International Conference on Ideas, Impact and Innovation in Mechanical Engineering (ICIIIME 2017) ISSN: 2321-8169
Volume: 5 Issue: 6 1183 – 1191

CFD Analysis of Air Flow Distribution over Hot Rolled Coils

S.H.Bobade[1], A.V.Deshpande[2]

[1]*Assistant Professor, Department of Mechanical Engineering, Smt.Kashibai Navale College of Engineering, surajbobade90@gmail.com*
[2] *Assistant Professor, Department of Mechanical Engineering, Veermata Jijabai Technological Institute, avdeshpande@vjti.org.in*

ABSTRACT

This paper analyzes the Computational Fluid Dynamics (CFD) modelling of air flow distribution from an axial fan over coils of hot rolled sheet. To avoid blade complicated geometry, an approach of body force model used rather than multi reference frame model. In body force model, the input pressure-air flow relationship is taken from fan curve provided by fan manufacture. The Dittus boelter equation used to determine the convection heat transfer coefficient for cooling coils of hot rolled sheets. Results are obtained for three parameters, viz: effect of position of a baffle plate to increase heat transfer rate, effect of spacing of adjacent coils of hot rolled sheets to increase heat transfer rate, effect of using baffle plate at end of the last coil to increase heat transfer rate. At the end, the streamlines, obtained from fan model is verified with experimental data which shows good agreement with each other.

Keywords*: Computational fluid dynamics (CFD), hot rolled sheet, axial fan, fan curve, Streamlines etc.*

Introduction

Axial flow fan most extensively used in many engineering application. Its adaptability has resulted in implementing into industry. The process of Open Coil Annealing (OCA) is used to prepare different metal sheet for further processing. After annealing a coil is then removed from the furnaces and placed on bed. Cooling at this point is down by force convection using axial flow fan.

The benefits using axial flow fan for a purpose of augmenting heat transfer is particularly in cooling application. The extended use of axial flow fan for a fluid movement and heat transfer has result in detailed research into performance attribute. In some industry, more practical example cooling of hot rolled coil using axial flow fan receive more importance in understanding flow characteristics and heat transfer.

A fan draws air down from first coil axial through the last of coil. Fan is placed in front of coils which kept one after the other on a ground of small support steel. The first coil usually cools fast as compare to other coils because it gets more velocity over it. Development of model results in increased heat removal rate through a hot rolled coil. The model is used to investigate the velocity distribution in the coil so that better understanding of coil behaviour can be attained.

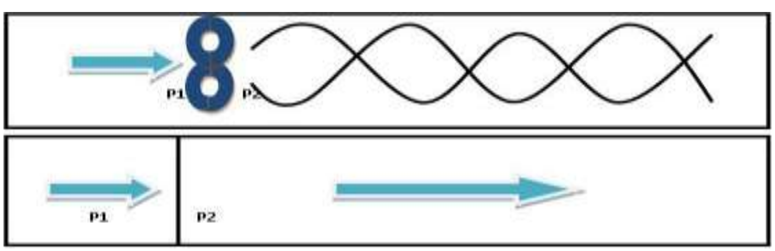


Figure 1: Schematic of pressure drop

CFD has expensive capability and extensive development in recent year. CFD can model the fan and predict air flow through it. The approach used body force model in CFD for fan model. In body force model, fan cannot model but fan characteristics used on fan surface. In fan characteristics, relationship between pressure-flow rate provided by manufacture which is used. The model provides accurate prediction over a range of operating condition. The blades of the fan are not modelled as shown in Figure.1 The advantage of this type of modelling is that it saves a lot of time and computational power but the drawback is the swirl which cannot be modelled with lumped parameter modelling.

2. Literature review

Ligong Yang [1] developed a fan model in fluent for predicting air flow through fan. The results obtained from fluent fan model have good agreement with experimental data. Louis Lowrence [2] developed computer model which analyze cooling of coil using fan and providing a information related to air drawn from fan to cool coils in minimum time. Tobias Berg, Anna Wikstrom [3] used two CFD approach for fan model. First one is body force model in which fan cannot model but fan characteristics applied in fan surface. In second approach, multi reference frame model (MRF) fan model in rotating reference frame and other domain maintain as a stationary. MRF showing good agreement of results. Morris et al [4] used a hot wire to measure axial, radial and tangential velocity components in wake of an automotive cooling fan. The volume flow rate through the system was measured using a unique moment of momentum flux. Nourse [5] developed a fan test facility and procedure to measure the detailed velocity field downstream of the fan using similar to Morris.

3. Problem setup

The present study on air flow distribution over hot rolled coil shown in figure 4, which consists of axial flow fan, hot rolled coil and baffle plate shown in respective figure 2 and figure 3. The fan and fan motor is mounted on shroud. Hot rolled coils mounted on iron base support which is kept on ground and baffle plate which kept at end of last coil. The CFD model for this

Figure 2: Schematic of Axial fan

Figure 3: Hot rolled coils

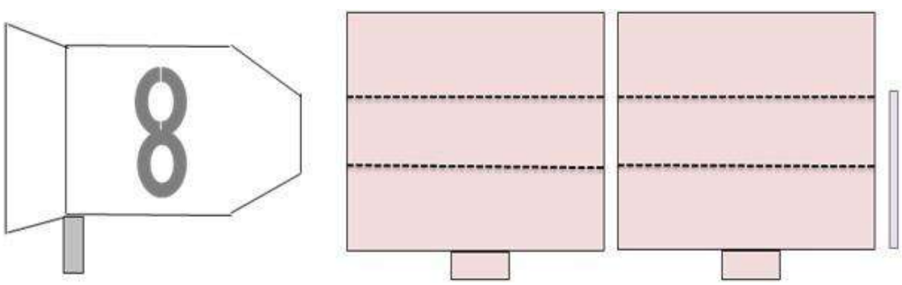


Figure 4: Fan with hot rolled coils

study is built using commercially available software. The code solves the Reynolds average Navier Stokes equations. The effect of turbulence modelled using k-epsilon model. To make the simulation timely economical, scalable wall functions are used to resolve the wall flows. The simulation is believed to be converged when maximum residuals are reducing to 1.0e-04. The base line results obtained from CFD analysis are compared with experimental result.

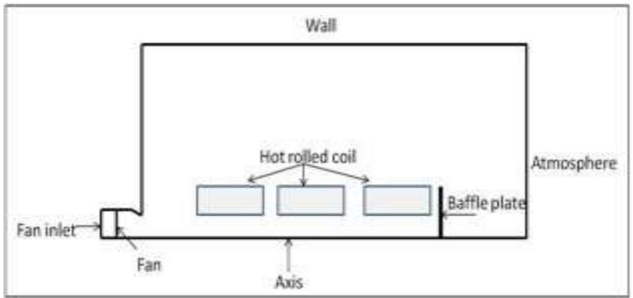
4. Numerical CFD model

4.1 Computational domain

Figure 5 shows the geometric construction details of air flow distribution over hot rolled coil. The coils with symmetry domain are chosen for present numerical study. Due to symmetry of the geometry, the computational domain is confined. Normally hot rolled coils placed in atmosphere to cool by fan with force convection. To consider CFD point view, it should consider an imaginary domain in all sides of coils. Domain should maintain large for accurate results. The length, breath and height of the computational domain are 12m, 1.5m and 6 m respectively. To minimize the error due to flow oscillation and flow reversing effect which might get induced

ISSN: 2321-8169
1183 – 1191

numerically, the computational domain is extended in stream wise direction of air. Additional domain consider over a hot rolled coil.



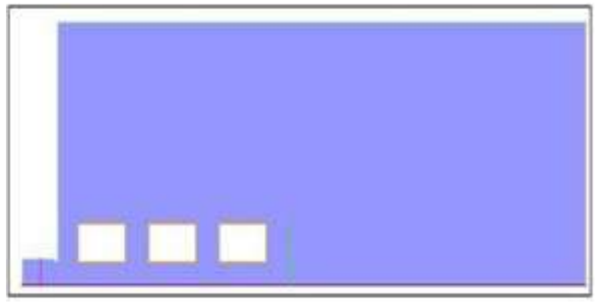


Figure 5: Computational domain **Figure 6: Descritization**

4.2 Descritization

To convenient for software, geometry created by grid matches with grid interface. Geometry should divide into number of volume for hexagonal meshing. OpenFOAM software used for pre-processing to model the geometry. More grid points generated in path of flow of air so as to get accurate results of velocity contour. Coarser grid created on a surface which is away from coil and coil faces does not mesh, shown in figure 6.

4.3 Boundary condition

Fluid in the domain is air. The compressibility effects are ignored due to the low speed of fan. Compressibility effect to be consider when Mach number greater than 0.3. Mach number are calculated as.

$$V_d = \frac{Q}{A}$$

We calculated velocity which is 13 m/s. So Mach number less than 0.3 so we used incompressible model.
The flow inside the domain is turbulent regardless the Reynolds number. The existence of several coils with the vortices created by the fans makes the flow regime turbulent inside the domain.

Surface	Boundary condition
Inlet	Pressure inlet(Zero static pressure)
Outlet	Pressure outlet(Zero static pressure)
Wall	No slip
Symmetry	Axis
Fan	Fan characteristics (Static pressure VS ow rate
Coils	No slip

Table 1: Boundary conditions

4.4 Fan characteristics

Fan characteristics curve is nothing but relationship between static pressure and flow rate through fan. Fan characteristics shown in figure 7 for Aerotech fan with 1440 rpm generally fan manufacture

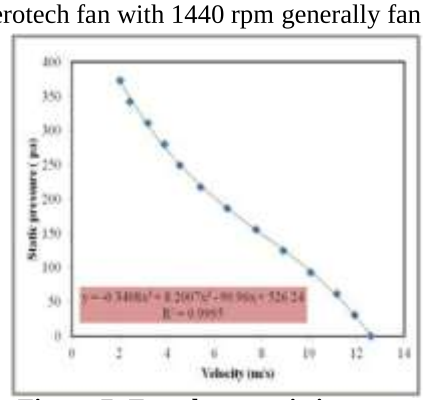


Figure 7: Fan characteristics curve

International Conference on Ideas, Impact and Innovation in Mechanical Engineering (ICIIIME 2017) ISSN: 2321-8169
Volume: 5 Issue: 6 1183 – 1191

$$P = f_1 + f_2{:}u^1 + f_3{:}u^2 + f_4{:}u^3$$

provides the fan characteristics for fan, only fan dimensions and fan rpm should be known. For 1440 rpm with fan diameter 600 mm, fan characteristics given by manufacture as below

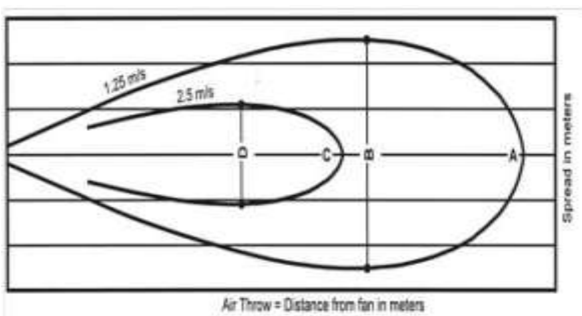
5. Results and discussion

5.1 Validation of fan case

An Aerotech one of the fan manufacturers which provide the air throws distance from fan in meters VS Spread in meters for 1440 rpm same as speed in our case. Table 2 show below which provided fan specification and figure 8 provided distance cover by fan air velocity.

Fan dia(mm)	Speed (rpm)	Motor (kW)	Free Air flow(l/s)	Fan velocity outlet(m/s)	A (m)	B (m)	C (m)	D (m)
610	1440	1.1	4320	14.8	28	8.4	19.6	3.6

Table 2: Aerotech fan specification



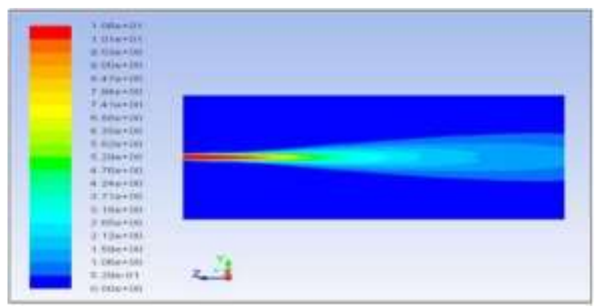


Figure 8: Stream line of velocity through Aerotech fan Figure 9: Stream line of velocity through fan by CFD

To verify results with Aerotech fan, CFD model of fan with empty domain created. Fan characteristics applied on fan surface which provide at middle at entry domain. Pressure outlet applied on outlet of domain and on rest of surfaces provide as wall boundary condition. The stream line of velocity through fan with empty domain shown in figure 9 using CFD. Result of axial distance cover air through fan using CFD is match with distance cover by air through Aerotech fan shown in Figure.10

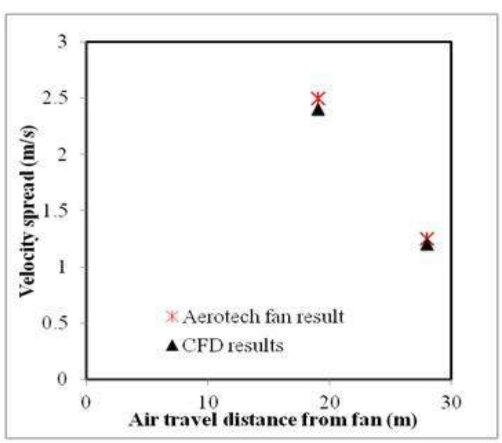


Figure 10: Comparison of fan results

5.2 Effect of end baffle plate on heat transfer rate

Baffle plate at end of coil make an importance on heat transfer rate. When there is no baffle plate at end of coil flow goes inside of hot rolled coils but velocity obtain on inner diameter to outer diameter on front face of last coil is less. Velocity obtain on front face and back face of third coil is

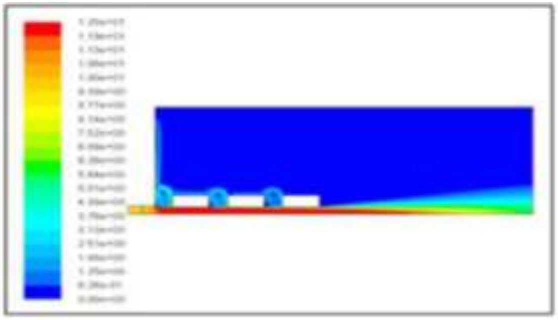


Figure 11: Velocity contour with no baffle plate at end of last coil

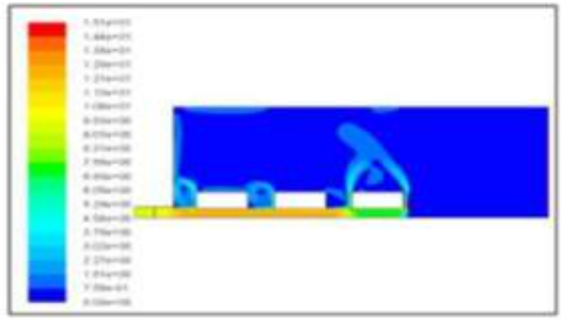


Figure 12: Velocity contour with baffle plate at end of last coil

greater in case of baffle at end of coil, shown in Figure 13 and Figure 14.So maximum heat transfer takes place when there is baffle plate at end of coil.

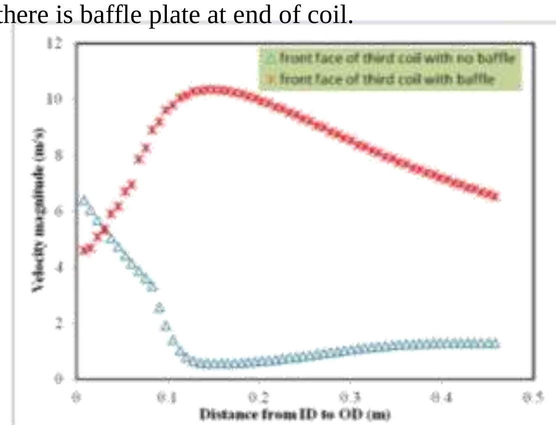


Figure 13: Velocity magnitude from ID to OD on third coil

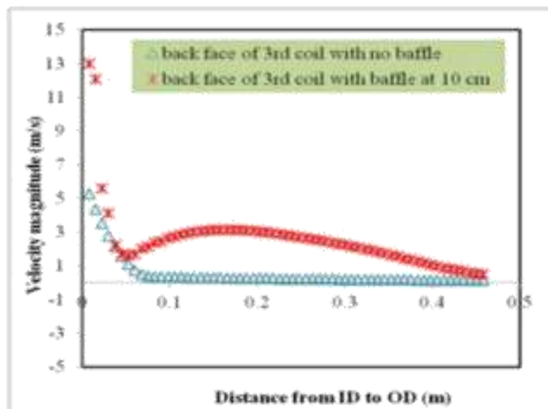


Figure 14: Velocity magnitude from ID to OD on third coil

5.3 Heat transfer coefficient inside a circular coil

Dittus boelter equation used to determine the convection heat transfer coefficient inside a circular coil which is given as below.

$$Nu = 0.023 * Re^{0.8} * Pr^{0.33}$$

$$\frac{h * d_h}{K} = 0.023 * \left(\frac{\rho * V * d_h}{\mu}\right)^{0.8} * \left(\frac{\mu * C_p}{K}\right)^{0.8}$$

From above equation it clearly seen that heat transfer coefficient directly proportional to velocity of air through fan. Rather than the velocity all another parameter remains constant because these are properties of air. So heat transfer coefficient increases with velocity of air through fan. From bulk mean velocity, heat transfer coefficient for flow inside circular coil is 48 W/(m.K).Standard value of air properties taken from data manual those are constant.

5.4 Effect of coil gap on heat transfer rate

Baffle plate at end of last coil is important for heat transfer through a coil shown by first case. Results of varying gap between coils with gap viz: 10cm, 20cm and 50 cm down in CFD shown respectively in figure 13, figure 14 and figure 15. Results of 10 cm gap between coil shows good agreement shown in. Maximum heat transfer takes place through coils when gap is 10 cm because air also passes over back face of coils from inner diameter to outer diameter of coil.

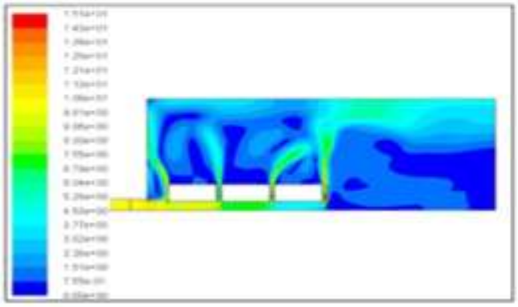


Figure 15: Velocity contour with gap between coils are 10 cm

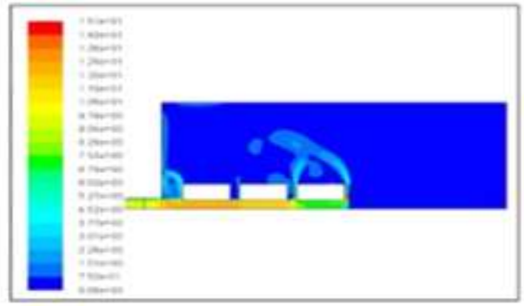


Figure 16: Velocity contour with gap between coils are 20 cm

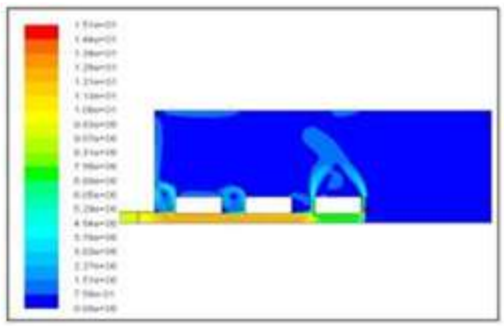


Figure 17: Velocity contour with gap between coils are 50cm

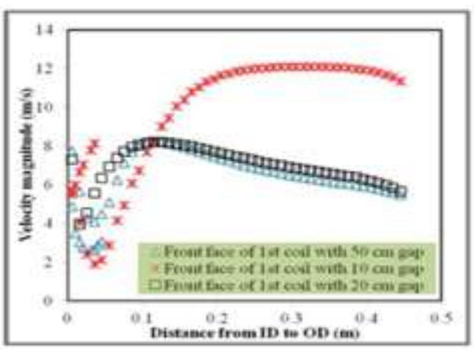


Figure 18: Velocity magnitude from ID to OD with varying gap

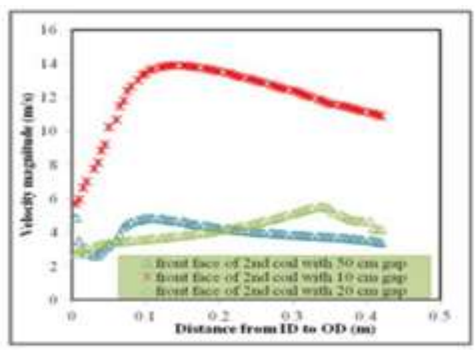


Figure 19: Velocity magnitude from ID to OD with varying gap

 ISSN: 2321-8169
1183 – 1191

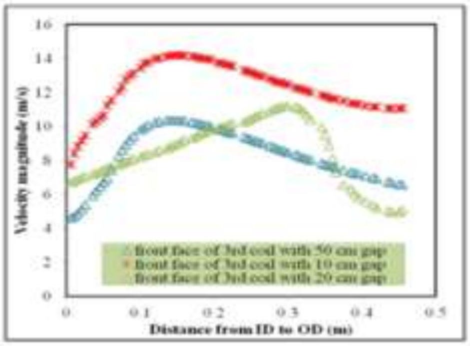


Figure 20: Velocity magnitude from ID to OD with varying gap

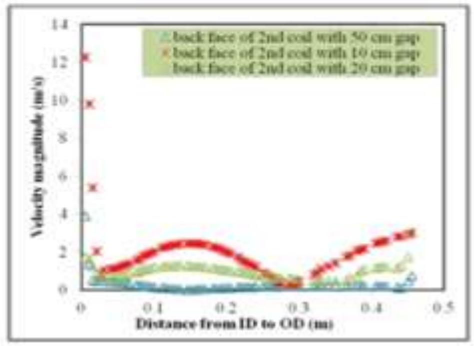


Figure 21: Velocity magnitude from ID to OD with varying gap

5.5 Effect of position of end baffle plate on heat transfer

Baffle plate at distance 10 cm showing greater heat transfer from coil because ait strike on front

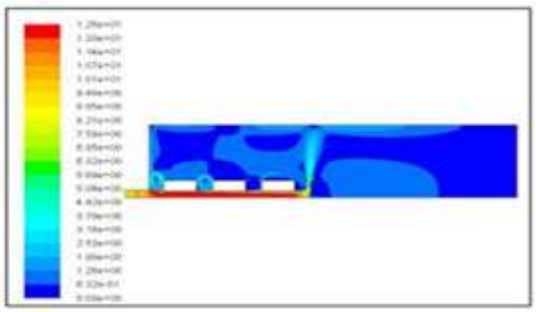


Figure 22: Velocity contour when baffle plate at 50 cm

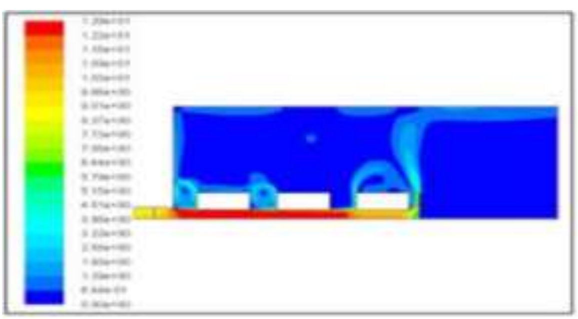


Figure 23: Velocity contour when baffle at 20 cm

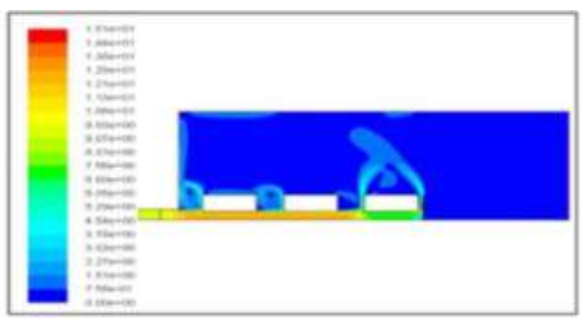


Figure 24: Velocity contour when baffle at 10 cm

ISSN: 2321-8169
1183 – 1191

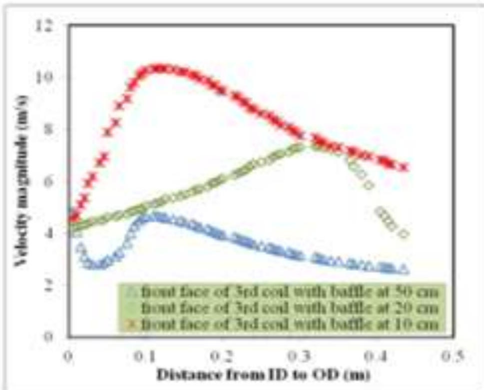


Figure 25: Velocity contour when baffle at 10 cm

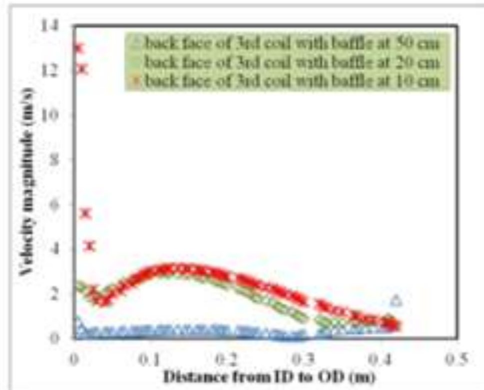


Figure 26: Velocity contour when baffle e at 20 cm

face of third coil and also back face of third coil. Velocity on front face of first and second coil remains constant in three cases. Velocity magnitude mostly seen on front and back face of third coil larger when baffle plate distance is at 10 cm compare to 20 cm and 50 cm.

6. Conclusion

CFD analysis is carried out on hot rolled sheet. Stream line velocity through fan validate with Aerotech fan manufacture.CFD analysis showing good agreement of results with Aerotech fan. Following are the conclusions identified from present CFD study.

1. When there is no baffle plate at end of coil, flow developed fully inside coils and passed through it but there is less heat transfer takes place when it passed through inside a coil. There is small gap between hot rolled coils so air present in between them so less chances of heat transfer. Once baffle placed end of last coil, maximum air strike on inner diameter to outer diameter of third coil so baffle at end important for maximum heat transfer.
2. Results for coil gap 10 cm showing maximum heat transfer through it. Velocity strike on front faces of three coils from inner diameter to outer diameter is maximum when coil gap 10 cm as compare to 20 cm and 50 cm.
3. Study of placing baffle at 50 cm distance, result of this case same as no baffle plate. Baffle plate at 20 cm from last coil showing less velocity strike on front face of third coil as compare to 10 cm baffle plate from end coil. Also large velocity strike on back face of third coil from inner diameter to outer diameter in case of 10 cm baffles distance.

Nomenclature

Latin symbols

A	Area of fan, m2
C	Velocity of sound, m/s
dh	Hydraulic diameter, m
fn	Polynomial coefficients
h	Heat transfer coefficient of air, W/(m.K)
K	Thermal conductivity of air, W/m.K^{2}
M_a	Mach number, dimensionless
N_u	Nusselt number, dimensionless
P	Pressure rise across the fan, Pa
P_r	Prandlt number, dimensionless
Q	Volume flow rate through fan, m^{2}
R_e	Reynolds number, dimensionless

ISSN: 2321-8169
1183 – 1191

References

[1] Ligong. Yang, Use of experimental data to generate swirl velocity in the fluent fan model, M.Sc thesis, Mechanical, Automobile and Material department, university Windsor, 2000.

[2] Louis Lawrence Gambogi, OPEN COIL BOX ANNEALED STEEL COOLING ANALYSIS, M.Sc thesis, University of Pittsburgh, 2003.

[3] Tobias Berg, Anna Wikstrom, Fan modelling for front end cooling with CFD, Master thesis, Mechanical department, Lulea university of technology.

[4] Morris, S.C., Good, J.J. and FOSS, Velocity Measurement in Wake of an Automobile Cooling Fan, M.sc thesis, Experimental and thermal science.

[5] Nourse, P.J, Development of an Automotive fan test facility, M.Sc thesis, Mechanical, Automobile and Material department, university Windsor, 2000.

[6] EMRE OZTURK, CFD ANALYSIS OF HEAT SINKS CPU COOLING WITH FLUENT, Master thesis, Mechanical department, MIDDLE EAST TECHNICAL UNIVERSITY, December 2004.

[7] S.Jain, Y.Deshpande, CFD modelling of a Radiator Axial Fan for air flow distribution, World academy of science, Engineering and technology volume: 6, 2012-11-28.

[8] Frank P. Incropera, David P. Dewitt,Fundamentals of Heat and Mass Transfer, Fourth Edition, John Wiley and Sons, 1996.

[9] H.K.VERSTEEG, W.MALALASEKERA,An Introduction to Computational Fluid Dynamics.