliability, low operational cost, simple installation and friendly environment.

Constant Mixing Area (CMA) of ejector is an ejector with mixing area of constant primary and secondary flow. Constant mixing area of ejector can remove mass flow more than constant pressure mixing (CPM).

Convergent nozzle can produce total secondary flow greater than convergent-divergent nozzle. To obtain maximum recirculation flow, it is carried out simulation particularly using CFD.

Standard k-epsilon is considered the simplest complete model of turbulence in CFD simulation. This model is widely used in industrial flow simulation due to robustness, economy, and reasonable accuracy for wide range of turbulent flow.

Many researchers have been carried out to optimize the system of secondary flow at ejector using CFD by standard turbulence model of k–epsilon. Utomo.T [2] have carried out simulation using CFD Fluent to the influence of divergent area angle for ejector performance using standard k–epsilon. Riffat [3] carried out simulation to the influence of nozzle exit position for ejector performance using standard and RNG K-epsilon T. Sriveerakul [4] carried out simulation using CFD to find out the flow phenomenon at steam ejector. K Pianthong [5] carried out simulation using CFD to find out the flow phenomenon and performance of constan pressure area ejector and constan mixing are of ejector. The result of simulation showed that CFD could predict ejector performance and describe the flow structure at ejector.

This research purposed to find out the flow phenomena and the maximum flow at updraft gasifier using ejector through the variation of nozzle exit position, diameter of ejector, diameter of mixing area and length of mixing area.

The 4th International Meeting of Advances in Thermofluids (IMAT 2011)
AIP Conf. Proc. 1440, 936-941 (2012); doi: 10.1063/1.4704307

RESEARCH METHODOLOGY

Prior to simulation, it makes 3D models of solid reactor of updraft gasifier. Working principle of simulated reactor can be described as follows. Main air of combustion is air 1 that enters from the down part of reactor. Air 2 (primary air) enters from ejector used to entrain recirculation air through recirculation pipe. Air exits from middle side part of reactor as shown in Figure 1.

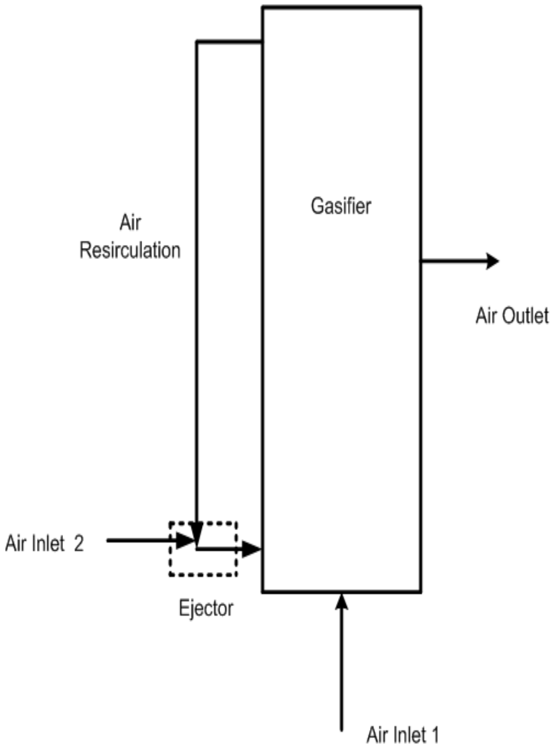

FIGURE 1. Recirculation Model.

Ejector used is an ejector with the type of constant mixing area and type of nozzle used is a convergent type. Ejector model used is shown in Figure 2 by dimension as in Table 1.

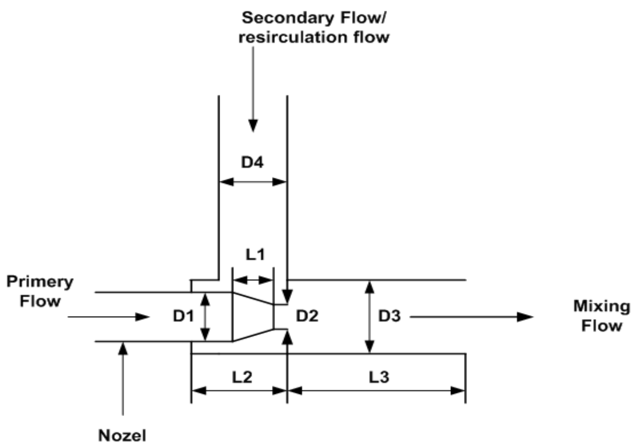

FIGURE 2. Ejector Model.

TABLE (1). Dimension of Ejector.

Dimension	lenght (cm)	Dimensión	Diameter (cm)
L1	4	D1	2.5
L2	7.5	D2	1.5
L3	7.5	D3	5
		D4	5

Reactor model from the next solid is carried out by *meshing*. Mesh Type used is tetrahybrid. Simulation is carried out using the equations of mass, momentum, energy conservation, Turbulence of Standard k-epsilon.

The boundary conditions used are as follows:
Air inlet 1 is a velocity inlet 1.2 m/s
Air inlet 2 is a velocity inlet 0.6 m/s
Air outlet is a pressure outlet with atmosphere pressure, 0 Pascal
Simulation is carried out by varying some variations to obtain optimal condition and velocity is maintained constantly.

1. Simulation of condition without any recirculation pipe.
2. Simulation using recirculation pipe without any ejectors.
3. Simulation using ejector
    a. Variation of Nozzle Exit Position/NXP: Nozzle exit position at positive and negative direction as shown in Figure 3 .

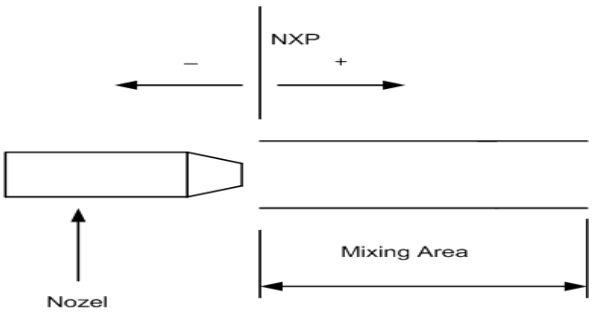

FIGURE 3. The Change of Nozzle Exit Position.

    b. Variation for the length of mixing area
    c. Variation of mixing area diameter
    d. Variation of nozzle diameter

RESULTS AND DISCUSSIONS

Figure 4 shows velocity contour in reactor without using external recirculation pipe and ejector. It is shown that air flow rate from air outlet at middle of reactor and at top of reactor respectively 50% about ± 80 lpm. Then it can be conclude that total flow is recirculated about ± 80 lpm.

Figure 5 shows the velocity contour obtained from simulation using external recirculation pipe without using ejector of total flow circulating about 0.35 lpm. Almost no flow recirculating occurring. Figure 6 shows velocity contour in reactor using external recirculation pipe and ejector where there is greater velocity countour in recirculation pipe line than without without ejectors. Its caused primary jet velocity exit ejector make lower pressure around of nozzle exit than secunder flow escape in resirculation pipe entrained by primery jet velocity.

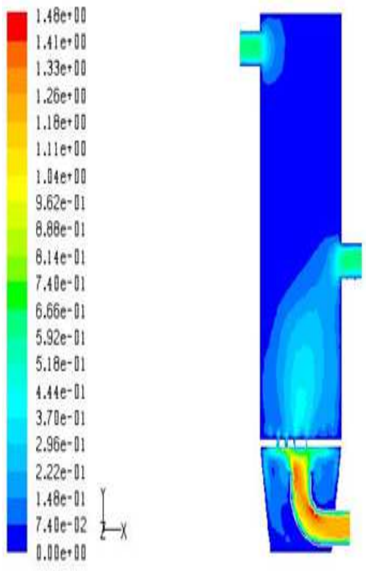

FIGURE 4. Velocity Contour in Reactor Without Recirculation Pipe and Ejectors.

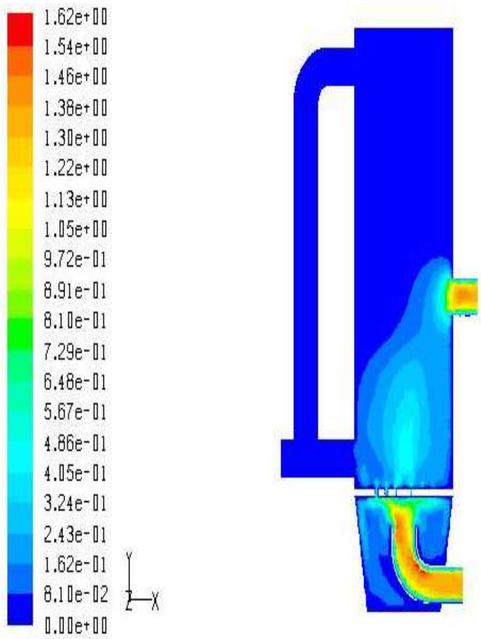

FIGURE 5. Velocity Contour in Reactor with Recirculation Pipe without Ejectors.

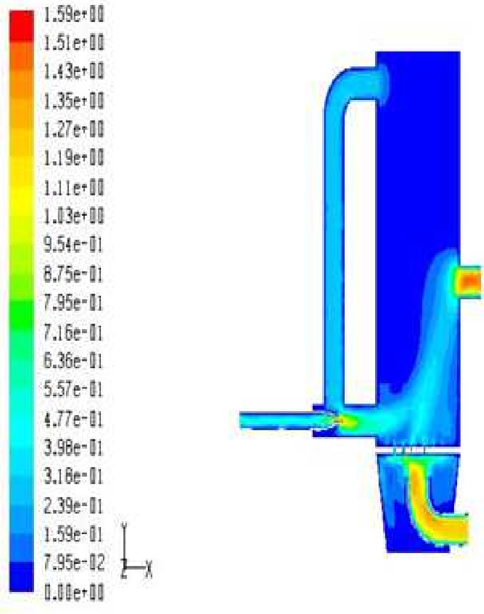

FIGURE 6. Velocity Contour in Reactor with Recirculation Pipe and Ejectors.

Figure 7 shows the greater total flow of recirculation if NXP is moved to negative direction for each condition. But going too far to negative direction will cause greater momentum of primary flow and total flow of recirculation becomes low, then there is an optimum point at negative direction.

Figure 7 also shows variation influence for the length of mixing area, diameter of mixing area and nozzle diameter toward total flow of recirculation produced. Figure 8 shows that the change of nozzle diameter (D_2) from 1.5 cm to 0.75 cm by any other dimension maintained constantly, giving great significant influence to total flow of recirculation produced compared by other variation. Where there is total flow of recirculation between 60 to 81 lpm occurring the increase around 100% . This is caused by downsizing nozzle diameter make the increase of primary velocity for nozzle exit from ± 1 m/s to ± 5 m/s . Then it more increases momentum produced by primary flow to entrain secondary flow. The increasing average velocity of secondary flow from ± 0.2 m/s to ± 0.6 m/s as shown in Figure 8.

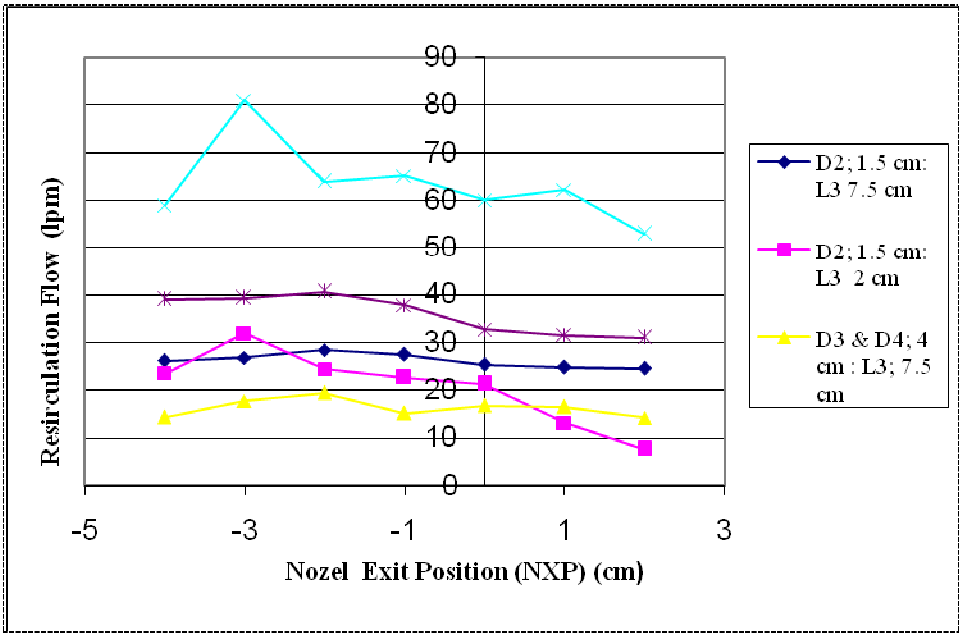

FIGURE 7. Variation of Ejector Vs Total Flow of Recirculation.

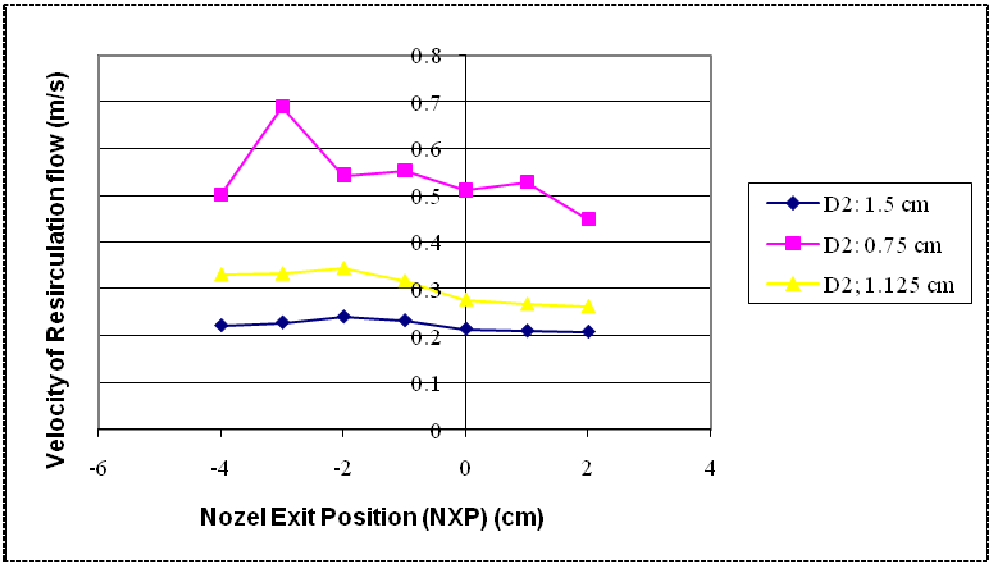

FIGURE 8. Change Influence of Nozzle Diameter (D2) toward Velocity Recirculation.

CONCLUSION

The result of simulation having maximum total flow can be recirculated 81 lpm at nozzle diameter 0.75 cm. Total flow 81 lpm enables to use only one ejector and to entrain recirculation air 80 plm. The change of nozzle diameter gives great influence at change of total flow for recirculation compared by the change of: diameter and the length of mixing area, and the nozzle exit position (NXP).

ACKNOWLEDGMENTS

The outhors would like to thank the Ministry of Education of Indonesia for funding this research through Hibah Penelitian Disertasi Doktor Tahun 2010 with contract 2314/H2.R12.3/PPM.00/Penelitian/2010.

REFERECENCES

1. Bridgwater,AV.,*" Thermal Processing of Biomass for Fuels and Chemical"* , Paper,. 6th Asia- Pacific International Symposium on Combustion and Energy Utilization, Paper, May 2002.
2. Utomo. Tony, Ji.Myongkuk, Kim. Pilhwan, Jeong.Hyomin, Chung. Haushik, *CFD Analysis on the Influence of Converging Duct Angle on the Steam Ejector Performace*,2008.
3. Riffat SB, Omer SA, CFD Modelling and Eksprimental Investigation of an Ejector Refrigeration Using Methanol As Working Fluid, Int J Eng Res,25,2001, pp 115 -128.
4. Sriveerakul.T, Aphornratama. S, Chunnanond. K, Performance Prediction of Steam Ejector Using Computational Fluid Dynamic. International Journal of Thermal Science (4),2007, pp 823–833.
5. Pianthong . K, Seehanam. W, Behnia. M, Sriveerakul. T, *Investigation and Inprovment of Ejector Refrigeration System Using Computational Fluid Dynamics Tecnique",* International Journal of Energy Conversion Management, Vol 48, Issue 9,2006, pp 2556 – 2564.