International Journal of
Innovation in Mechanical Engineering & Advanced Materials (IJIMEAM)
https://publikasi.mercubuana.ac.id/index.php/ijimeam

Vol.4 (No.1). 2022. pp. 1-6
Published online: August 12, 2022
ISSN: 2477-541X

CFD SIMULATION FOR AIRSPEED AND TURBULENCE VALIDATION IN MAIN DUCTING OF OFFICE BUILDING

H. P. Sabilly[1] and S. K. Deni[1,2]*

[1] Mechanical Engineering Master Program, Universitas Mercu Buana, Jakarta, INDONESIA
[2] Research Center for Physics, Indonesian Institute of Sciences (LIPI), Tangerang Selatan, INDONESIA

Abstract
This paper focuses on the calculation of sizing ducting based on cooling load requirements the main ducting of office building following regulation airspeed requirements using American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) and Computational Fluid Dynamics (CFD) simulations. The purpose of this research is to validate the airspeed and turbulence that occurs in the main ducting between manual calculations and CFD simulations. From the calculation, the cooling load requirement is 58.22 kW, for the cooling process an air flowrate of 7117 L/s is needed which is designed to pass through the main ducting in rectangular shape. The main ducting size uses 1200 mm x 500 mm at a speed of 12.7 m/s according to ASHRAE. Autodesk Inventor software is used for ducting modeling and Autodesk CFD is used for airflow simulation. CFD simulations are performed by applying boundary conditions and input parameters. The results showed that the velocity of the ducting design was suitable at 12.7 m/s with laminar flow. The ducting geometry must be designed aerodynamically to reduce the pressure drop which can cause the speed to increase so that it is outside the required limits. Thus, the CFD simulation results have verified the validity of manual calculations.

Keywords: ASHRAE, Cooling load, Ducting, Computational Fluid Dynamics (CFD)

*Corresponding author: Tel. +6281291759857
E-mail address: sabilly.handi.pradana@gmail.com

1. Introduction

Nowadays, the need for an air-conditioning system in buildings is required to obtain comfortness, especially when the building is an office where there are a lot of people activities and utilities that generate heat. It is common to design an air-conditioning system by using an heating, ventilation and air-conditioning systems (HVAC)[1]. AC functions to produce cold air which will be distributed throughout the room using a ducting system. In designing cold air to be distributed, it must pay attention to the cooling load needed to reduce the temperature from the initial conditions to the final conditions[2]. Thus, to ensure the analytical calculations on the design are appropriate. It can be evaluated by experimentation of the air flow rate through the ducting system. There are several parameters that must be maintained, such as air velocity and pressure. Meanwhile, in design testing, it is not necessary to do the real experiments since it costs time and money for the production process[3].

Computational fluid dynamics (CFD) is used as a numerical approach method in simulating fluid flow, including changes in velocity and fluid pressure[4]. This study focuses on modeling the air flow that passes through a main ducting from a source in a downward position towards the ducting section above. The simulated airflow is obtained from the cooling load temperature difference (CLTD) calculation and the calculation of sizing ducting following the permissible velocity requirements referring to the American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) standard. The numerical approach based on CFD is an effective approach to fluid flow investigations that have been validated by previous studies with various experiments[5].

The application of CFD for cooling air distribution system using ducting has been carried out by several previous researchers. Vishal[6], simulating fluid flow in ducting with a round and box shape with an air capacity of 1230 CFM distributed to 5 rooms, namely: offices 1, 2, conference rooms, lunch break rooms with a main duct diameter of 15.7 inches which have been designed, obtained variations of air velocity based on position variations and displays the pressure drop ratio.

In a rectangular shape, the air flow turbulence is greater than in a round shape but has the advantage of being able to adjust the dimensions of the available area. The geometric shape of the transport media affects fluid flow[7]. Nowak[8], simulating solid fan

Vol.4 (No.1). 2022. pp. 1-6
Published online: August 12, 2022
ISSN: 2477-541X

airflow on a hydro cyclone using Gambit and Fluent software to determine the effect of changes in air velocity through changes in cone angle and diameter variations. The simulation results show that the tangential flow increases with the applied pressure. Generally, this study will also apply a similar concept in validating such fluid flow velocity; however, with different research objects. The purpose of this study is to validate the speed and turbulence of cooling air passing through a main ducting between manual calculations using ASHRAE reference with CFD simulation.

2. Experimental and Procedures

2.1 Research Object

This research was conducted on the main ducting system design of a palm oil mill office building according to Fig. 1 with a size of 1200 mm x 500 mm. The ducting was designed in a rectangular shape that is installed starting from the AC type variable refrigerant flow (VRF) output to the branch ducting with a capacity of 7177 L/s. The data is obtained from the calculation of cooling load requirements in all office spaces following the ASHRAE equation as shown in Table 1.

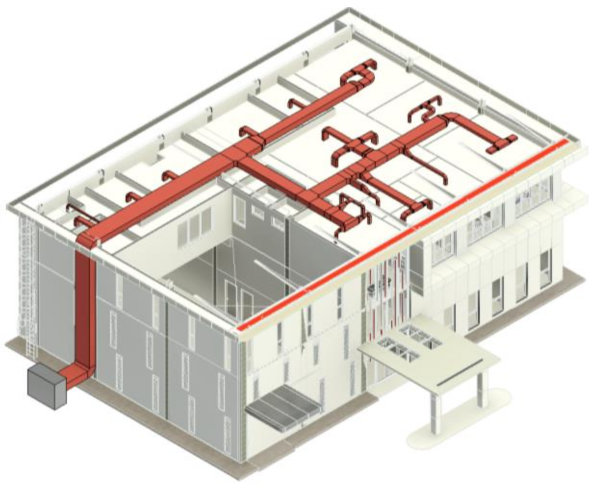

Fig. 1. Office ducting design

Table 1. Cooling load calculation[9]

Cooling Load	Load Type	Formula	Note
Glass	Radiation	Q = U x A x ΔT	U_g = 5.9 W/m²-K
	Conduction	Q = SHGF x CLF x A x SC	SHGF = 230; CLF = 0,1-0,5; SC = 0,95
Roof	Conduction	Q = U x A x ΔT	U_{roof} = 0.52 W/m²-K
Wall	Conduction	Q = U x A x ΔT	$U_{concrete}$ = 2.69 W/m²-K ; U_{gypsum} = 1.37 W/m²-K
Occupant	Sensible	Q_s = n x Q_s x CLF	Q_s = 0.067 kW ; CLF = 0-0.84
	Latent	Q_l = n x Q_l	Q_l = 0.055 kW ; CLF = 0-0.84
Lamp	Sensible	Q_s = 3.41 x W x F_u x F_{sa} x CLF	F_u = 1 ; F_{sa} = 1.2 ; CLF = 0.1-1
Electrical equipment	Sensible	Q_s = Q_{in} x F_u x F_r x CLF	F_u = 1 ; Fr = 1 ; CLF = 0.1-1
	Latent	Q_l = Q_{in} x F_u	F_u = 1
Ventilation & Infiltration	Sensible	Q = 1.08 x CFM x ΔT	
	Latent	Q = 4840 x CFM x ΔW	

where
Q : cooling load, kW
Q_s : sensible heat, kW
Q_l : latent heat, kW
U : thermal transmittance coefficient, W/m²-K
n : quantity, unit(s)
CLF : cooling load factor
SCL : solar cooling load
SC : shading coefficient
SHGF: solar heat gain factor
CFM : cubic feet per minute
F_u : usage factor
F_r : radiation factor
A : area, m²

The room criteria (RC) level category used for this office is 40-45 (N) which is equivalent to a sound intensity of 50-55 dB (A). The maximum allowable air velocity for the main ducting following the RC level is 12.7 m/s[9].

The conversion from cooling load power requirements to air capacity follows the equation:

$$\frac{BTU}{hr} = CFM\ x\ \Delta Tair\ x\ 1.08 \quad (1)$$

Eq. 4 is to determining the size of the ducting from the terminology of the concept of head and pressure, the equation is

$$p_v = \frac{\rho V^2}{2} \quad (2)$$

For air conditions (ρ_{air} = 1.204 kg/m³), Eq. 2 becomes

$$p_v = 0.602\ V^2 \quad (3)$$

Air velocity is obtained from the following equation.

$$V = 0.001Q/A \quad (4)$$

Vol.4 (No.1). 2022. pp. 1-6
Published online: August 12, 2022
ISSN: 2477-541X

where

p_v : velocity pressure, Pa
V : average airspeed, m/s
Q : air capacity, L/s
A : ducting cross-sectional area, m^2

The simulations are performed in a steady state condition, using a turbulent flow and k-ε model. This condition is appropriate for evaluation of airflow and heat transfer inside the closed domains. The results obtained refer to the velocity data inside the ducting. Performing numerical simulations carried out by Autodesk CFD, the differential equations of heat transfer and fluid mechanics were determined[10].

The controlling equations of fluid flow represent mathematical statements of the conservation laws of physics[11]:

- The mass of a fluid is conserved.
- The rate of change of momentum must be equal to the sum of the forces on a fluid particle.
- The rate of change of energy must be equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle.

Conservation of mass (incompressible fluid)

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{5}$$

Conservation of momentum

$$\rho\frac{Du}{Dt}=\frac{\partial(-p+\tau_{xx})}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+S_{Mx} \tag{6}$$

$$\rho\frac{Dv}{Dt}=\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial(-p+\tau_{yy})}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+S_{My} \tag{7}$$

$$\rho\frac{Dw}{Dt}=\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial(-p+\tau_{zz})}{\partial z}+S_{Mz} \tag{8}$$

Conservation of energy

$$\frac{\partial}{\partial t}(\rho E)+\nabla.\big(v(\rho E+p)\big)=-\nabla.\left(\textstyle\sum_j h_j J_j\right)+S_h \tag{9}$$

To determine the flow turbulence in ducting, the concept of Reynolds Number is used. Flow-through pipe/ducting are classified into 3 main flows: laminar flow R<2000, critical flow 2000>R>4000, and turbulent flow R > 4000[11].

$$R_e=\frac{\rho.d.v}{\mu} \tag{10}$$

where

R_e : Reynold number
P : density of the fluid, kg/m^3
d : diameter of ducting, m
u : flow speed, m/s
μ : dynamic viscosity of the fluid, Pa·s

2.2 Boundary condition

The boundary conditions applied for this simulation are as follows:

- In the inlet and outlet areas, the air capacity, pressure and temperature are given according to the data in Table 2.
- The parameters of the fluid and transport media using air and mild steel are according to Table 3.
- Incompressible fluid material so that it follows the rules of conservation of mass, energy and momentum.

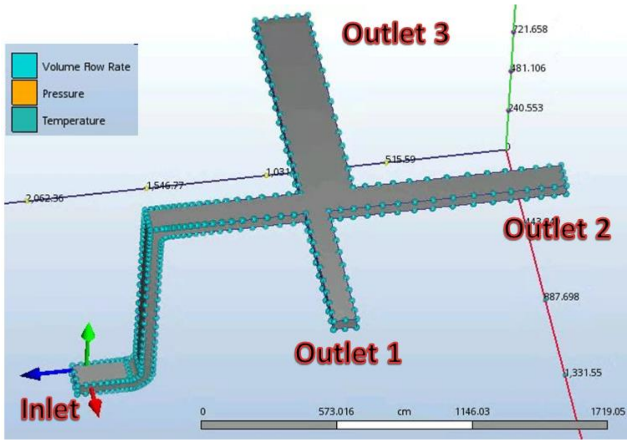

Fig. 2. Parameter input and meshing process

Table 2. Simulation initial conditions parameters

Parameter	Inlet	Outlet 1	Outlet 2	Outlet 3
Flowrate (L/s)	7117	0	0	0
Pressure (Pa)	200	0	0	0
Temp. (°C)	16	36	36	36

Table 3. Air and steel material parameters

Material	Parameter	Properties
Air	Density	Equation of state
	Viscosity	1.817e-05 Pa-s
	Conductivity	0.02563 W/m-K
	Specific heat capacity	1004 J/kg-K
	Compressibility	1.004
	Emissivity	0.9
Mild Steel	Density	7833 kg/m^3
	Specific heat capacity	465 J/kg-K
	Emissivity	0.3
	Transmissivity	0

2.3 Simulation Procedure

The main ducting design is modeled from the results of manual ASHRAE calculations using Autodesk Inventor and then converted into an Autodesk CFD file to simulate fluid flow performance such as flow-rate, temperature, speed and turbulence. Material properties air and ducting used follow the data in Table 3 using material references from Autodesk CFD software. This research was conducted based on the following flow chart steps:

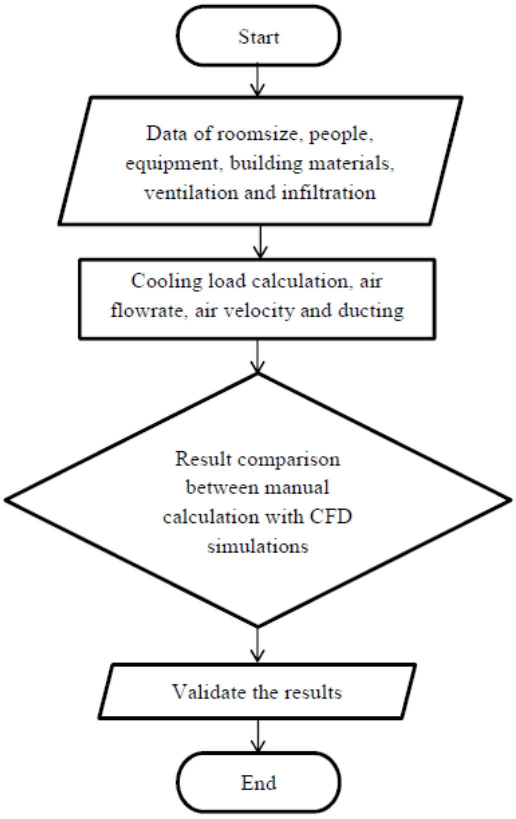

Fig. 3. Flowchart method

3. Results and Discussion

3.1 Result of Calculation

The results are presented comparatively for 2 studied cases between manual and simulation in the main ducting of a building office. Manual calculations of air supply requirements to meet the cooling load for office space based on ASHRAE references have been carried out. From the equations that are presented in Table 1, it is found that the power load requirements are 58.22 kW with an air supply of 7117 L/s. Eq. 4 is used in determining the size of the ducting with the condition that the maximum air velocity through the ducting is 12.7 m/s in order to obtain a comfortable sound condition from noise.

In the design, air transport media is used in a rectangular shape in order to obtain size flexibility. Ducting is made by using plates based on mild steel material. From the calculation results, the size for the minimum speed requirements achieved is 1200 mm x 500 mm, as shown in Table 4.

From the calculation of Eq. 10, it is found that the Reynold number is 555.9. From this data it is found that the flow will be laminar, since R<2000, as listed in Table 4.

Table 4. Data analysis based on calculation

Description	Unit	Data
Air volume	L/s	7117
Duct size	mm	1200 x 500
Duct material		Mild Steel
Abs. Roughness	mm	0.15
Eq. diameter	mm	827.3
Hydraulic diameter	mm	705.9
Duct velocity	m/s	11.9
Max. velocity	m/s	12.7
Velocity Pressure	Pa	84.7
Reynold numbers		555.9
Friction factor		0.015
Duct friction	Pa/m	1.8

3.2 Result of CFD Simulation

Autodesk CFD software has been used to simulate the velocity and turbulence of the flow applied to a main ducting with the input from Table 4. The simulation result value is represented by color. From Fig. 4 and Fig. 5, it shows the magnitude of the velocity of air flowing from the bottom towards the top through 2 bends. The model was designed to only follow the size without regard to the geometry of the elbow and is not aerodynamic and can produce speeds in the range of 12-16 m/s beyond the standard limits. This happens because the geometry is not aerodynamic, so that there is a large pressure drop on the bend which can cause the airspeed to exceed the allowable limit conditions as marked.

Fig. 6 and Fig. 7 show the improved ducting model using a bend radius. It shows that the air velocity is in the range of 10-12 m/s, this follows allowable limit standards. The model produces laminar flow so that turbulence does not occur. In the whole part of the duct, the air velocity is uniformed except at the bend area. From the results, bend area and size play an important role in the case of velocity distribution.

Vol.4 (No.1). 2022. pp. 1-6
Published online: August 12, 2022
ISSN: 2477-541X

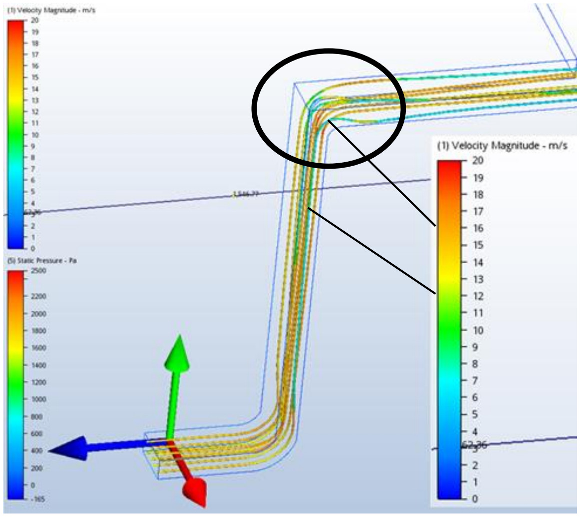

Fig. 4. Air velocity initial design

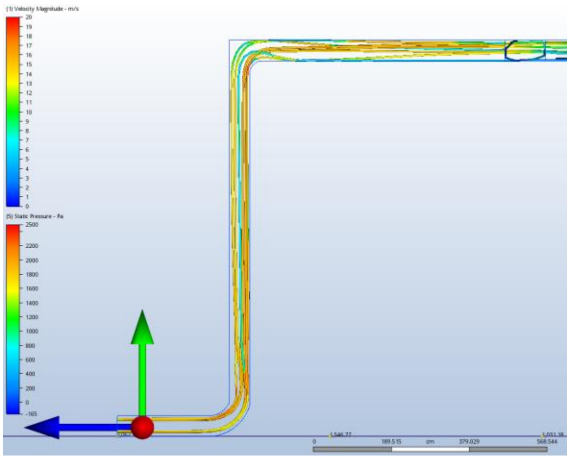

Fig. 5. Air velocity initial design (section)

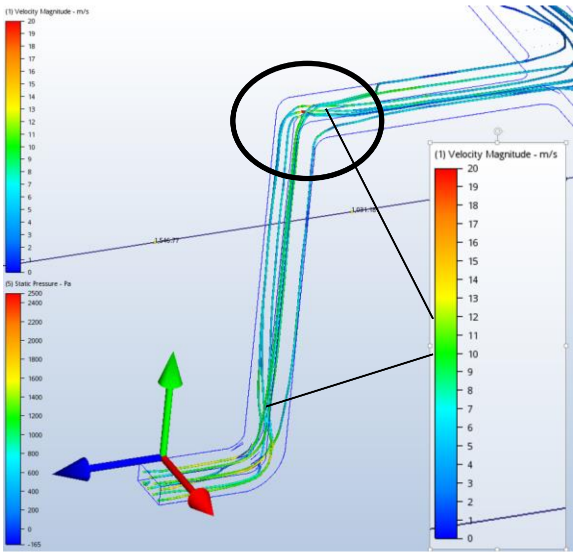

Fig. 6. Air velocity improvement design

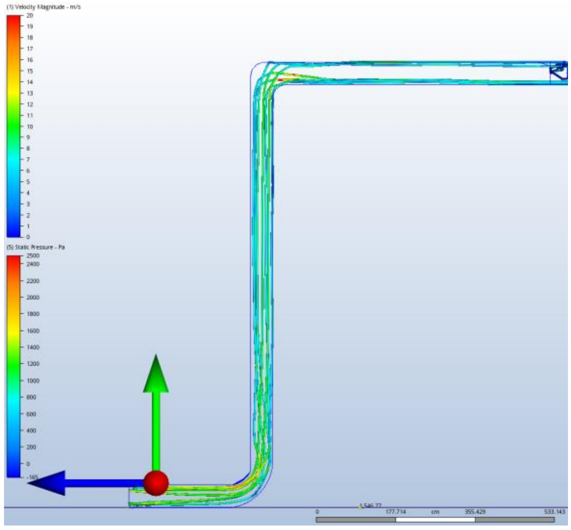

Fig. 7. Air velocity improvement design

Comparing the two results between manual calculation and simulation yields the same values for air velocity and laminar flow. The result is the same value of about 12 m/s, there is a difference in geometry that is not aerodynamic about 12-16 m/s resulting in a difference of up to 33%. From the CFD simulation results, it can be seen that there are other things that determine the velocity distribution besides the dimensions, namely the geometric shape must be aerodynamic. This result is in accordance with Didwania[12] experiment on the flow rate through bending, non-aerodynamic geometries can produce higher pressure drop resulting in increased air velocity. Refer to all the simulation results, the flow produced is laminar, and there is no turbulence. This result is in accordance with the Reynold Number calculation of 555.9 as per data Table 4. Thus, the CFD simulation results have verified the validity of manual calculations.

4. Conclusions

This paper examines the validation of ducting size calculations in main ducting office buildings through the allowable airspeed requirements based on ASHRAE with CFD simulations using Autodesk CFD. Simulation experiments can validate the speed and turbulence between manual calculations and simulations. By using an air capacity of 7177 L/s on the main ducting with a size of 1200 mm x 500 mm. The results showed that the velocity of the ducting design was suitable at 12.7 m/s with laminar flow condition. The ducting geometry must be designed aerodynamically to reduce the pressure drop which can cause the speed to increase that overestimate the required limits. The CFD simulation results have verified the validity of manual calculations.

5. Acknowledgements

The author would like to thank the Master of Mechanical Engineering of Mercubuana University that helped the process of carrying out the research.

References

[1] C. Huizenga, S. Abbaszadeh, L. Zagreus, E. Arens. Air Quality and Thermal Comfort in Office Buildings: Results of a Large Indoor Environmental Quality Survey. Proceeding of Healthy Buildings, Lisbon. 2006; Vol. III; 393-397.

[2] G. Srivastava, A. Kumar, D. Chandel, H. Dabas, A. Mishra, Saurabh. CFD Simulation of Air Conditioning System of the Classroom. International Journal of Trend in Scientific Research and Development (IJTSRD). 2019; Vol.3; 521-523.

[3] M. M. Tukiman, M. N. M. Ghazali, A. Sadikin, N. F. Nasir, N. Nordin, A. Sapit. CFD Simulation of flow through an orifice plate. IOP Conf. Series: Materials Science and Engineering 243. 2017; 1-6.

[4] V. V. Khakre, A. M. Wankhade. Review paper on Design and Computational Analysis of Air Flow Through Cooling Duct. International Journal of Scientific & Engineering Research (IJSER). 2014; Vol.5; 935-938.

[5] W. L. Oberkampf. Verification and validation in Computational Fluid Dynamics. Accepted in journal of Progress in Aerospaces Sciences. 2002; 1-122.

[6] Vishal S., M. H. Patil. Duct Designing in Air Conditioning System and Its Impact on System Performance. International Research Journal of Engineering and Technology (IRJET). 2019; Vol.6; 1392-1397.

[7] P.S. Dhakar. CFD Analysis of Air Conditioning in Room Using Ansys Fluent. Journal of Emerging Technologies and Innovative Research (JETIR). 2018; Vol.5; 436-441.

[8] Nowak, Z. Stegowski, B. Tora, M. Kurzac. The New Approach to the hydrocyclone modeling using Computational Fluid Dynamics (CFD) simulation. Proceedings of 23rd International Mineral Processing Congress. 2006.

[9] Charles G., Louis S. ASHRAE Handbook HVAC System and Equipment. Georgia: ASHRAE Publication. 2016.

[10] G. Popovici. HVAC system functionality simulation using ANSYS-Fluent. Sustainable Solutions for Energy and Environment. Energy Procedia. 2017; 360-365.

[11] H. K. Versteeg, W. Malalasekera. An Introduction to Computational Fluid Dynamics. Second Edition. Pearson Prentice Hall. 2007; 10-26.

[12] M. Didwania, L. Singh, A. Malik, M. S. Sisodiya. Analysis of Turbulent Flow over a 90^{o} Bend of Duct Using in Centralized A.C. Plant by CFD Code. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). 2014; Vol.11; 41-48.