

Simulation of Mass Flow Rate of Air through Rectangular Split Duct



Vaibhav N. Deshmukh, Sagar A. Sirsat

Abstract: Adequate supply of oxygen is essential for the complete combustion of fuel in the furnace of a boiler and defective duct design may cause serious loss in the boiler efficiency due to incomplete combustion. In boiler there is a system which controls the air flow rate reaching to the furnace. FD fan helps to build the pressure and force the air to pass through the duct and wind box. In wind box, this air will be sent to the furnace for helping combustion process. The main objective of this work was to simulate the flow of air from blower to the inlet of a wind box through rectangular split duct to see whether it could provide improved flow conditions and to see whether there was equal mass flow rate of air through both the legs of a split duct. The purpose of providing a split duct was to eliminate one blower which would otherwise be required for two separate ducts arrangement. A 3D model using CFD was prepared to capture the changes in the mass flow distribution in the duct at two different outlets. Through this analysis, it was found that the mass flow rates of air through the two legs of the split duct were not equal. The damping needs to be provided for the duct having longer length in order to have equal mass flow rate of air through both the legs of the duct.

Keywords: CFD, rectangular duct, split duct, mass flow rate

I. INTRODUCTION

In general, to design a system in which the behaviour of flowing fluid needs to be considered is very difficult. But now with the help of high configuration computers, it is possible to get an idea about fluid flow behavior in the product before manufacturing the actual product. Engineering design optimization both shortens design cycle time and finds new designs that are not only feasible, but also optimal based on the design criteria. Traditional engineering design processes involve strategies such as trial and error, use of previous experience till the requirements are either met or changed to fit the performance. Often, the process is time consuming and does not produce the best design. In this work, the duct system was analyzed and preliminary design concepts for the redesign of the systems based on the detailed flow analysis are

provided. Proper supply of oxygen is essential for any combustion process. In boiler there is a system which controls the air flow reaching to the furnace. FD fan helps to build the pressure and force air to pass through the duct and wind box. In wind box, this air will be sent to furnace for helping combustion process. This paper is about the analysis of different mass flow rate input from FD fan to duct which is connected to wind box. A CFD analysis for rectangular duct is conducted in order to understand and observe the mass flow rate changes induced in the split rectangular duct. Details and results of the analysis are presented in this paper. The proper mass flow rate through duct system is important which directly affects the efficiency of that system. This paper includes the geometry of rectangular duct which provides air from FD fan to inlet of wind box. There is slit in rectangular duct which might affect the mass flow rate of air at two outlets. To do analysis of this split duct, it was decided to use ANSYS workbench version 16.0. Once the system analysis was done, the next important step in analysis was meshing and getting results from post processor after providing input value in solver. This split rectangular duct analysis will be useful to decide future design aspect of duct and further modification into the split duct.

II. LITERATURE REVIEW

There has been work done by few researchers on the measurement of gas flow through rectangular ducts. Some of the work is discussed here.

Piotr Ostrowski et al. [1] proposed the method of measurement of gas flow through rectangular ducts with short straight sections and with considerable cross-section. The measurement was conducted at the neck of the duct with a single-point sensor such as the Prandtl tube. This measuring method may be used for newly designed gas ducts, as well as for those already in service, such as air conditioning or ventilation systems and power boilers.

P. Devakumaran et al. [2] carried out a CFD analysis of flue gases coming out of the boiler and going to the economizer. The objective was to predict the flow behavior and to analyze the even distribution of flow through the gas duct. Vanes were used for even distribution. Totally four cases were tried out consisting of a base case without the guide vanes and three cases with guide vanes. Duct model was modified by modeling number of guide vanes and angles to get distributed flow of hot gases inside the duct. The fourth case having guide vanes at all places gave the best result.

Manuscript published on January 30, 2020.

* Correspondence Author

Vaibhav N. Deshmukh*, Department of Mechanical Engineering, DVK MIT World Peace University, Pune, India.

E-mail: vndeshmukh@outlook.com

Sagar A. Sirsat, Department of Mechanical Engineering, DVK MIT World Peace University, Pune, India. E-mail: sagarapatil.sap@gmail.com

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an [open access](http://creativecommons.org/licenses/by-nc-nd/4.0/) article under the CC-BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>)

Simulation of Mass Flow Rate of Air through Rectangular Split Duct

Laszlo Czetany et al. [3] developed a simple 1D theoretical model useful in designing ventilation ducts that are capable of providing uniform air distribution at the outlets. Detailed analysis was carried out for rectangular ducts with constant height and variable width. Optimal geometry was described by the width profile.

The non-linear differential equation derived from the 1D model was solved numerically. The influence of different dimensionless parameters on optimal geometry was investigated. The model was validated with experiments and gave acceptable accuracy.

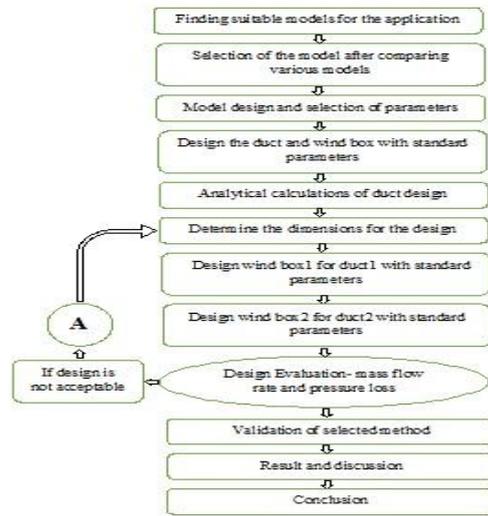
Bhavesh K. Patel et al. [4] carried out CFD analysis of mini channel heat exchanger. This compact heat exchanger can be used for electronic devices cooling. CFD analysis was carried out for parallel and counter flow mini channel heat exchanger for constant inlet temperature at hot and cold side. Results obtained from simulation indicated that by varying mass flow rate on hot and cold side, heat transfer obtained from counter flow heat exchanger was higher than that obtained from parallel flow.

Vatsal Brahmabhatt et al. [5] carried out CFD analysis of carburetor of SI engine for different positions of aerodynamic shape throttle valve and fuel nozzle angle. The objective behind carrying out CFD analysis was to find the pressure drop and velocity profile for different aerodynamic shape throttle valve angle and at best fuel discharge nozzle angle. It was found that the pressure decreased with the increase in opening of the throttle plate. Also, the pressure at the throat decreased with the increase in opening of the throttle plate as flow of the air from the float chamber into the throat increased. The Velocity profile was uniform in aerodynamic shape compared with the existing flat plate throttle valve. Mixing of air fuel was also uniform, reducing unburnt fuel increasing the efficiency of the carburetor.

Petrica Iancu et al. [6] focused on reducing NO_x emissions by presence of steam. The presence of steam decreases the peak temperature and greatly affects the reaction mechanism, decreasing the amount of NO_x produced. The introduction of steam in the combustion chamber together with the fuel and air is simple and cheap option to decrease NO_x emissions. CFD analysis showed that addition of steam to the fuel is more intensive than adding steam to combustion air. When 25% steam was added in fuel, the quantity of NO_x reduced to almost half in comparison with the dry case.

III. METHODOLOGY

The following flowchart shows the methodology followed to carry out the simulation work.



IV. MASS FLOW RATE MEASUREMENT PRINCIPLE

There are three basic Fluid Mechanics principles that govern the measurement of air mass flow rate. The first principle is the energy balance equation, which conveys the relationship between pressure, density, velocity and elevation of the flow measurement condition. The physical form of the energy balance equation between any two points is given by;

$$P_1 + (1/2) \rho (V_1)^2 + \gamma z_1 = P_2 + (1/2) \rho (V_2)^2 + \gamma z_2 + h_{loss} \quad (1)$$

The energy balance equation applies to any points with flow conditions on a streamline in a steady, inviscid flow. This equation states that the total effect of pressure, density, velocity, specific weight and elevation of a fluid flow in a system is constant in every measurement points, with consideration of the head loss. The constant total effect of all factors of flow conditions confirms that an increase in one factor will cause a decrease in the others. Conversely, a decrease in one factor will cause an increase in the others. The air mass flow rate measurement uses this basic understanding as a foundation for all the calculation process. Since the mass flow rate is derived from calculation of flow velocity, the relationship between pressure and velocity from the energy balance equation governs the calculation of mass flow rate.

The use of the energy balance equation to relate flow pressure and flow velocity must be incorporated with the conservation of mass principle as mentioned in the following equation;

$$(\rho A V)_1 = (\rho A V)_2 \quad (2)$$

This basic principle states that the mass flow rate is constant at every measurement point within the flow. The mass flow rate is calculated using the following equation;

$$m = \rho A V \quad (3)$$

Equation (2) states that the mass flow rate is obtained as a product of flow density, cross sectional area of the flow and the flow velocity.

This principle reaffirms that the total effect of flow velocity and cross sectional area of the flow is constant at every point. Reducing the cross sectional area of the flow will increase the flow velocity. Conversely, an increase in the cross sectional area of the flow will reduce the flow velocity as illustrated in equation (2). The equation for the conservation of mass (also known as the continuity equation) allows the regulation of flow velocity by modifying the cross sectional area of the flow.

V. CFD MODELING AND SETUP

One of the Views of rectangular duct can be seen in Figure 1. The duct has a rectangular cross section adjacent to the blower and the wind box. A geometrically 3-dimensional model of that duct was assembled to capture the mass flow rate in the system. Three dimensional incompressible steady flow computations were carried out using the commercially available ANSYS-Fluent. Figure 1 shows the 3D model of rectangular duct.

The air is supplied at the ambient temperature by FD fan with mass flow rate, the main reason for this analysis was to find out the distribution of the mass flow rate at two outlets. The requirement was the equal distribution of mass flow rate of air at the two outlets of the duct which are also shown in Figure 1. CFD analysis consists of three main steps;

1. Pre-processor
2. Solver
3. Post-processor

In the pre-processing, the inputs of our problem were given to the software which subsequently converted these inputs into a form suitable for use of the solver. Initially the geometry of the duct was created and after that grid generation of the geometry was created. The fluid properties were also defined in the pre-processing.

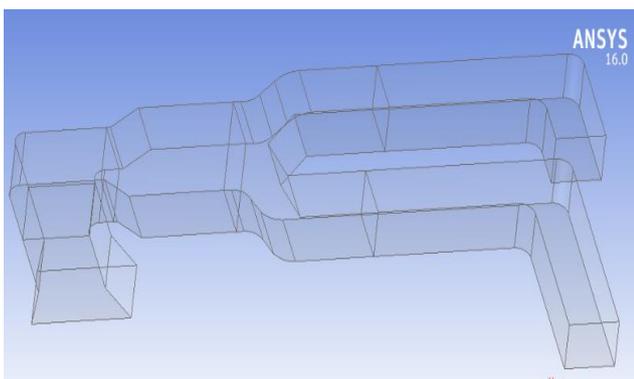


Figure 1: Computational model of rectangular duct

In solver, the option of general model setting in ANSYS Fluent showed the mesh quality and also time option was selected as steady state with velocity formulation absolute. In the solver the cell zone and boundary conditions were mentioned with the solution method SIMPLE. Before starting the CFD simulation, ANSYS needs the initial value for solution field. These values were generated by selecting the option of standard initialization. After this the number of

iterations were given and final run calculation option was selected.

A. Inlet condition

The boundary conditions for the inlet were mass flow rate along with temperature value. The air provided by the FD fan was at 321 K with different mass flow rates. The direction specification method was normal to the boundary to analyse the dedicated duct and wind box of boiler for the required parameters.

Maximum temperature of air at inlet of the duct was 321 K. Table 1 shows the four different conditions that were considered for analysis.

Table 1: Different cases under control volume

Case No.	Inlet mass flow rate (kg/s)	Max. Temperature of air at inlet of duct (k)
1	20	321
2	30	321
3	40	321
4	50	321

B. Outlet condition

The outlet boundary condition was “Pressure outlet” and zero gauge pressure was specified as a constant value in the boundary condition for the outlet. The back flow temperature was also specified as outlet condition.

C. Meshing of the computational model

The mesh was generated. Meshing breaks solid object into finite number of points which are connected with each other, and makes one complete solid. So when we run an analysis, software puts equations at one point, solves it and gets its results at that point. After that it considers those results as an initial inputs for solving next equations at next point. Mesh diagram of rectangular duct is shown in Figure 2.

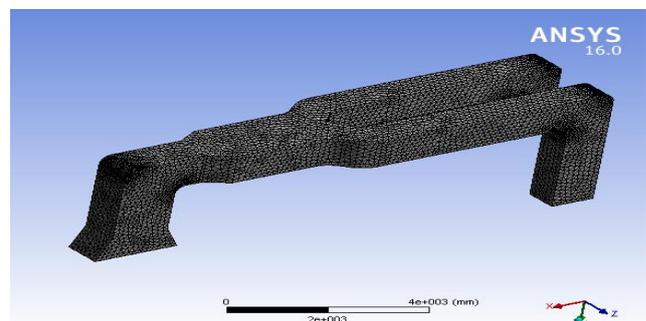


Figure 2: Mesh diagram of rectangular duct

Mesh matrix consists of

- 1) Mesh quality
- 2) Skewness
- 3) Orthogonal quality

Simulation of Mass Flow Rate of Air through Rectangular Split Duct

1. Mesh quality

Table 2 shows the level of acceptability of the mesh quality and Figure 3 shows the obtained result of mesh quality.

Table 2: Element quality metric

0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.98	0.98-1
Excellent	Very Good	Good	Acceptable	Bad	unacceptable

Statistics	
Nodes	73463
Elements	402824
Mesh Metric: Element Quality	
Min	0.16872
Max	0.99924
Average	0.82692
Standard Deviation	0.10625

Figure 3: Element quality metric

2. Skewness

Figure 4 shows the skewness quality metric.

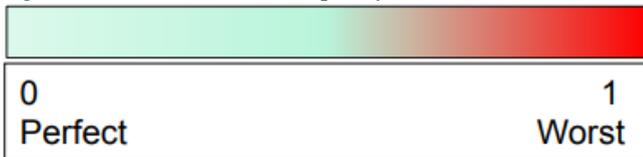


Figure 4: Skewness quality metric

Skewness is distortion in edges or angle in hexahedral and tetrahedral meshing. In case of tetrahedral meshing, the angle is 60 ° and for quadrilateral mesh it is 90 °. So skewness is the variation in angle or edges. This average skewness for given model was 0.24472.

Statistics	
Nodes	73463
Elements	402824
Mesh Metric: Skewness	
Min	1.1049e-003
Max	0.89385
Average	0.24472
Standard Deviation	0.13925

Figure 5: Skewness quality from mesh results

3. Orthogonal quality

Orthogonal quality is worst when its value is near to zero and perfect for near to one. The poor quality of mesh can affect the results. It leads to variation in results with respect to good result.

Statistics	
Nodes	73463
Elements	402824
Mesh Metric: Orthogonal Quality	
Min	0.20962
Max	0.99753
Average	0.85331
Standard Deviation	9.2202e-002

Figure 6: Orthogonal quality from mesh results

After the mesh generation, the geometry was exported as a mesh file for use in ANSYS Fluent. The ANSYS Fluent was launched from the workbench.

VI. RESULT

The set of initial conditions of numerical simulation were performed using ANSYS with air in rectangular duct along with mass flow rate and maintaining the same initial temperature in the duct.

A. Case 1

In this case, the initial mass flow rate was 20 kg/sec, which is very high and it resulted in the increased pressure at the corner of the duct as shown in Figure 9.

Table 3: Difference in outlet condition under control volume

Mass Flow Rate 20 kg/sec			
Iteration	Outlet_S	Outlet_L	Difference
5	11.19308	8.834092	2.358984
10	11.12831	8.887985	2.240321
15	11.07463	8.917583	2.157051
20	11.03079	8.964595	2.066197
25	10.9915	9.010231	1.981269
30	10.95499	9.047117	1.907878
35	10.91764	9.082369	1.835275
40	10.88338	9.114262	1.769122
45	10.85451	9.143467	1.711039
50	10.82778	9.171436	1.656348
Average	10.98566	9.017314	1.968348

The above table shows the mass flow rate at outlet positions with respect to inlet mass flow rate in control volume.



The average difference between the outlet_S and outlet_L was 1.968348.

40	16.32639	13.67032	2.65606553
45	16.28383	13.71636	2.56746346
50	16.24406	13.75797	2.48609668
Average	16.48099	13.52676	2.95422727

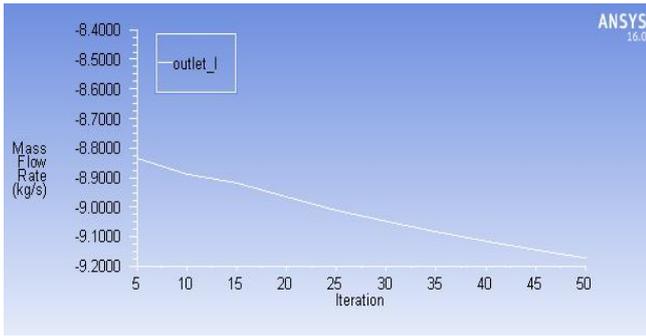


Figure 7: Mass Flow Rate vs. Iteration at Outlet_L

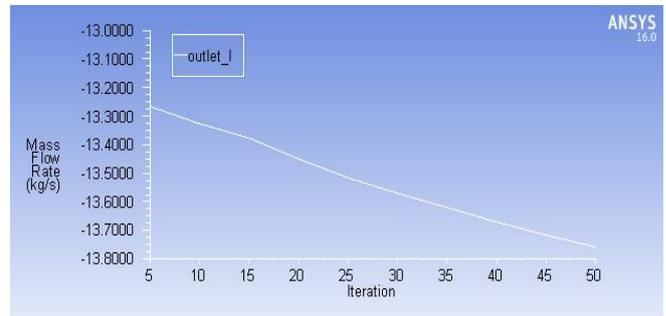


Figure 10: Mass Flow Rate vs. Iteration at Outlet_I

The negative sign indicates that mass leaving the system with respect to the inlet condition.

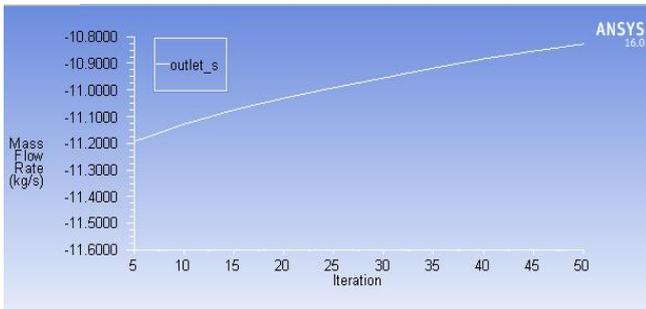


Figure 8: Mass Flow Rate vs. Iteration at Outlet_S

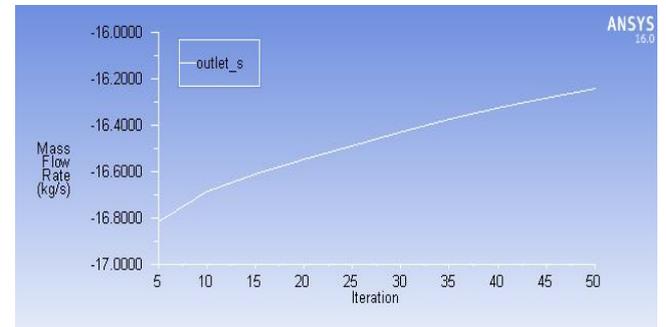


Figure 11: Mass Flow Rate vs. Iteration at Outlet_S

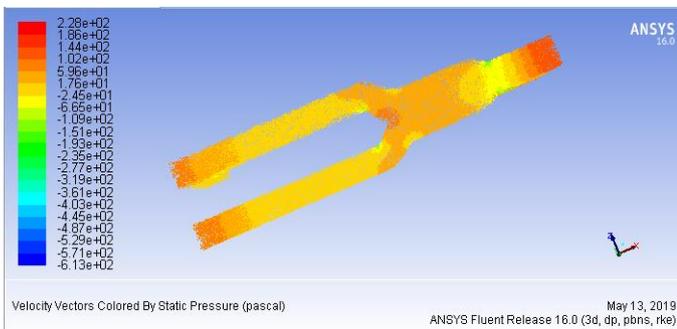


Figure 9: Velocity Vector colored by Static pressure

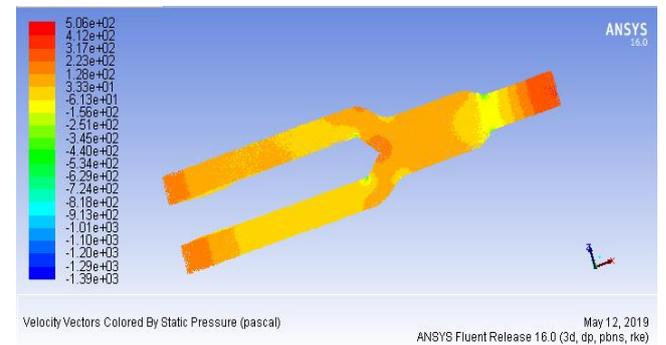


Figure 12: Velocity Vector colored by Static pressure

B. Case 2

Table 5: Difference in outlet condition under control volume

Mass Flow Rate 30 kg/sec			
Iteration	Outlet_s	Outlet_L	Difference
5	16.81636	13.26571	3.55065004
10	16.68619	13.32628	3.35991113
15	16.61095	13.37634	3.23461012
20	16.54761	13.44925	3.09835935
25	16.48928	13.51607	2.97320391
30	16.43015	13.5699	2.86024884
35	16.37505	13.61938	2.7556636

C. Case 3

Table 5: Difference in outlet condition under control volume

Mass Flow Rate 40 kg/sec			
Iteration	Outlet_S	Outlet_L	Difference
5	22.40843	17.65491	4.753517
10	22.25128	17.77837	4.472914
15	22.14711	17.83399	4.313122

Simulation of Mass Flow Rate of Air through Rectangular Split Duct

20	22.06248	17.92918	4.1333
25	21.98524	18.02316	3.962089
30	21.9093	18.09632	3.812978
35	21.83319	18.15905	3.674135
40	21.76685	18.22177	3.545073
45	21.71044	18.28376	3.426681
50	21.66072	18.34601	3.314711
Average	21.9735	18.03265	3.940852

Table 6: Difference in outlet condition under control volume

Mass Flow Rate 50 kg/sec			
Iteration	Outlet_S	Outlet_L	Difference
5	28.0066	22.09036	5.9162393
10	27.81174	22.20863	5.60311072
15	27.68356	22.28976	5.39380001
20	27.57663	22.41292	5.16370636
25	27.48203	22.52774	4.95429025
30	27.38732	22.61827	4.76904493
35	27.29396	22.69863	4.59533591
40	27.21238	22.782	4.43038594
45	27.13929	22.86086	4.27843436
50	27.07365	22.92945	4.14419645
Average	27.46672	22.54186	4.92485442

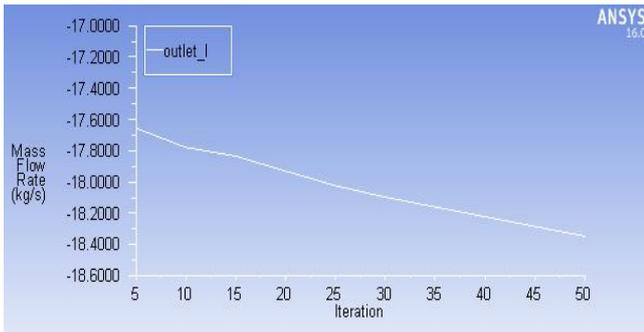


Figure 13: Mass Flow Rate vs. Iteration at Outlet_L

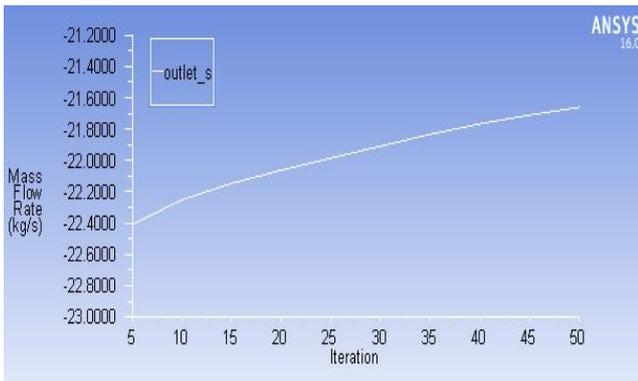


Figure 14: Mass Flow Rate vs. Iteration at Outlet_S

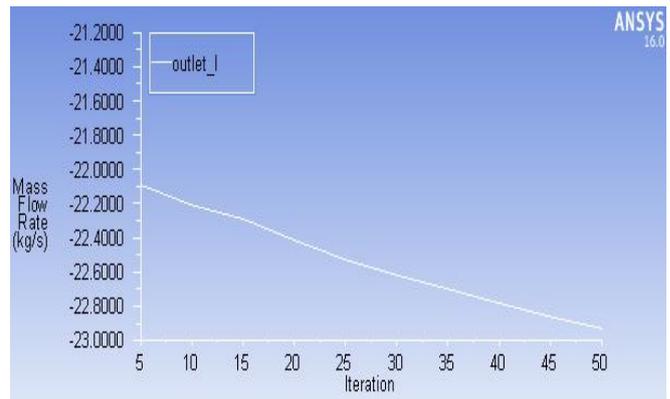


Figure 16: Mass Flow Rate vs. Iteration at Outlet_L

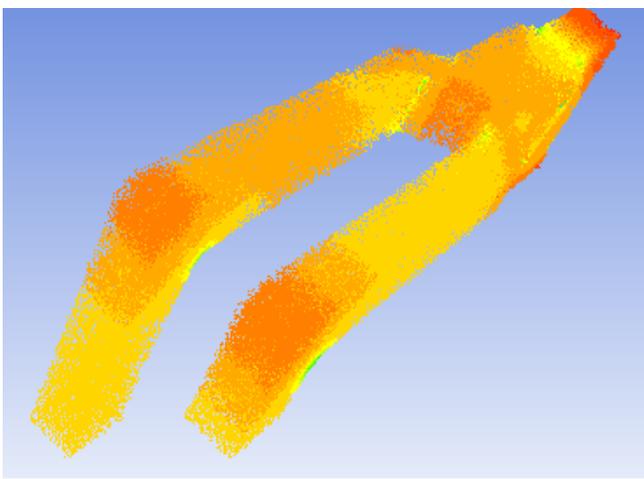


Figure 15: Velocity Vector colored by Static pressure

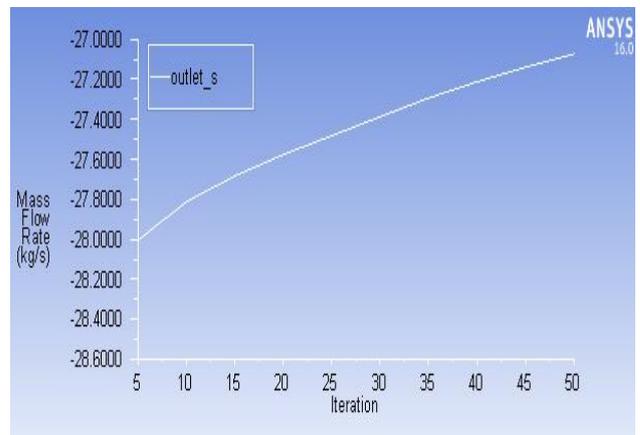


Figure 17: Mass Flow Rate vs. Iteration at Outlet_S

D. Case 4

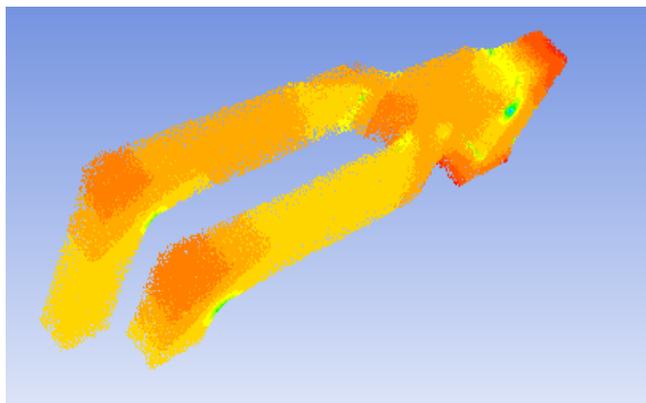


Figure 18: Velocity Vector colored by Static pressure

E. After consolidation of all results

Table 7: Difference in outlet condition under control volume w.r.t. inlet

Mass Flow Rate	Damping required at outlet_S
20	1.9683484
30	2.9542273
40	3.940852
50	4.9248544



Figure 19: Damping required w.r.t flow rate in control volume

The above chart shows the mass flow rate at inlet and difference at outlet positions. The difference shows the relation between the mass flow rate change and the damping required at each outlet. In this study, the gradual increase in the mass flow rate resulted in the gradual increase in difference of mass flow rate at the outlet.

V. CONCLUSIONS

In this CFD analysis of industrial duct, the mass flow rate at outlet_L was 16.15 kg/sec and the mass flow rate of outlet_S was 15.90 kg/sec. The difference between outlet_L and outlet_S was 0.25 kg/sec. The mass flow rate of the outlet_L was higher by 1.59% than the mass flow rate at the outlet_S.

It means that the mass flow distribution through both the outlets was not equal. If we want equal mass flow rate at both the outlets of the rectangular duct then the duct with extra length needs some damping to minimize the flow rate through it.

Thus, computational investigation can effectively help to decide the damping requirements for the industrial ducts which can result in proper design of any flow system in primary stages of projects. Different mass flow rates also affect the damping requirement of industrial ducts which can also be predicted using CFD analysis. The relation between different mass flow rates at inlet of duct and the damping requirement can also be found by using CFD.

REFERENCES

1. Piotr Ostrowski, Leszek Remiorz “Measurement of gas flow in short ducts, also rectangular”, Flow Measurement and Instrumentation, vol. 30, April 2013, pp. 1-9.
2. P.Devakumaran, C.Balakrishnan, S.Chandran, C.Azhagurajan, B.Veluchamy, M.Vivekanandan, “Attainment of fully developed flow in air distribution duct of boiler using CFD analysis” IJARIE-ISSN(O)-2395-4396, vol. 4, Issue 2, 2018, pp. 1021-1025.
3. Laszlo Czetany, Zoltan Szantho, Peter Lang, “Rectangular Supply Ducts with Varying Cross Section Providing Uniform Air Distribution”, Applied Thermal Engineering, vol. 115, 2017, pp. 141-151.
4. Bhavesh K. Patel, Ravi S. Engineer, Mehulkumar H. Tandel, “CFD Analysis Of Mini Channel Heat Exchanger Using Water as a Working Fluid”, IJARIE-ISSN(O)-2395-4396, vol. 3, Issue 3, 2017, pp. 422-429.
5. Vatsal Brahmabhatt, Dr. Pravin P. Rathod, “CFD Analysis of Carburettor for Different Positions of Aerodynamic Shape Throttle Valve and Fuel Nozzle Angle”, IJARIE-ISSN(O)-2395-4396, vol. 3, Issue 2, 2017, pp. 5017-5025.
6. Petrica Iancua, Salvador Vilas-Bonafoux, Jose M. Iglesias Fernandezb, Valentin Jordi Bonetb, Alexandra E. Bonet Ruizb and Joan Llorens, “Computational Fluid Dynamics (CFD) Simulation of Fuel Gas and Steam Mixtures to Decrease NOx Emissions of Industrial Burners”, 27th European Symposium on Computer Aided Process Engineering – ESCAPE 27, 2017, pp. 565-570.

AUTHORS PROFILE



Vaibhav N. Deshmukh is PhD in Mechanical Engineering. He is working as an Assistant Professor in School of Mechanical Engineering at Dr. Vishwanath Karad MIT World Peace University, Pune. He has 16 years of teaching experience and 2 years of industry experience. He teaches subjects like Heat Transfer, Fluid Mechanics. His research area is Nanofluid Heat Transfer and its applications in heat exchangers. He has five publications with his name. His areas of interest are Biomedical Nanotechnology and Biomicrofluidics.



Sagar A. Sirsat has done his M. Tech in Heat Power Engineering from School of Mechanical Engineering at Dr. Vishwanath Karad MIT World Peace University, Pune. He is currently working as Graduate Trainee Engineer at Thermax Ltd., Pune. He has done his graduation in Mechanical Engineering.