VOL. 9, NO. 9, SEPTEMBER 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

CHARACTERISTIC OF VORTEX IN A MIXING LAYER FORMED AT NOZZLE PITZDAILY USING OPENFOAM

Suheni and Syamsuri
Department of Mechanical Engineering, Adhi Tama Institute of Technology Surabaya, Indonesia
E-Mail: irsuheni@gmail.com

ABSTRACT

PitzDaily nozzle was the most substantial component in a gas turbine. This nozzle was used to mix air with propane. In the application value of turbulence and vortex center was very important for this type of nozzle. Reynolds number was a parameter used to see its effect on the value of turbulence and vortex center. The method applied was a numerical simulation by using OpenFOAM. This simulation was performed to determine the distribution of pressure, streamline, turbulence, and vortex center. The study was conducted by varying Reynolds numbers 12210, 50000, and 100000. Grid independent test was made to validate with the results of previous research. By this simulation results indicated that this method was feasible and the solver was highly accurate. The results showed that the higher the value of Reynolds number, the further away of vortex center rear nozzle. The mark of turbulence and vortex length were also increased. In addition to that the larger the value of this then the mixture of air and propane formed a fine grained, so it became more perfect combustion.

Keywords: nozzle pitzdaily, openFOAM, variation of reynolds number.

INTRODUCTION

Currently, the developing countries in the world, a lot of technologies make growing and the modern. One of which is the technology widely used for research on the internal and external flow of fluid flow in the nozzle flow such as a daily case. Nozzle Pitz is a tool to expand the fluid so that the speed increases. Nozzle daily Pitz case is one of the important components of the gas turbine, wherein the nozzle is used to mix the air and propane [1]. The potential energy is converted into kinetic energy through multiplication of mass flow rate of combustion gases with the incoming flow velocity changes and the exit of the nozzle produces thrust.

The flow in the nozzle is usually accelerated ranging from inlet to outlet. Gas flow rate in the throat at the range of one mach. With the no-slip condition, then there is a difference in speed between the inner surface of the nozzle with a speed around the axis of the nozzle. The velocity difference generates the flow velocity profile around the surface of the wall.

Object of this study is the daily Pitz nozzle case because it is one of the convergent nozzles shaped (cone). Flow condition in the nozzle is simulated by using Computational Fluid Dynamics (CFD). Simulation results are in the form of contour flow parameters, both within and behind a nozzle. Additionally, simulation results can also be expressed in the quantitative form.

In its application of these studies can be performed using a numerical visualization software OpenFOAM 2.1.1. There are several researchers that simulate fluid flow using OpenFOAM [[2], [3], [4], [5], [6] and [7]]. PitzDaily nozzle case study using OpenFOAM's also been done by [8] and [1] with a different discussion with this study. This study focused on the characteristics of the vortex with Reynolds number variation. With the use of OpenFOAM software may release information about the flow phenomena that cross daily Pitz nozzle case. The simulation results further reprocessed to obtain the velocity profile around the internal surface of the nozzle, the pressure distribution on the outer, inner wall and the radial section.

The utilized of OpenFOAM for nozzle flows in the daily Pitz using varying Reynolds number. Reynolds number can affect the flow across these models. In this problem, we are trying to analyze the influence of the Reynolds number of the flow in the model.

THEORY

Mathematic model of flow

Laminar flow is a fluid that has a certain viscosity, and it is an incompressible flow can be described by the Navier-Stokes equations in an Eulerian reference frame [9],

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j}\right) = \rho F_i + \frac{\partial \sigma_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial u_i}{\partial x_j} = 0 \tag{2}$$

where: u_i is a velocity component

ρ is a density

F_i is a gravity component

$\partial\sigma_{ij}$ is a stress tensor

While the equation for stress tensor can be seen in the following equation:

$$\sigma_{ij} = -p\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

where: μ is a dynamics viscosity

p is a pressure

δ_{ij} is a cronecker delta

The k-ε model for turbulent case

To model the turbulent flow in numeric simulations typically use a variety of methods, among others Reynolds stress, large eddy simulation, mixing length models, Reynolds average model of the Navier-Stokes and k-ε models. The k-ε model is widely used because it has several advantages, among others: to have stability in the numerical simulations, the results are more accurate, and more efficient. Standard models for the equation to the model shown in the following equation:

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\upsilon_T}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}\left[\upsilon_T\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}k\delta_{ij}\right]$$

$$\frac{\partial \bar{u}_i}{\partial x_j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{4}$$

Due to the flow is incompressible flow calculations performed on the average flow area. The equation can be simplified to

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\upsilon_T}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}\upsilon_T\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)$$

$$\frac{\partial \bar{u}_i}{\partial x_j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{5}$$

The form of the diffusion equation ε has the same advantages as the form of the equation k by means of analogy. For turbulent viscosity can be written as,

$$\upsilon_T = C_\mu k^2/\varepsilon \tag{6}$$

where C_μ is a constant model.

Equations of k and ε are together with the specification of υT. This model became complete because it does not require special specifications such as turbulent length scale in the other models.

The k-ε model consists of four components; two equation model is solved for k and ε. Turbulent viscosity is defined by $\upsilon_T = C_\mu k^2/\varepsilon$. Reynolds stress is found from the turbulent viscosity hypothesis and the equation solved for Reynolds.

Standard value for the constants k-ε model for the turbulent equations used in the model is [10].

$$C_\mu = 0.09, \quad C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.92, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3$$

NUMERICAL METHOD

For the case of nozzles Pitz-Daily

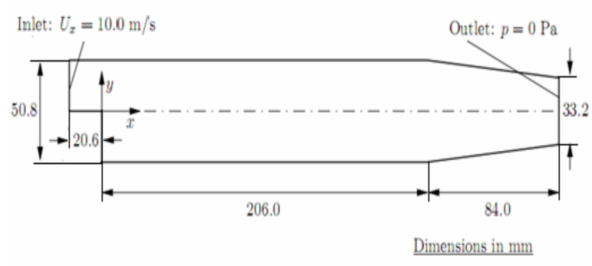


Figure-1. The geometry of the rear surface.

Initial Condition

U = 0 m/s, p = 0 Pa

Boundary Condition

Inlet (left) for constant velocity U = (10, 0, 0) m/s, Outlet (right) for constant pressure p = 0 Pa

No slip at the boundary

Transport properties

Kinematic viscosity for air $\upsilon = \mu / \rho = 18.1 \times 10^{-6} / 1.293 = 14.0\ \mu m^2/s$

Turbulent model

Standard κ - ε, coefficient: $C_\mu = 0.09$; $C_1 = 1.44$; $C_2 = 1.92$; $\alpha_\kappa = 1$; $\alpha_\epsilon = 0.76923$.

RESULTS AND DISCUSSIONS

Validation of the model

Tabe-1. Validation with other researcher [1].

Number of Cells	Center of Vortex	
	X	Y
815	0.067	-0.012
1840	0.0685	-0.013
3470	**0.072**	**-0.012**
Experiment [1]	**0.073**	**-0.011**

From Table-1 above shows that between this study with previous research results [1] there are similarities. The difference in the results is about 1.4 %.

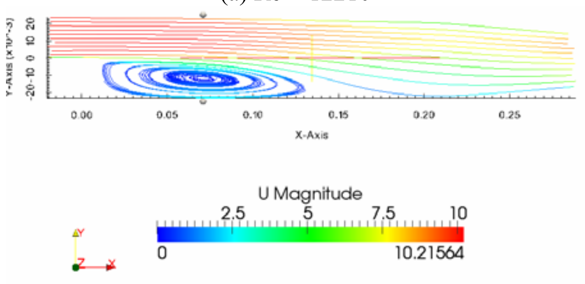
The comparison of streamline for Re is equal to 12210, 25400, 50000 and 100000

Here is the comparison of the simulation results for Re is equal to 12210, 25400, and 50000 and 100000.

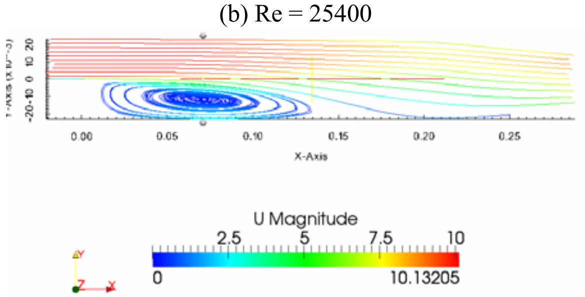
VOL. 9, NO. 9, SEPTEMBER 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

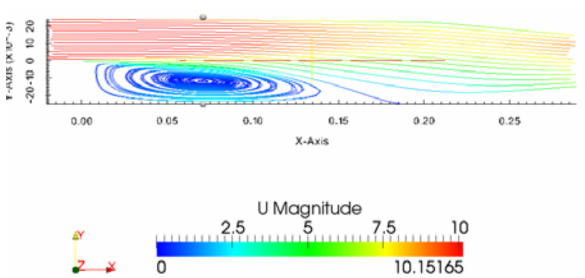
(a) Re = 12210



(b) Re = 25400



(c) Re = 50000



(d) 100000

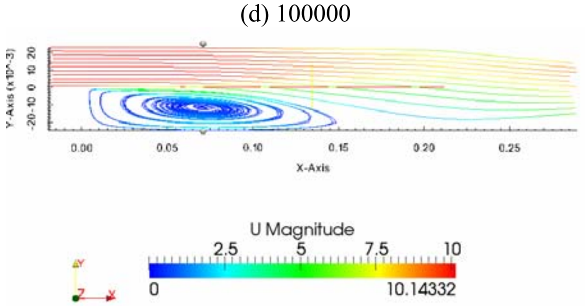


Figure-2. Display a streamline comparison for Re = 12210, 25400, 50000 and 100000.

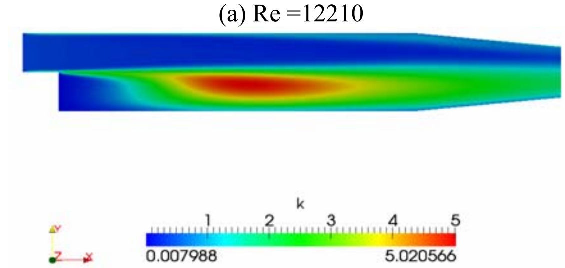
From Figure-2, the streamline comparison of Re = 12210, 25400, 50000 and 100000 a trace of the flow by an eddy is formed. However, in other streamline results show different with the variation of Reynolds number. The difference lies in the center of the eddy generated in the simulation. By this Figure, the differences in the centre of eddy are shown in Table-2 below.

Tabe-2. The comparison result of center of vortex.

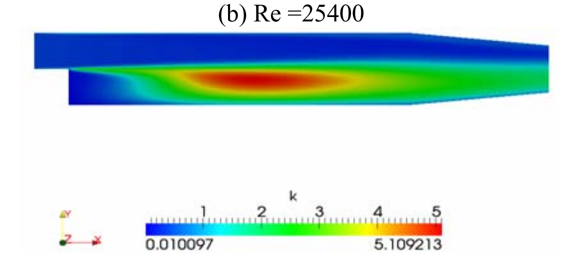
Reynold Number	Center of Vortex	
	X	Y
25400	0,072	-0,012
12210	0.07	-0,011
50000	0,0734	-0,0123
100000	0,0739	-0,013

The comparison of turbulence distribution for Re = 12210, 25400, 50000, and 100000

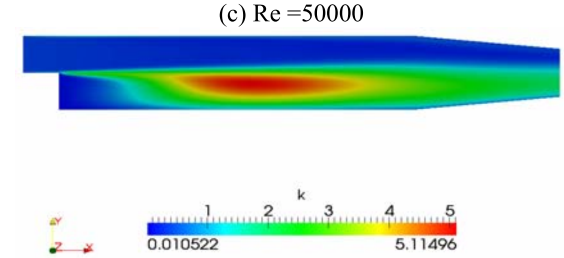
(a) Re =12210



(b) Re =25400



(c) Re =50000



(d) Re =100000

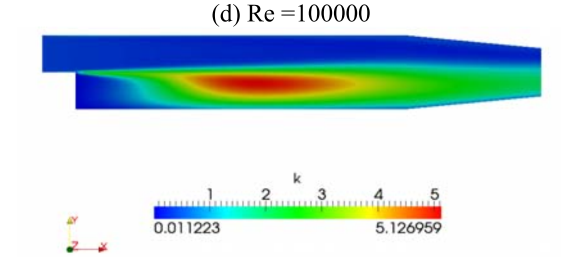


Figure-3. The comparison of turbulence distribution for Re = 12210, 25400, 50000, dan 100000.

Figure-3 is the comparison of the distribution of the turbulence for Re = 12210, 25400, 50000, and 100000 which have the area of high turbulence. However, in any distribution of turbulence the result shows the different results for the variation of Reynolds number. The difference lies in the value of the high turbulence area marked with the colour of red. For Re 25400 the turbulence value is equal to 5.105213, Re 12210 the turbulence value 5.020566, the value of turbulence 5.11496 at Re 50000, and the value of turbulence 5.126959 at Re 100000. It can be concluded that the higher the value of the Reynolds number, the larger the value of the resulting turbulence.

Tabe-3. The comparison result for the length of maximum turbulence region.

Reynolds number	The length of area of max. turbulence
12210	0,102
25400	0,103
50000	0,105
100000	0,108

The comparison of velocity distribution at position X = 0.0708653 for Re = 12210, 25400, 50000, and 100000

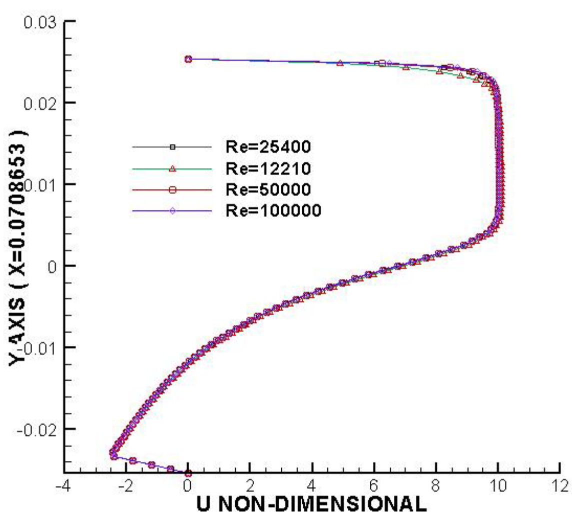


Figure-4. Display of data plot for velocity distribution along the line Y at position x = 0.0708653 pitzDaily case with Re = 12210, 25400, 50000, and 100000

From the graph for the velocity distribution with the variation of Re = 12210, 25400, 50000, and 100000 obtain the velocity distribution with the same trend. This section shows the velocity distribution along the Y at position X = 0.0708653. On the other rate distribution, results can be seen the difference. At position y = -0.02 to y = 0, the simulation shows that with the small Reynold number, the trend of speed is higher than with high Reynolds number. While At position y = 0 to y = 0.02, the simulation shows that with a small Reynolds number, the trend of speed tends to be smaller than with the high Reynold number. The greater the Reynolds number then the trend will flow nearer to the wall. This is because the lower the viscosity value so that the smaller the shear stress.

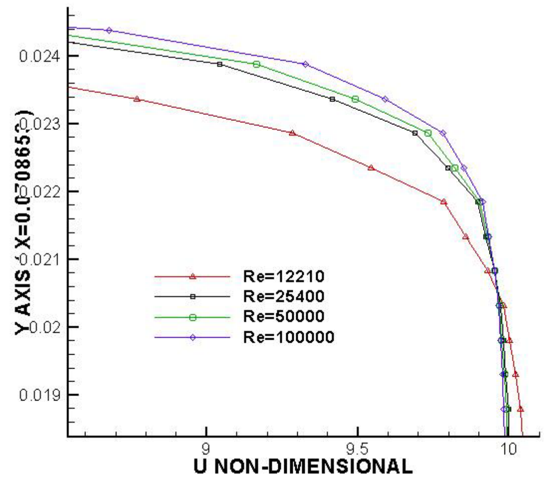


Figure-5. The comparison of velocity in the position of X=0.072.

From Figure-5 shows that the maximum velocity occurs when the Reynolds number equal to 100000.

CONCLUSIONS

This study can be concluded as follows:

a) Validation results obtained from the experiment and the results of other studies show little difference from the location of the center of the vortex is generated in the experiment at X = 0.073 and Y = -0.011 while the point of the research is on point and the point X = 0.072 Y = - 0.012. It is proved that the simulation's methods are capable of simulate this case.
b) From the results of experiment with the higher value of the Reynolds number, the more remote area of eddy center occurs in the rear nozzle.
c) In this study with the variation of Reynolds number, the higher the Reynolds number the greater the value of turbulence to be generated. With the growth in the value of the turbulent mixing process of air and propane will obtain granules finer, so it will be more perfect combustion.

REFERENCES

[1] T. Kobayashi. 2006. Large eddy simulation for engineering application. Fluid Dynamic Research. 36: 84-107. Science Direct.

[2] H.G. Weller, G. Tabor, H. Jasak and C. Fureby. 1998. A tensorial approach to computational continuum mechanics using object-oriented techniques. Computer in Physics. 12(6): 620-631.

[3] J. L. Favero, A. R. Secchi, N. S. C. Cardozo, H. Jasak. 2009. Viscoelastic flow simulation: development of a methodology of analysis using the software OpenFOAM and differential constitutive equations. 10th International Symposium on Process Systems Engineering.

[4] F. Habla, H. Marschall, O. Hinrichsen, L. Dietsche, H. Jasak, J. L. Favero. 2011. Numerical simulation of viscoelastic two-phase flows using openFOAM. Chemical Engineering Science. 66(22): 5487-5496.

[5] E. Furbo, J. Harju and H. Nilsson. 2009. Evaluation of turbulence models for prediction of flow separation at a smooth surface. Unpublished.

[6] G.A. Harikrishnan, PSA. Ebin and JS A Jayakumar. 2014. CFD Simulation of Subcooled Flow Boiling using OpenFOAM. International Journal of Current Engineering and Technology. Special Issue-2.

[7] J. Izarra Labeaga. 2013. Two phase pipe flow simulations with OpenFOAM. Master Thesis, Department of Energy and Process Engineering, Norwegia University of Science and Technology.

[8] E. Furbo. 2010. Evaluation of RANS turbulence models for flow problems with significant impact of boundary layers. UPTEC F10061.

[9] J.H. Ferziger and M. Peric. 2002. Computational Methods for Fluid Dynamics. Springer 3rd Ed.

[10] S.B. Pope. 2000. Turbulent Flows. Cambridge University Press.