International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:11, No:6, 2017

# CFD Modeling of Air Stream Pressure Drop inside Combustion Air Duct of Coal-Fired Power Plant with and without Airfoil

Pakawhat Khumkhreung, Yottana Khunatorn

***Abstract*****—**The flow pattern inside rectangular intake air duct of 300 MW lignite coal-fired power plant is investigated in order to analyze and reduce overall inlet system pressure drop. The system consists of the 45-degree inlet elbow, the flow instrument, the 90-degree mitered elbow and fans, respectively. The energy loss in each section can be determined by Bernoulli's equation and ASHRAE standard table. Hence, computational fluid dynamics (CFD) is used in this study based on Navier-Stroke equation and the standard k-epsilon turbulence modeling. Input boundary condition is 175 kg/s mass flow rate inside the 11-$m^2$ cross sectional duct. According to the inlet air flow rate, the Reynolds number of airstream is $2.7 \times 10^6$ (based on the hydraulic duct diameter), thus the flow behavior is turbulence. The numerical results are validated with the real operation data. It is found that the numerical result agrees well with the operating data, and dominant loss occurs at the flow rate measurement device. Normally, the air flow rate is measured by the airfoil and it gets high pressure drop inside the duct. To overcome this problem, the airfoil is planned to be replaced with the other type measuring instrument, such as the average pitot tube which generates low pressure drop of airstream. The numerical result in case of average pitot tube shows that the pressure drop inside the inlet airstream duct is decreased significantly. It should be noted that the energy consumption of inlet air system is reduced too.

***Keywords*****—**Airfoil, average pitot tube, combustion air, CFD, pressure drop, rectangular duct.

## I. Introduction

CFD is a tool that is commonly used in engineering problem analysis, by using mathematical techniques and procedures to simulate the answer which cannot be acquired directly through calculation. In the other words, the development of CFD system has opened more options to deliver an optimized design for engineering architecture and has become a large factor for helping designers in making hard decisions [1]-[3]. In this case, CFD software named ANSYS CFX® has been used to analyze and optimize the design for the combustion air system of a 300 MW coal-fired power plant in order to find the pressure drop in each section of suction side system and to suggest the method to decrease power consumption in production process [4].

Pakawhat Khumkhreung was with Chaing Mai University, Muang, Chaing Mai 50200 Thailand. He is now with the Electricity Generating Authority of Thailand, (EGAT), Mae Moh Power Plant, Mae Moh, Lampang 52220 Thailand (e-mail: pakawhat.k@egat.co.th).

Yottana Khunatorn is Assistant Professor Doctor with the Chaing Mai University, Muang, Chaing Mai 50200 Thailand (phone: 66-053-944-146; fax: 66-053-944-145; e-mail: piakman@gmail.com).

## II. CFD Modeling Setup

The major components of combustion system consist of four fans which are divided into two primary air fans (PAF) for carrying coals into steam generator (boiler) and two forced draft fan (FDF) which are functioning to provide sufficient oxygen for the combustion process, which is approximately 320-360 kg/s of air (depend on heating value) into boiler [5]. Currently the power consumption for operating the fans is approximately 3,400 kW during normal operation condition [6]. CFD modeling pre-process of combustion system is set as heading A to E and the validated process detail is in heading *F*.

### *A. Modeling and Mesh*

1. Model Generate

The generated model size has 1:1 scale ratio. The symmetry technique is used in this research, to reduce calculation domain and computational time, therefore the mass flow rate will be considered only half of using in the actual system (160-180 kg/s) [7].

2. Mesh Generate

Type of mesh that is used for calculation is tetrahedral unstructured mesh approximately 3,639,803 mesh (663,563 nodes) [8].

### *B. Grid Dependency Study*

The purpose of grid dependency study is to find the least mesh amount that has minimum effect on numerical result in CFD modeling in order to efficiently reduce calculation time and to acquire an accurate and reasonably results [9]. In this case, air velocity shall be calculated from three different mesh resolutions, and we will compare each result with the actual velocity measured from the calibration point of flow transmitter as shown in Fig. 3 (see more details in the section of "Validation of Numerical Results").

### *C. Fluid Properties*

Air properties observed in this model are dry air under standard conditions (25 °C temperature, atmospheric pressure 1 atm, density ($\rho$) 1.185 kg/$m^3$, and viscosity coefficient ($\mu$) $1.831 \times 10^{-5}$ Pa.s). However, under actual operation condition, the temperature is varied between 20 and 35 °C and may affect other parameters, disrupting the result of calculation. To decrease the discrepancy, the plot comparison of mass flow rate is done in terms of Reynolds number (Re).

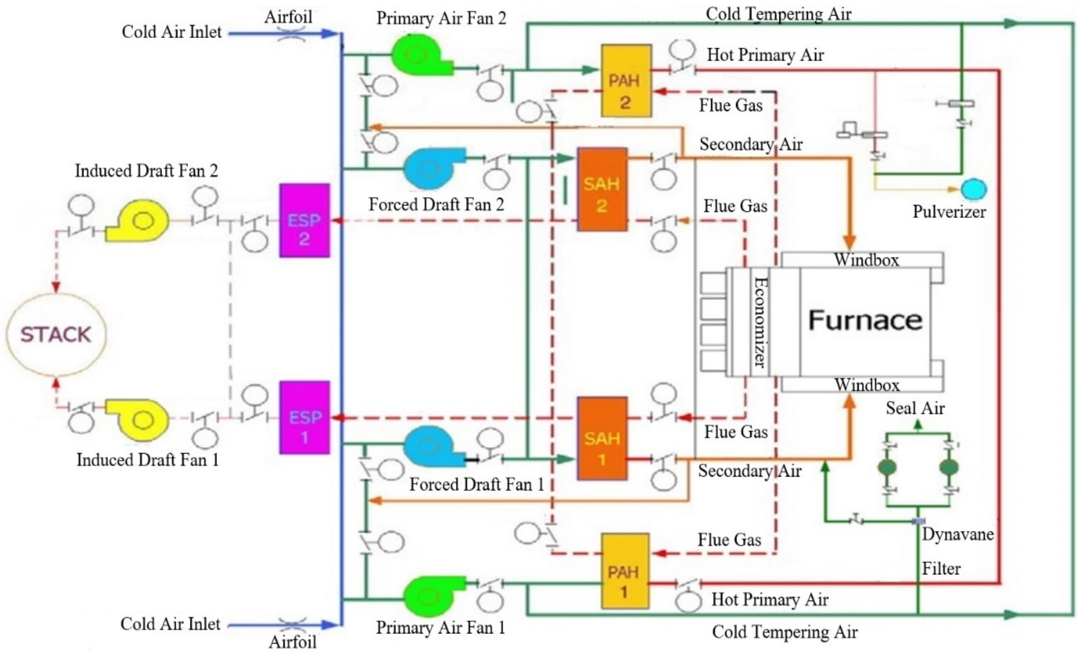


Fig. 1 Combustion system diagram of 300 MW Mae-Moh coal-fired power plant [6]

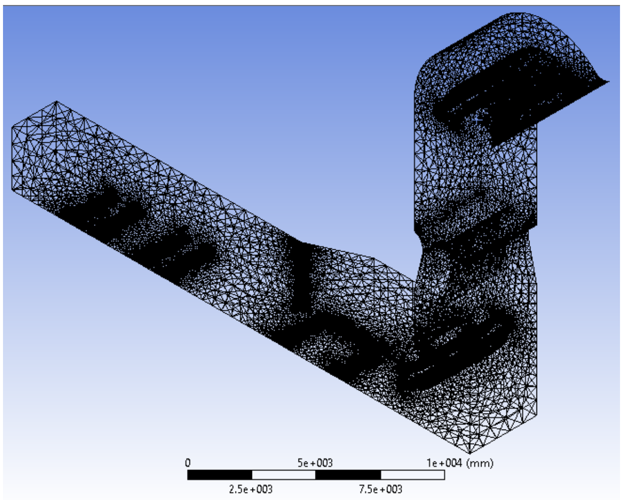


Fig. 2 Model and mesh generate

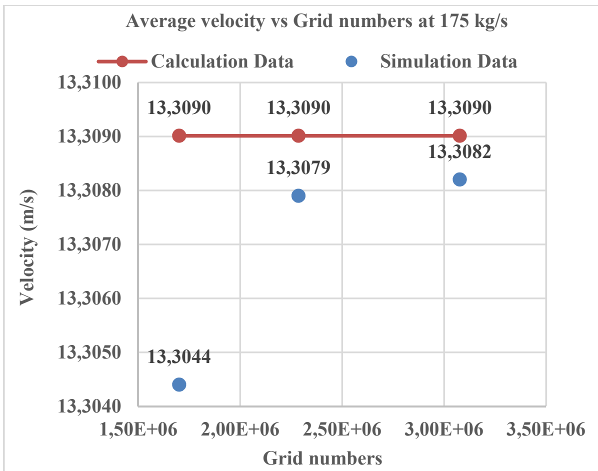


Fig. 3 Grid independency study at mass flow rate 175 kg/s

*D. Boundary Condition*

Boundary condition in CFD modeling considered from normal operation condition of Mea-Moh (Lampang, Thailand) power plant at unit 8 to 13. The air flow range in combustion process varies between 330-360 kg/s. The selected flow rate is 350 kg/s condition, the proportion of air drawn to the PAF and force draft fan is 150 and 200 kg/s or 3/7 and 4/7, respectively [5], [6]. As problem pattern of model is symmetrical [7], the boundary conditions of inlet and outlet gas flow are defined as shown in Table I.

TABLE I
INLET AND OUTLET BOUNDARY CONDITIONS

| 1. Inlet 1 | Static Pressure | | | | Pa |
|---|---|---|---|---|---|
| 2. Outlet 1 | 70.71 | 72.85 | 75.00 | 77.14 | kg/s |
| 3. Outlet 2 | 47.14 | 48.57 | 50.00 | 51.42 | kg/s |
| 4. Outlet 3 | 47.14 | 48.57 | 50.00 | 51.42 | kg/s |
| Summary | 165.00 | 170.00 | 175.00 | 180.00 | kg/s |

*E. Solver Control and Analysis*

After configuring the fluid properties and boundary conditions of the model, data evaluation should be done, and the user should consider the following factors in order to calculate the preferable result and to provide the most optimized data processing [8].

TABLE II
SOLVER CONTROL AND ANALYSIS SETTING

| Parameters | Configure of project |
|---|---|
| 1. Convergence criteria determination | RMS target = 0.00001 |
| 2. Conservation target determination | 0.01 |
| 3. The condition of the answer from solving the problem | steady-state |
| 4. Choosing turbulence model [2], [10] | *k*-*ε* model |

### *F. Validation Numerical Results*

The measured data are the velocity head which is obtained by pitot tube according to standard ASTM D3514-00 [11]. The data are collected before and after yearly inspection [5]. It is used to validate numerical results. Measurement regions are shown in Figs. 4 and 5, respectively [5], [11].

TABLE III
PRESSURE DROP EACH SECTION

| | Re | 2,662,983 | 2,743,679 | 2,824,376 | 2,905,072 |
|---|---|---|---|---|---|
| Plane | Mass flow rate (kg/s) | 165 | 170 | 175 | 180 |
| 1. Inlet 1 – Plane 1 | Inlet elbow (mmWG) | 1.50 | 1.59 | 1.68 | 1.78 |
| 2. Plane 1 – Plane 2 | Flow instrument (mmWG) | 16.27 | 17.26 | 18.29 | 19.34 |
| 3. Plane 2 – Plane 3 | 90° mitered (mmWG) | 2.85 | 3.02 | 3.20 | 3.38 |
| 4. Plane 3 – Plane 4 | Inlet PAF (mmWG) | 0.46 | 0.49 | 0.52 | 0.55 |
| 5. Plane 4 – Plane 5 | Inlet reducer (mmWG) | 0.37 | 0.39 | 0.42 | 0.44 |
| 6. Plane 5 – Plane 6 | Inlet force draft fan 1 (mmWG) | 0.24 | 0.26 | 0.27 | 0.29 |
| 7. Plane 6 – Plane 7 | Inlet force draft fan 2 (mmWG) | 0.18 | 0.19 | 0.20 | 0.21 |
| Summary (mmWG) | | 21.87 | 23.20 | 24.57 | 25.98 |

TABLE IV
COMPARE PRESSURE DROP BEFORE AND AFTER MODIFY

| Re | 2,662,983 | 2,743,679 | 2,824,376 | 2,905,072 |
|---|---|---|---|---|
| Mass flow rate (kg/s) | 165 | 170 | 175 | 180 |
| ΔP Airfoil (mmWG) | 16.27 | 17.26 | 18.29 | 19.34 |
| ΔP Average pitot tube (mmWG) | 0.87 | 0.92 | 0.98 | 1.03 |
| Decrease (%) | 94.62 | 94.63 | 94.64 | 94.64 |
| ΔP System before modifying (mmWG) | 21.87 | 23.20 | 24.57 | 25.98 |
| ΔP System after modifying (mmWG) | 6.48 | 6.87 | 7.27 | 7.68 |
| Decrease (%) | 70.36 | 70.39 | 70.43 | 70.45 |

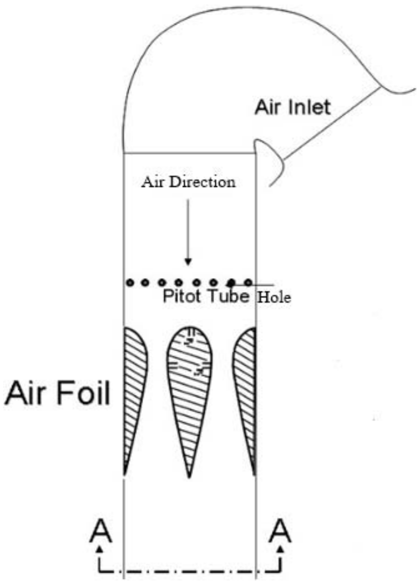


Fig. 4 Data measurement location [5]

Velocity head from measurement is averaged to represent the whole data by (1)

$$Velocity\ head = \left(\frac{\sum\sqrt{Velocity\ head\ data}}{Number\ of\ points}\right)^2 \qquad (1)$$

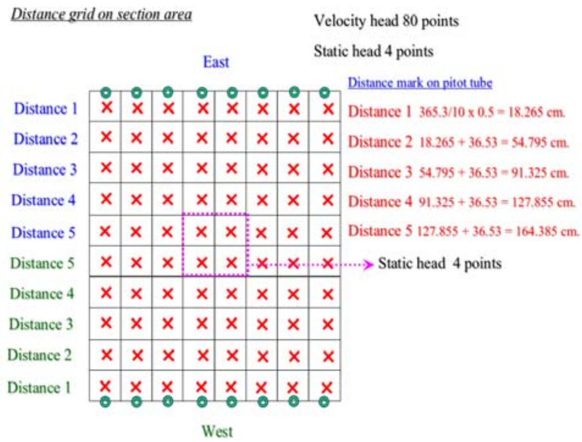


Fig. 5 Data collection point [5]

CFD modeling result and data from measurement will be considered by making comparison with the Reynolds number, which is calculated from flow rate at 160-180 kg/s and average velocity head as shown in Fig. 6.

The relationship between Reynolds number and average velocity head can have a linear relation as shown in (2)

$$Y = 7X\,x\,10^{-6} - 9.2173 \qquad (2)$$

where $Y$ is the average velocity head (mmWg), and $X$ is the

Reynolds number.

After substituting Reynolds number from the CFD modeling, it is found that the errors from mesh level 1, 2, and 3 are 1.5%, 1%, and 0.5 %, respectively.

As a result, the CFD modeling and measurement data are well agreed when the number of mesh is 3,639,803.

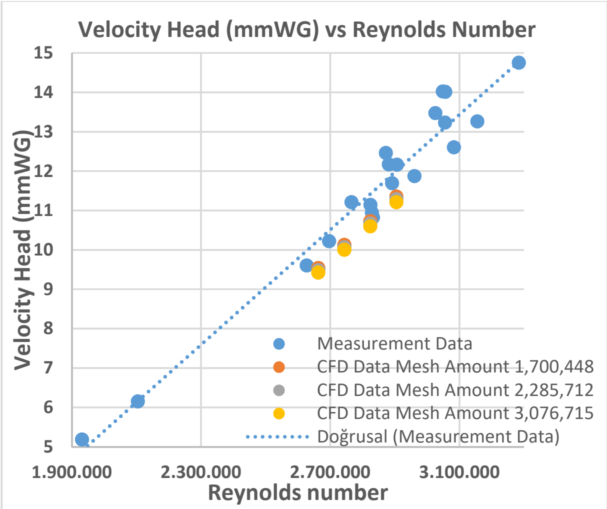


Fig. 6 Comparison result between measurement and CFD modeling

## III. Result and Discussion

Table III represents the pressure drop at each section from CFD modeling along suction system, as shown in Fig. 7 [8].

It is found that the maximum pressure drop in suction system occurs across the mass flow rate instrument installation area, which is about 75% of the whole domain pressure drop.

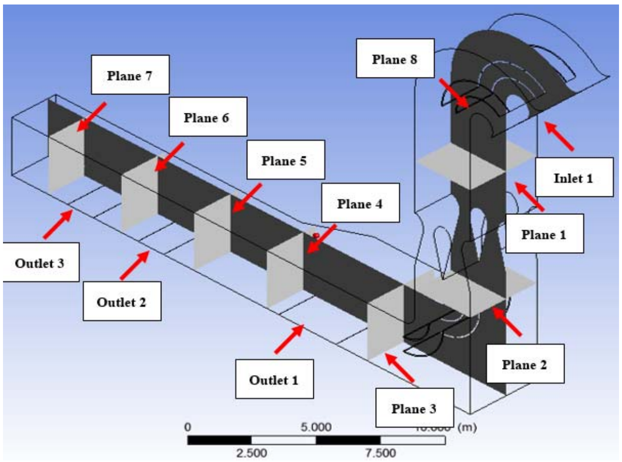


Fig. 7 Cut plane location

Figs. 8 and 9 show that flow rate instrument causes the maximum pressure to drop in suction system. Magnitude of average velocity increases through the instrument. Both figures confirm the result of numerical presented in Table III.

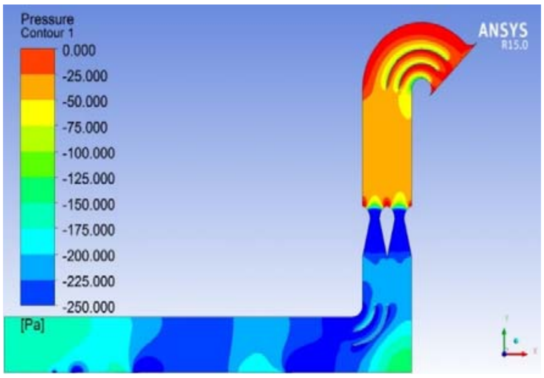


Fig. 8 Contour pressure at mass flow rate 175 kg/s

In order to decrease the pressure drop in suction system, original instrument, airfoil type, is replaced by the average pitot tube in CFD modeling. Both instruments use the same method to measure mass flow, but the replacement has the advantages in the smaller dimensions.

Figs. 10 and 11 show the pressure contour and magnitude of average velocity contour for uniform flow from cold air inlet through suction. Effect of the decreased pressure drop makes the range of pressure contour narrow down.

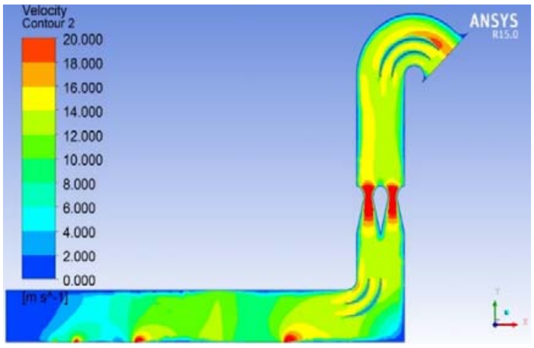


Fig. 9 Contour velocity at mass flow rate 175 kg/s

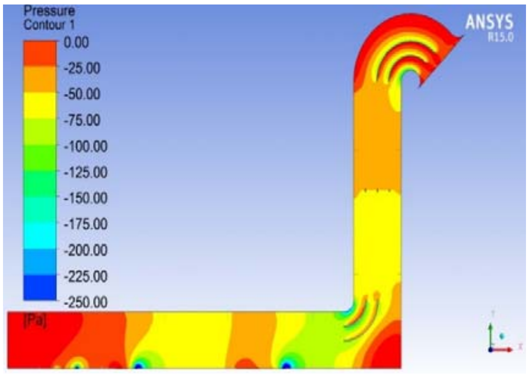


Fig. 10 Contour pressure at mass flow rate 175 kg/s after modifying CFD modeling

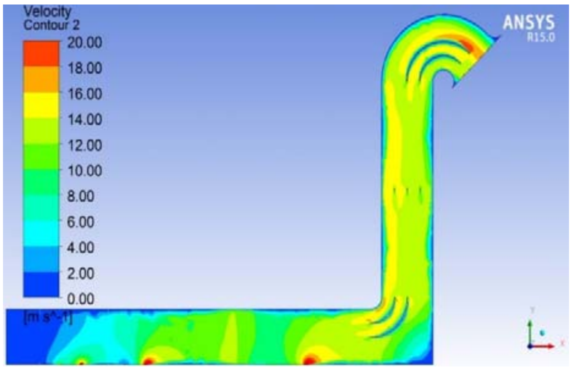


Fig. 11 Contour velocity at mass flow rate 175 kg/s after the modification of CFD modeling

It is clearly shown that the main of pressure drop in system comes from flow obstacle in measurement area, which was decreased by changing air flow instrument with the new one that has less obstructive area.

Table IV shows results before and after replacing the average pitot tube in CFD modeling. The systems pressure drop decrease is approximately equal to 71%. It can be estimated in terms of suction duct work gain in (3) at 350 kg/s (symmetry model) and 17.3 mmWG (169.81 Pa) [12].

$$w = \frac{\dot{m}\Delta P}{\rho \eta_t \eta_m \eta_f} \tag{3}$$

where $w$ is the fan power (Watt), $\dot{m}$ is the mass flow rate (kg/s), $\Delta P$ is the decreased pressure drop (Pa), $\rho$ is the air density (kg/m$^3$), $\eta_t$, $\eta_m$, and $\eta_f$ are the transmission efficiency, the motor efficiency, and the fan efficiency, respectively.

Equation (3) can proximate duct work gain which is 398,142 kWh.

## IV. Conclusion

The combustion system pressure drop suction side of 300 MW Mae-Moh coal fired power plant is analyzed by CFD, in order to evaluate preliminary results before and after modifying the system. CFD modeling is validated with measurement air flow data in the case of real operation. It is found that the numerical results agree with the real data at mesh amount of 3,639,803 with error 0.5%. The maximum pressure drop occurs at region flow instrument airfoil type. Thus, the original is replaced in CFD modeling with the average pitot tube which generates low pressure drop. The loss in system suction side was reduced by 71% after the airfoil had been replaced. Furthermore, the concept of this research can be applied to other systems of power plants and other industries.

## Acknowledgment

Thank you to the Electricity Generating Authority of Thailand (EGAT), who is the research sponsor and support ANSYS CFX® software.

## References

[1] R. W. Lewis, P. Nithiarasu and K. N. Seetharamu, Fundamentals of the Finite Element Method for Heat and Fluid Flow, John Wiley and Sons Ltd, 2004.
[2] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics, Pearson Education Ltd, 2007.
[3] J. D. Anderson Jr, Computational Fluid Dynamics The Basics with Applications, McGraw-Hill, 1995.
[4] Dr. N Al-Khalidy, Design Optimization of Industrial Duct Using Computational Fluid Dynamics, CSIRO, Melbourne, Australia, 10-12 December 2003.
[5] Electricity Generating Authority of Thailand, Technical Data Book Plant Operation Division 2, MAE MOH Power Plant Operation Department, 1989.
[6] Electricity Generating Authority of Thailand, Mae-Moh 8-13 System Description Level 1 Boiler Auxiliary Operation: Primary and Force Draft Fan, MAE MOH Power Plant Operation Department, 2007.
[7] C. Bhasker, Simulation of Air Flow in the Typical Boiler Windbox Segments, Advances in Engineering Software 33 (2002) 793–804.
[8] ANSYS CFX® Manual, ANSYS Ltd, 2015.
[9] K. Sudo, M. Sumida and H. Hibara, Experimental Investigation on Turbulent Flow in a Square-Sectioned 90-Degree Bend, Experiments in Fluids 30 (2001) 246-252 @ Springer-Verlag 2001.
[10] D. C. Wilcox, Turbulence Modeling for CFD, DCW Indutries Inc, 1994.
[11] The American Society for Testing and Materials, Standard Test Method for Average Velocity in a Duct (Pitot Tube Method) Designation D 3154 – 50, 1995.
[12] ASHRAE, Fundamentals. American Society of Heating, Refrigeration and Air Condition Engineer, Atlanta, 1997.