Research Article   ISSN: 2394 - 658X

Computational Fluid Dynamics (CFD) Simulation in Air Duct Channels Using STAR CCM+

AE Ikpe, EK Orhorhoro and OG Ogiemudia

[1]Department of Mechanical Engineering, Coventry University, UK
[2]DepartmentofMechanicalEngineering, University of Benin, Edo State, Nigeria
ikpeaniekan@gmail.com

_____

ABSTRACT

Pressure loss in air duct channels is caused by friction between the fluid and the walls of the pipe conveying liquid or gaseous product from one location to another. In recent times, a number of tools have developed for analysing this relationship but STAR CCM+ was adopted in this study for analysing the pressure losses at predetermined velocities. The velocities used were 5m/s, 10m/s, 20m/s and 40m/s respectively and the pressure losses at each simulation cycle where 5.94, 21.91, 81.58 and 285.66Pa according to the set velocities. Hence, it was observed that the pressure losses increased as the velocity and inlet pressure increased.

Keywords: Pressure loss, Air duct, Fluid, Velocity, Mass flowrate

_____

INTRODUCTION

Transportation of different products in both liquid and gaseous form is a major activity that occurs in our industries on daily basis ranging from the oil industry, brewery industry, power sector, institution etc. With proper flow analysis and control, major disasters, downtime, losses and decrease in profits in the industry can be prevented. Therefore, it is imperative to select the best possible materials during the installation and operation process of air duct channels, as the type of duct fittings used play a vital role in the overall system performance [1, 6]. In this study, Computational Fluid Dynamics simulation in Air Duct Channels was carried out using STAR CCM+ to analyse the pressure losses at predetermined velocities. The K-epsilon turbulence model is one of the common turbulence models used by Star CCM+ in resolving turbulent flow and has been applied in the flow simulation carried out in this study. The model is recommended for use in flow applications that do not involve heat transfer [2, 4, 9]. The letter 'k' is the turbulent kinetic energy while ε is the rate of dispersion of the turbulent energy. K- Epsilon model resolves turbulence by finding the amount of kinetic energy per unit mass present in the turbulent fluctuations [3, 8]. Table -1 shows the Star CCM+ Parameters used for the analysis in this study.

METHODOLOGY

The air flow in a given duct was analysed to determine flow parameters and characteristics. The analysis was done in 2D using CFD Package-Star CCM+ software. In general, flow in a two dimensional plane is considered as a special case of a 3D if the geometry is symmetrical in one coordinate [7]. Experiments have shown that 2D models give a very close approximation to 3D model for symmetrical model [5].It has the following steps:

- Creation of the model in 3D (Fig. 1). This could be done in star CCM+ or with CAE software and then imported to Star CCM+. Since the given model has a simple geometry, it was drawn in Star CCM+.
- The 3D model was then converted to part, follow by assigning of regions to parts (Fig. 2).
- The model was then meshed and converted to 2D as shown in Fig. 3 and 4.
- The next step was setup the physics for the simulation after which the boundary conditions were specified.
- Running of simulation and post-processing in which the result obtained was analyse. A number of iterations were done until convergence was achieved.

_____

Table -1 Star CCM+ Parameters Used for the Analysis

Parameter	Menu Properties	Selection/ Value Inputted
Mesh Selection	Mesh type	Trimmer (for Volume Mesh), Surface Re-mesher (Surface mesh), Prism Layer Mesh (For the prism layer)
	Base size	15 mm
	Prism Layer thickness	Equal to the Boundary layer thickness for the given velocity
	Number of layers	20
	Prism layer stretching	1
	Space	2D flow
Physics Selection	Time	Steady
	Material	Gas
	Flow	Segregated flow
	Equation of State	Constant density
	Viscous Regime	Turbulent
	Reynolds-Averaged Turbulence	K-epsilon
	Inlet	Inlet Velocity
Boundary condition selection	Outlet	Outlet Pressure
	Wall	Wall
	Turbulent Intensity	10%
	Turbulence Specification	Intensity + Length scale
	Turbulent length scale	7% of the Hydraulic diameter
	Turbulent velocity scale	5% of the free steam velocity
	Temperature	293K
	Wall condition	No-slip

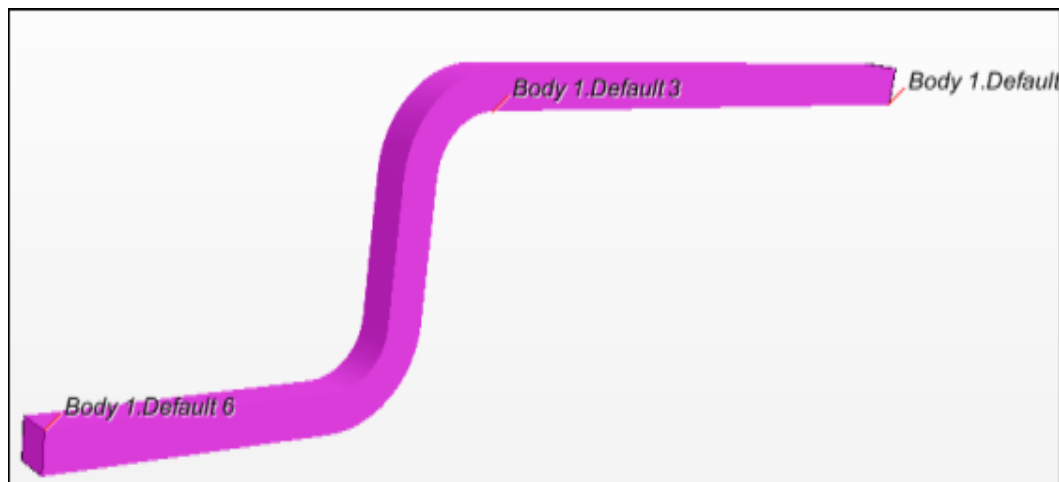**Fig .1 Creating the geometry in 3D**

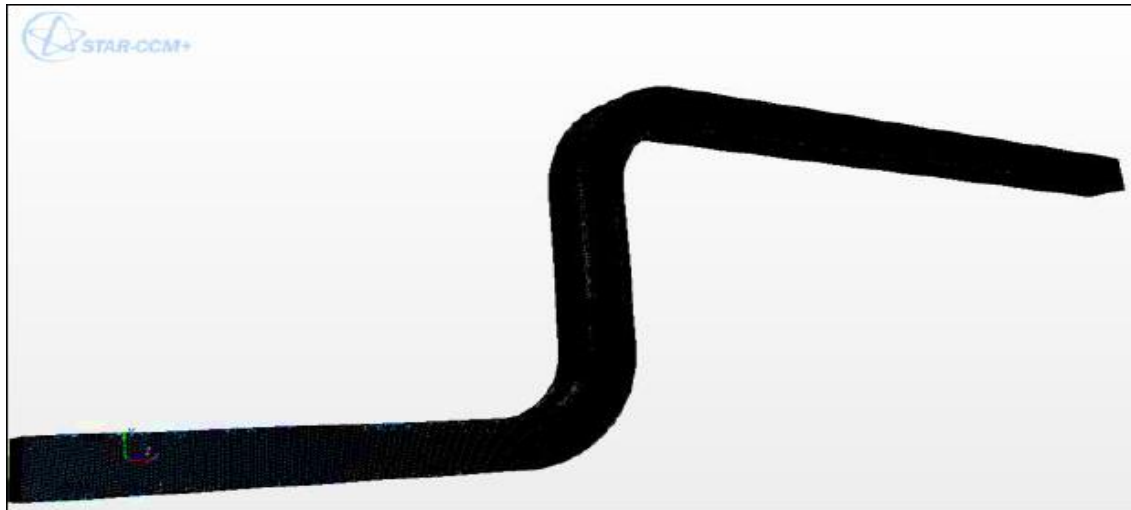**Fig. 2 Creating the geometry in 3D**

Fig.3 Creation of 3D mesh

Fig. 4Converting 3D mesh to 2D mesh

Analysis in CFD is affected by the number of grid points (cells) generated to solve the computation. The number of cell generated is a function of the mesh size. Generally, as the number of cells is increased, the results obtained become more accurate while the computational time increases also. However, as the mesh size is made finer and the number of cells increased, a point is reached when the results obtained is not or is marginally affected by the mesh size. At the point the mesh is said to have converged. The results obtained at this point are usually taken as the solution of the computation. Due to the length of time required to obtain solutions using fine mesh, initial analyses were done using coarse mesh. The mesh size was gradually refined until convergence was achieved. Another important parameter which affects the result obtained from the simulation is the number of iteration to convergence. The iteration steps were increased until the results obtained are stabilized (i.e. when the results no longer change with time).

RESULTS AND DISCUSSION

Table -2 shows the result of mesh convergence study at 20m/s. With a velocity of 20m/s, the solution was found to convergence at a mesh size of 15mm.This mesh size was therefore used to run the analysis for other velocity (5m/s, 10m/s, and 40m/s,) cases, and the following boundary conditions were obtained (Table -2).

Fig. 5 and 6 show the plot of total pressure for 20m/s velocity and 40m/s velocity. Fig. 7 shows the magnified velocity vector and streamline plots for 40m/s. It can be seen that there was no separation at the bends. The same was observed for other velocities (5m/s, 10m/s, and 20m/s).Fig. 8 show that the velocities around the bends are the greatest. This can be explained by the law of conservation of mass. The bend restricts the movement of the fluid coming from the inlet. As the mass of fluid hit the restriction (i.e. the wall of the bend), the area available for the fluid to flow is reduced. Since the mass flow rate must be maintained, the velocity of the fluid passing through the bend is increased, hence maintaining continuity.

Table -2 Mesh Convergence Study at 20m/s

Mesh Size (mm)	Number of Cells (2D)	Total inlet Pressure (Pa)	Inlet Static pressure (Pa)	Max. Mass flow rate at the inlet (Kg/s)	Number of steps taken to stabilize
50	69633	1797.693	143.6292	9.6	2600
40	85628	371.0221	131.0221	9.6	2800
20	105498	399.6862	159.6862	9.6	3000
15	105646	389.2385	149.2442	9.6	2200
10	105646	389.2442	149.2385	9.6	2800

Table -3 Boundary Conditions Obtained after the Analysis for the Respective Velocities

Inlet Velocity (m/s)	Outlet Velocity (m/s)	Mass Flow Rate(Inlet) (kg/s)	Mass Flow rate Inlet) (kg/s)	Static Inlet Pressure (Pa)	Static Outlet Pressure (Pa)	Total Inlet pressure (pa)	Total Outlet Pressure (pa)
5	5.32	2.4	2.4	5.93	0	20.94	17.03
10	10.54	4.8	4.8	21.90	0	81.91	66.67
20	20.91	9.6	9.6	81.58	0	321.58	263.87
40	41.40	1.92	1.92	285.66	0	1245.66	1041.12

Table -4 Pressure Loss between the Inlet and Outlet

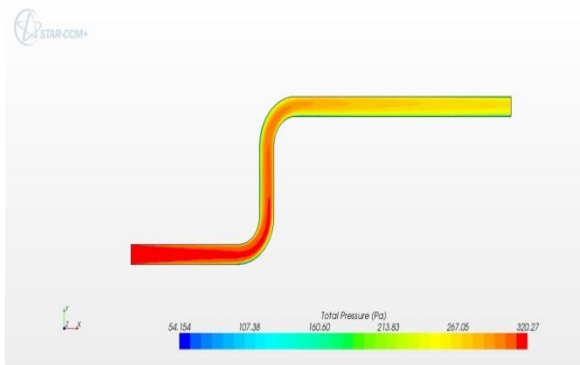| Case	Velocity (m/s)	Inlet Pressure (P_1) (Pa)	Outlet Pressure (P_2) (Pa)	Pressure Loss (P_2-P_1) (Pa)
1	5	5.94	0	5.94
2	10	21.91	0	21.91
3	20	81.58	0	81.58
4	40	285.66	0	285.66

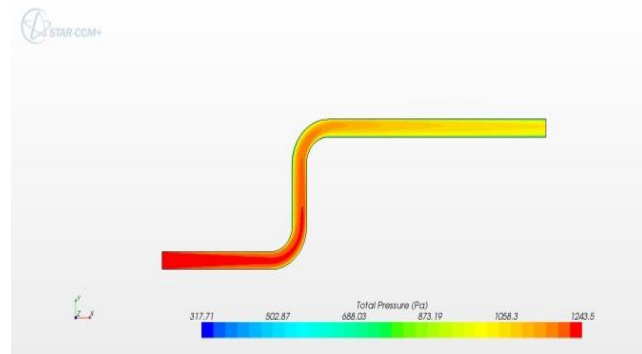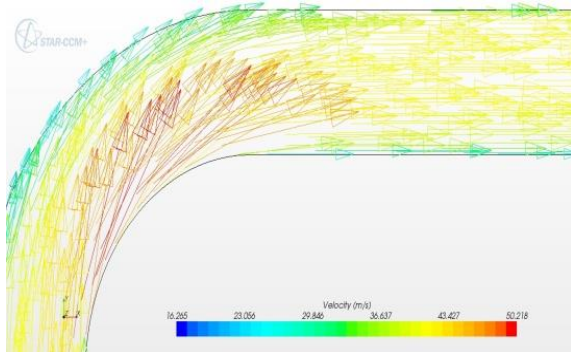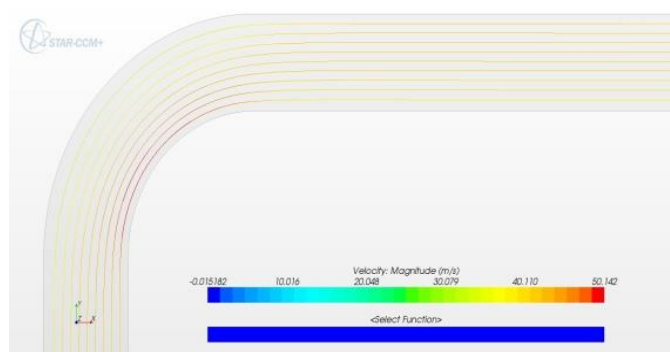**Fig. 5 Plot of total pressure for 20mls**

Fig. 6 Plot of total pressure for 40m/s

(a)

(b)

Fig. 7 Magnified vector plot scene (a) and streamline plot (b) at one of the bends showing that no separation occurred at 40m/s

Since the fluid is viscous, the velocity at the wall is zero due to the non-slip condition. Due to viscous effect too, the fluids closest to the layers in direct contact with the wall have velocities which are far much less than the velocity of the fluid (their velocity are nearer to the zero velocity at the wall). Because of this less velocity, a laminar sub-layer is created near the wall as shown in Fig. 9. The flow in the remainder of the duct is turbulent.

The pressure loss between the inlet and outlet is the difference in pressure between the inlet pressure and the outlet pressure. The static values of pressure have been used in finding the loss (Table -4). It can be seen that pressures increases as velocity increase. The increase is exponential because the velocity term in the pressure equation is squared. Also, it can be seen that the error decreases as velocity is increased from 5m/s to 40m/s.
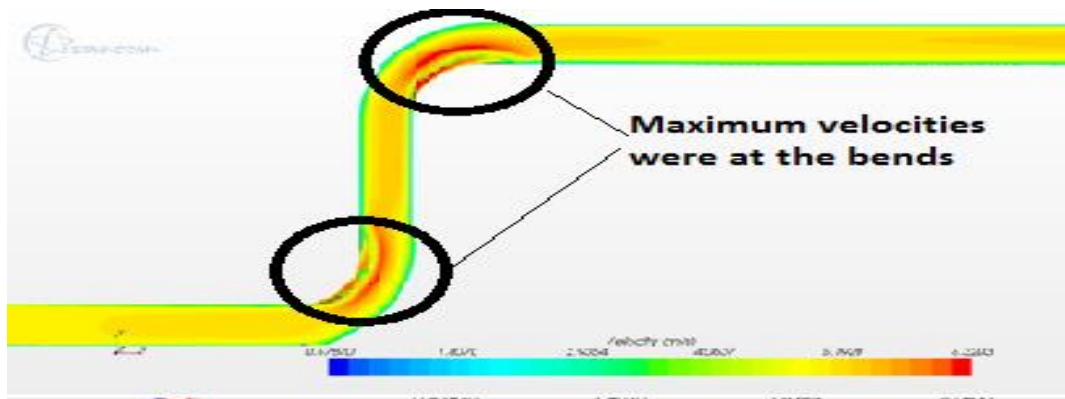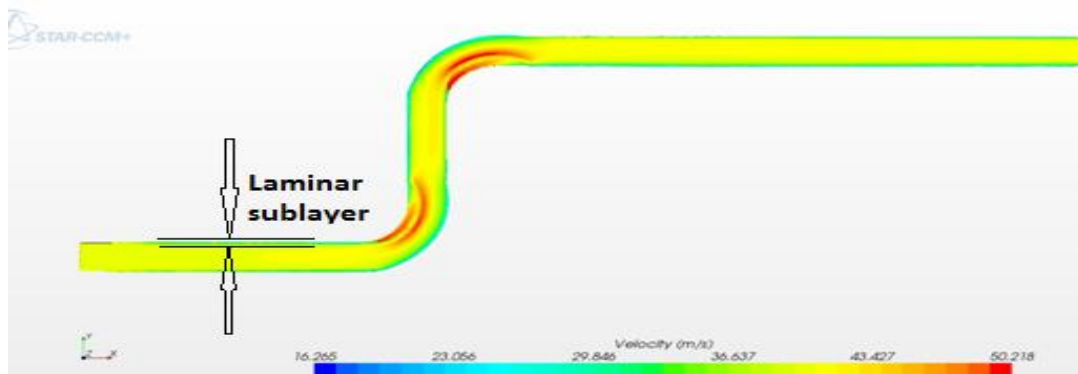
Fig. 8 Velocities at the bends at 5m/s

Fig. 9 Laminar sub-layer at 40m/s

CONCLUSION

In this study, CFD Package-Star CCM+ software was used to analyse flow through an air duct. The results obtained from the simulation showed that the software was able to simulate and determine certain flow behaviour in air duct channels. The outcome of the results obtained can be used to predict the velocity, inlet pressure, pressure loss, mass flow rate, mesh size, etc. in industrial applications. Although pressure losses and other important parameters can also be determined using hand calculation, simulation using CFD tools such as Star CCM+ may serve time, energy and resources while generating minimal possible errors which oftentimes is due to the input parameters and boundary conditions selected.

REFERENCE

[1] B Abushakra, D Dickerhoff, I Walker and M Sherman, *Laboratory Study of Pressure Losses in Residential Air Distribution Systems*, Lawrence Berkeley National Laboratory Report LBNL-49293, Berkeley, CA, **2002**.

[2] CD Adapco, *STAR-CCM+ Version 8.04*, <http://cumoodle.coventry.ac.uk/course/view.php?id=15595>, **2014**.

[3] LS Douglas, Wall Functions for the k - [epsilon] Turbulence Model in Generalized Non-Orthogonal Curvilinear Coordinates, *Iowa State University, Retrospective Theses and Dissertations,* Paper 9954, **1992.**

[4] Y Cengel, J Cimbala, R Turner and M Kanoglu, *Thermo-Fluid Sciences*, New York, McGraw-Hill, **2012**.

[5] K Ekambara, T Mahesh and B Jyeshtharaj, CFD Simulations of Bubble Column Reactors: 1D, 2D and 3D Approach, *Chemical Engineering Science*, **2005,** 60, 6733-6746.

[6] W Janna, *Introduction to Fluid Mechanics*, 3[rd]Edition, Boston: PWS Publishing Company**1993**.

[7] T Jiyuan, H Guan and C Liu, *Computational Fluid Dynamics: Principles and Applications*, Elsevier Limited, Amsterdam, **2005**.

[8] C Scott-Pomerantz, *The k-Epsilon Model in the Theory of Turbulence*, University of Pittsburgh, Pennsylvania USA, **2004**.

[9] H Versteeg and W Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2[nd]Edition, Pearson Education Limited, **2007**.