CFD Analysis of Air Cooled Condenser in Tumble Cloth Dryer

Mr.Amol. R. Khande is a Post Graduate Student in Dept. of Mechanical Engineering at Sinhgad College of Engineering, Pune, Maharashtra, India.
Dr. A. P. Pandhare is Professor in Dept. of Mechanical Engineering at Sinhgad College of Engineering, Vadgaon (Bk), Pune, Maharashtra, India.
Mr. Ranjeet D. Ghavate is Asst.Professor in Dept. of Mechanical Engineering at Sinhgad College of Engineering, Vadgaon (Bk), Pune, Maharashtra, India.

Abstract : A tumble or clothes dryer is a household appliance used to remove moisture from wet clothes after washing, Space requirement of dryer is also less compare to open cloth lines. When hot air is passed through cloths instead of exhausting this hot humid air outside close loop dryer uses heat transferring device as a heat exchanger to cool this air. Air cooled condenser is used to dehumidify the air. Flow and heat transfer analysis of this air condenser is done with the help of CFD software. Modeling of condenser is done by using Creo3.0 parametric software. Main aim of this thesis is to find out the effect of finned vs. nonfinned heat exchanger on pressure drop, heat transfer coefficient, heat transfer rate. Performance enhancement of condenser is done by using various types of fins. Rectangular, Trapezoidal and Triangular Fins types are used. Experiment is conducted in dryer testing Lab and Results are co-related with CFD Simulation.

***IndexTerms* - Air cooled condenser, Cloth Tumble Dryer, Fins, Heat Transfer enhancement, CFD Simulation.**

I. INTRODUCTION

A tumble dryer or clothes dryer is a household appliance used to remove moisture from wet clothes after washing. The dryers consist of a rotating drum called a "tumbler" through which heated air is circulated to evaporate the moisture from clothes, and drum is rotated to maintain air space between the articles so that more area comes in contact with heated air. Air is heated with the help of electric or gas heaters. Depending upon type of dryer, moist air is vented outside laundry room or dehumidified and recirculated through the drum. Tumble dryers provide a fast and convenient method of drying wet cloths, when space for drying outdoors is limited. The air can be moved into the ducts with the help of blower. Cross flow Condenser is used for dehumidifying the hot humid air. Dryer available into market in various types such as dryer with heat pump, condensing dryer and air vented dryers. To cool the humid air coming from drum, Cross flow heat exchanger is used. Performance improvement can be done with help of various types of fins such as Rectangular, Trapezoidal and Triangular fins. Energy consumed by dryer is more so our target is to reduce energy consumption per kg of load. Heat exchanger mainly used for verity of different application like milk and dairy industry, Radiators, Refrigeration, air conditioning, dryers, boilers. In this thesis we are optimizing

The air cooled condenser performance with variation of mass flow rate of hot fluid with cold fluid and inlet temperature for both fluids respectively. This will helps for getting the optimum design conditions for better performance of condenser.

II. LITERATURE REVIEW

The present work related with performance improvement of condenser in cloth dryer; the literature available on Tumble cloth dryer is referred.

Pradeep Bansal et al. [1] investigated the new design of cloth tumble dryer in which instead of electric heater they are using water to air finned heat exchanger which helps for heating air in cloths. Modified dryer reduces the value of extraction rate of Moisture by 0.1- 0.12 kWh/kg. This modified design of cloth dryer is having faster drying cycle time and more efficient over conventional dryer by 10 %when operated at same conditions.

P.K.Bansal et al. [2] discussed the four different designs of household clothes dryer with input as an electric power. They concluded that efficiency is lower for air vented while maximum for condenser dryer with heat recovery. While efficiency of condenser dryer for open and close cycle lies between Air vented and heat recovery condenser dryer. Efficiency of air vented dryer efficiency decreases with increase in relative humidity while increasing effectiveness it will increase.

Junior et al. [3] discussed about thermoelectric heat pump cloth dryer. In this heat pump is combined with recuperative type of heat exchanger. By this modification it increases the energy consumption. Since it is not having moving parts hence having more system life. Modified dryer Operates in very silent manner.

V.Yadav et al. [4] performed drying process theoretical analysis occurring inside electric tumble cloth dryer. Correlation for the moisture extraction rate is developed for determining energy consumption. It is observed that mass per unit area of cloths having significant effect on the MER. as increase in the inlet relative humidity will increases SMER due to decrease of partial vapour pressures.

J.Deans [5] investigated the performance of domestic tumble dryer. The performance test indicates that cloths distribution inside the drum of dryer depends upon density of the material and moisture contain. Overall efficiency of the dryer is higher. During testing input energy almost 16% leaves dryer in the exhaust gas, model was also developed to investigate a different method that recovers this exhaust loss of input energy. From this study it is clear that energy consumption is influenced by temperature of ambient air and its Relative humidity. Drying time was controlled by mass of drying load and power supplied to dryer.

J.E.Braun et al [6] investigated the new air heat pump cycle for cloth dryers. This new cycle shows 40 % energy improvement compare to electric dryer and 14 % improvement over the open and close cycle dyers. In this dryer low pressure air is used as working fluid so removal of lint is very easy for condenser. Air heat pump cycle dryer do not required venting to outside.

Yeongjin do et al [7] evaluated the improvement of closed loop Tumble dryer with air heat exchanger. From this it is clear that if heater power is more then drying time required is less, air temperature increases and condensation rate also increases due to higher humidity ratio in the air.

B.B.Kuchhadiya et al [8] investigated the behavior of plate fin heat exchanger for offset strip fins. Thermal performance is checked with varying mass flow rates and input parameters. Values obtained effectiveness and heat transfer coefficients from experimental are compare with values from correlation developed by researchers. From this it is clear that as cold fluid velocity increases, effectiveness is decreased while hot fluid velocity will increase. As both fluid velocity increases the value of heat transfer coefficient also increases.

Rafat Al waked et al [9] carried out the CFD analysis of air to air heat exchanger. Model was developed in CFD for Conjugate heat transfer analysis inside the membrane of heat exchanger. It consists of one flow side for hot fluid and adjacent for cold fluid. Effect of flow rate on effectiveness at inlet of heat exchanger is investigated.

Weiping Wang et al [10] investigated the characteristics of plate heat exchanger using porous media. How perforation fins will having impact the pressure drop and flow distributions is also determined. Addition of perforated fins will improve the flow distribution but it increases pressure drop values. It is difficult to model all fin channels so approach of porous media is used from the CFD Analysis point of view.

III. Problem identification

In current condition thermal performance of air-cooled condenser is found to be poor due to low heat transfer coefficient and without fins. Its performance is increased by using different types of fins. So in order to achieve better energy class you are going to design new condenser for enhancement of performance level.

IV.Objectives

1. Evaluate the effect of non-finned and finned air cooled condenser on different parameters such as temperature and pressure drops.

2. Analyze the effect of different types of fins on the performance of air cooled condenser.

3. Optimization of different parameters which are affecting the performance of air cooled condenser.

4. Find out the best fin geometry which gives maximum heat transfer.

V. Methodology

Following methodology is adopted for achieving above mentioned objectives.

Three dimensional model of heat exchanger is prepared by Creo 3.0 parametric software. The different geometries of fins are designed to enhance the rate of heat transfer. Analysis of trapezoidal, rectangular and triangular fins is done by varying the flow rate and inlet temperature as input parameters. CFD Analysis is done with the help following steps that are shown in the flow chart as follows.

Finally conclusion is drawn from the obtained CFD and experimental Results.

VI. CFD Analysis

For the CFD analysis we are using the periodicity approach because it is very difficult to simulate the full model simulation by considering CPU time, available resources and mesh counts. We are applying periodicity by one pitch for all models of condensers.

A. Non fin condenser model

a. CAD model details

Fig 1: Nonfin condenser

b. fluid model

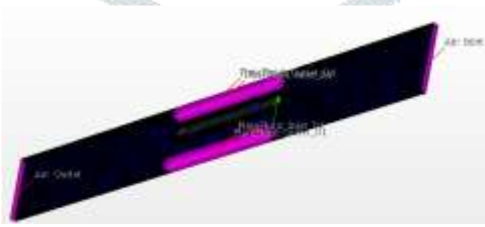


Fig 2: Nonfin Fluid model

c. Meshing models

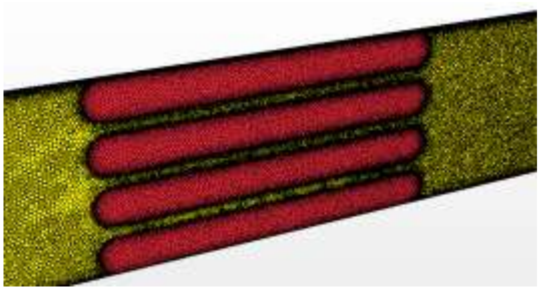


Fig 3: Mesh model of non-fin condenser

d. Material

Materials	Parts
Aluminum	Plates and Fins
ABS	Front frame and Rear frame
Rubber	Front and Rear Sealing

e. Boundary conditions

- *Non fin model*

hot Flow rate, m3/hr	cold Flow rate, m3/hr	hot Inlet Temperature, C	cold Inlet Temperature, C
197	200	42.1	22.19
201	250	47.1	22.68
250	250	55.7	23.02
253	200	62.3	22.7

- *Trapezoidal Fin model*

hot Flow rate, m3/hr	cold Flow rate, m3/hr	hot Inlet Temperature, C	cold Inlet Temperature, C
195	200	49.0	21.6
203	250	60.2	21.9
248	250	57.8	21.9
254	200	45.9	21.5

f. Assumptions

1. Flow is assumes to steady and compressible.

2. All outside surfaces are considered as adiabatic wall.

B.Trapezoidal fin model

a. Fluid model details

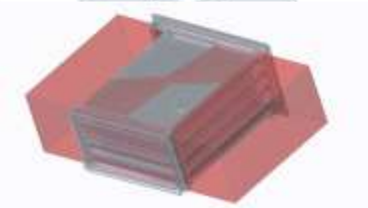


Fig 4: Trapezoidal fluid model

b. Meshing Scene

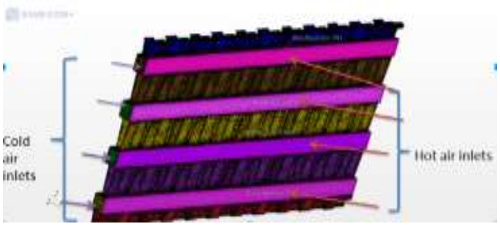


Fig 5: Trapezoidal mesh model

C. Simulation Results

A.For non-fin model

a. Temp scenes

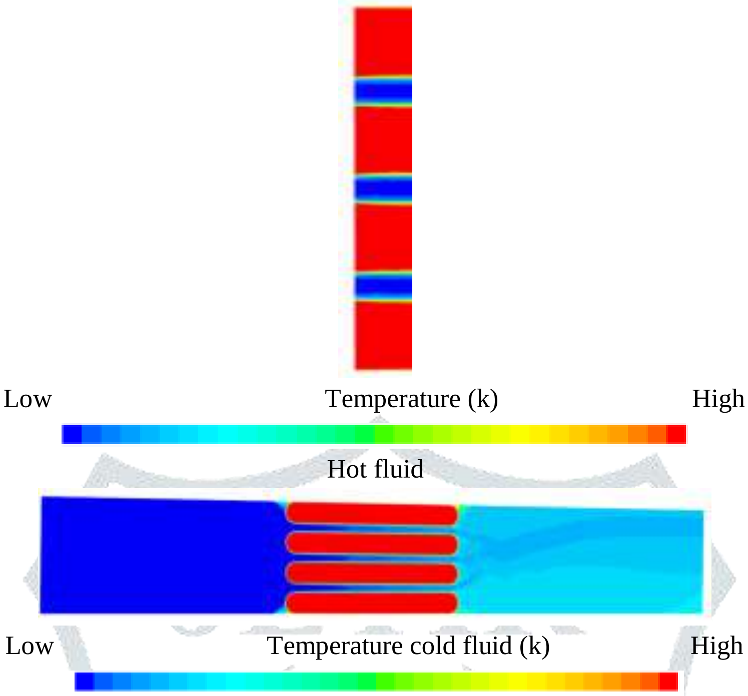
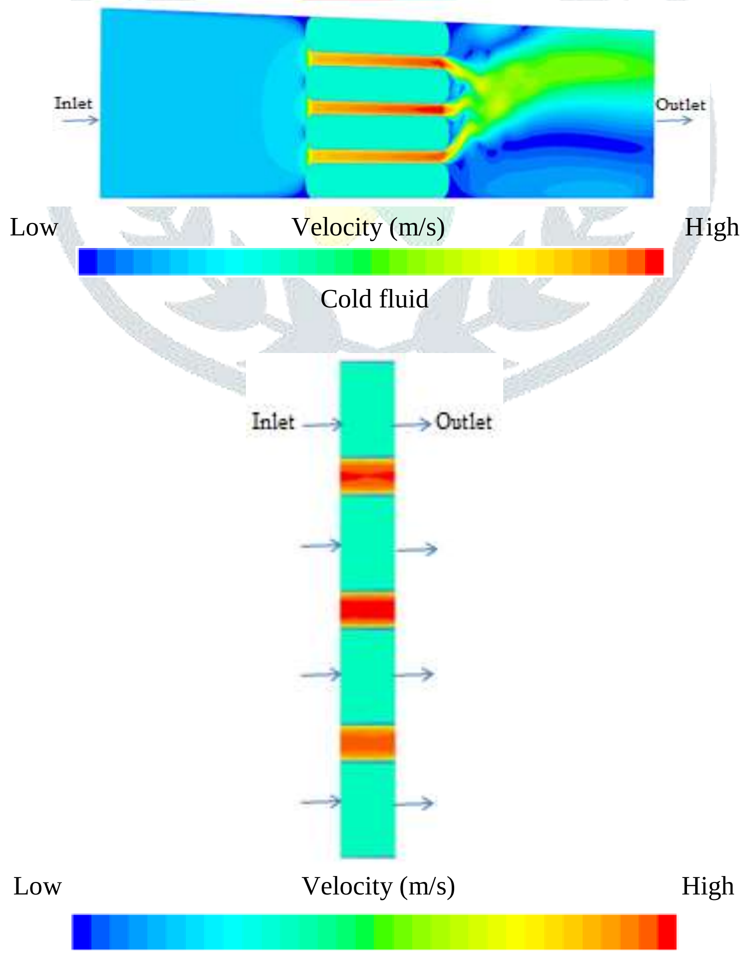


Fig 6: Temperature scene for hot and cold fluid

b. Velocity scenes



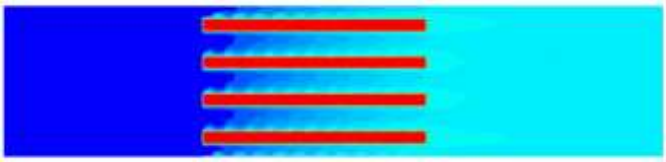
Hot fluid

Fig 7: Velocity scene for hot and cold fluid

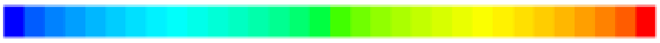
For hot air inlet temperature is taken as 42.1^{0}C. Flow of air takes place from left to right as shown in figure.at the edge of plate heat transfer takes place because of heat exchange between hot and cold air. Temperature variation is shown in scene. Velocity variation is parabolic in nature from inlet to outlet.

B. For trapezoidal model

a. Temperature scenes



Low Temperature (k) High

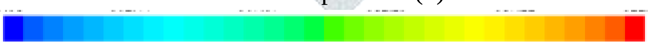


Cold fluid

Fig 8: Temp scene of hot fluid

For Trapezoidal fin inlet temperature is taken as 49^{0}C. At the edge of plate heat transfer takes place because of heat exchange between hot and cold air. Velocity variation is parabolic in nature.

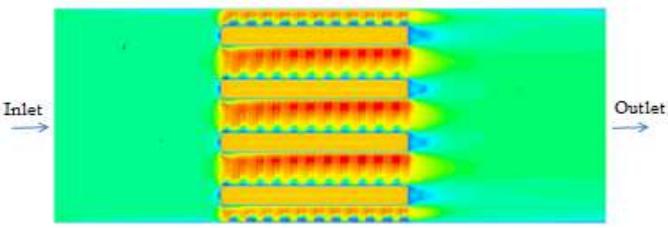
Low Temperature (k) High



Hot fluid

Fig 9: Temp scene of hot fluid

b. Velocity scenes



Low Velocity (m/s) High

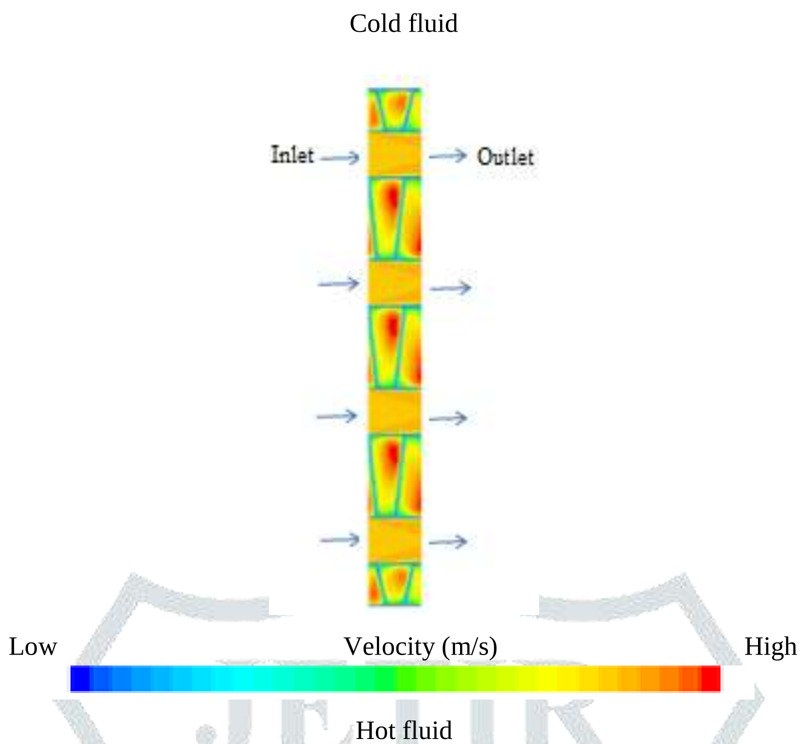


Fig 10: Velocity scene for hot and cold fluid

VII. Experimentations

A. Experimental setup

Aim of work is to find out the temperature, pressure drop and heat transfer coefficient for condenser. Experimental setup consists of Blower, Thermocouple, heater, rectangular channel, humidity and pressure sensors. Blower is attached at one end of Rectangular channel by using it air is taken into channel. Heat exchanger characterization is done by using the calorimeter and air cannon test. Calorimeter test setup is used for hot air side measurements while air cannon setup for cold air side measurements. Different types of sensors and pressure tapping with thermocouple are attached to condenser as shown in figure.

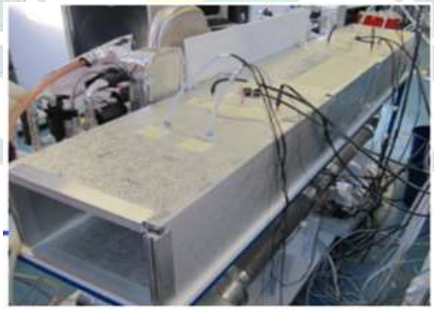


Fig 11: Inlet channel and sensors

Fig 12: Calorimeter test setup

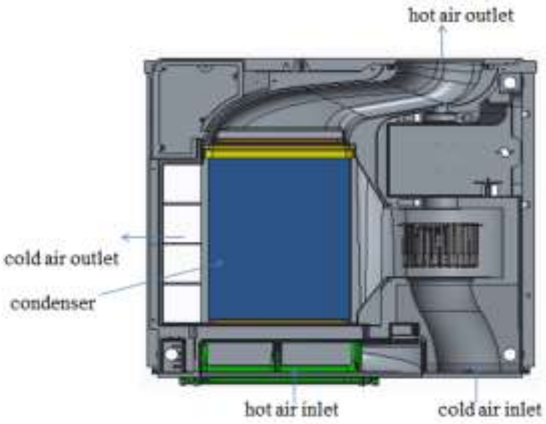


Fig 13: Condenser location in dryer

In Rectangular channel box is provided for component to be tested at one end of channel blower is attached and circular pipe with flow meter is used. Pressure measurement is done by differential pressure a sensor while humidity is also by humidity sensor air channel. Test is conducted in FAB Lab Italy at Atmospheric conditions 22^{0}c and 70 % RH. By varying the volumetric flow rate, Pressure and Temperature are measured at various locations.

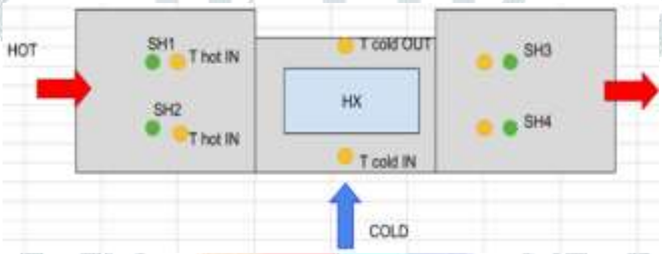


Fig 14: Schematic diagram for probe locations

Fig 15: Inside section of channel

B. Experimental procedure

At inlet of condenser hot air comes through the Rectangular channels so at inlet thermocouple and sensors are provided to measure the temperatures. After passing over condenser at outlet thermocouples are provided for temperature measurement again. From other side cold air comes in cross flow manner and passes over condenser for this one thermocouple is provided at inlet and outlet respectively. Adjust the flow of blower by using regulator. Various readings are taken by varying the flow rate and inlet temperature.

VIII. Results and Discussion

A.For non-fin model

Simulation and experimental results are having small variations, as shown in following graphs.

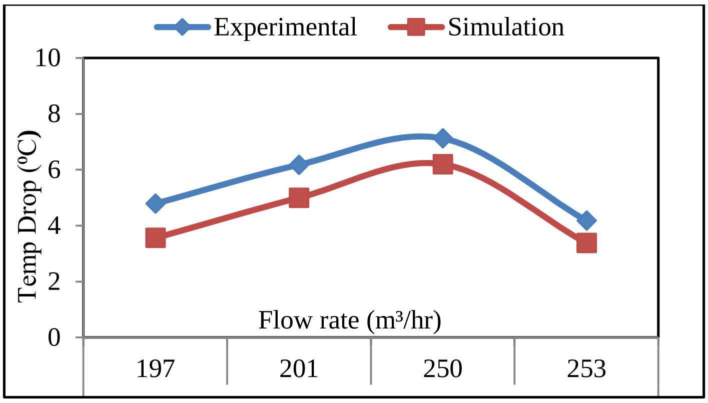


Fig 16: Simulation Vs Exp result for hot fluid Temp drop

Experimental values of temperature drop for hot fluid are more compared to simulation because of attachment of thermocouples makes that kind of errors. The approximated error 16%, thus method of solution is sufficiently accurate.

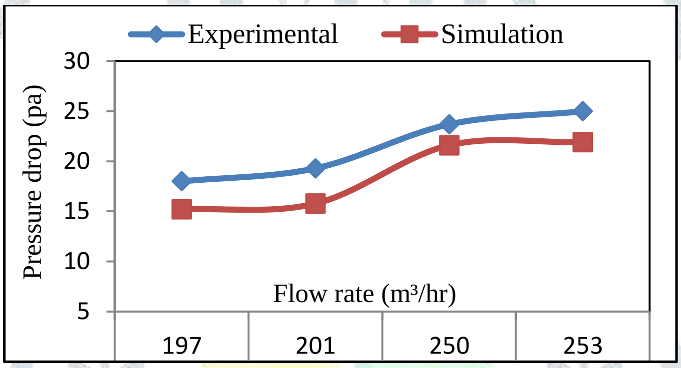


Fig 17: Simulation Vs Exp result for hot fluid pressure drop

Pressure drops are more in Exp results since it has small amount of unavoidable pressure losses. While simulating, those losses were not taken into consideration. Losses occur because of friction and due to some geometrical changes. Approximated error between CFD and Exp is 13.5 %.

B. for Trapezoidal Fin model:

For the validation of trapezoidal fin model temperature drop for hot and cold fluid, pressure drop parameters are taken.

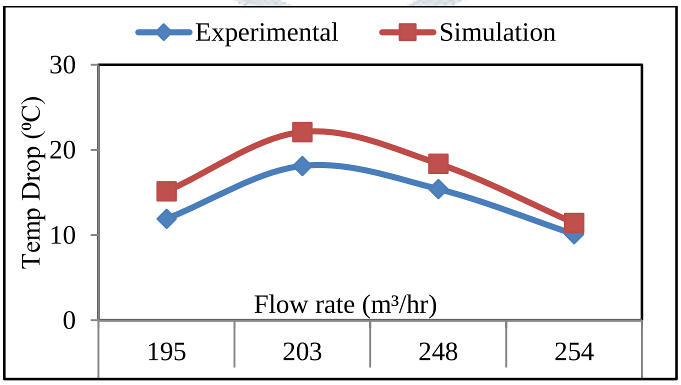


Fig 18: Simulation Vs Exp Result for hot fluid Temp drop

The approximated error 16.5 %, thus it is concluded that the algorithm used is accurate. Simulation values of temp drop are more comparing to Experimental results. Because of fins there should be disturbance occurs to fluid flow, hence some head losses occurs. CFD and experimental variation is 11%. Thus solution method used is sufficiently accurate. Pressure drops are more in experimental results since it has small amount of unavoidable pressure losses. While simulating, those losses were not taken into consideration. Losses occur because of friction and due to some geometrical changes.

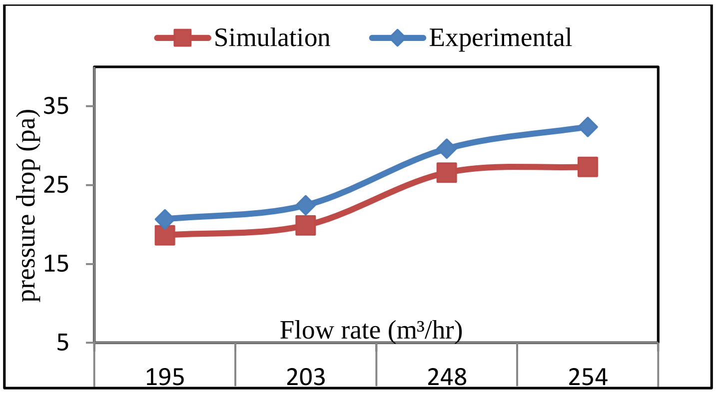


Fig 19: Simulation Vs Exp result for hot pressure drop

C. Improvement with Fins:

From the CFD simulation we get the result for the heat transfer coefficient for various types of fins.

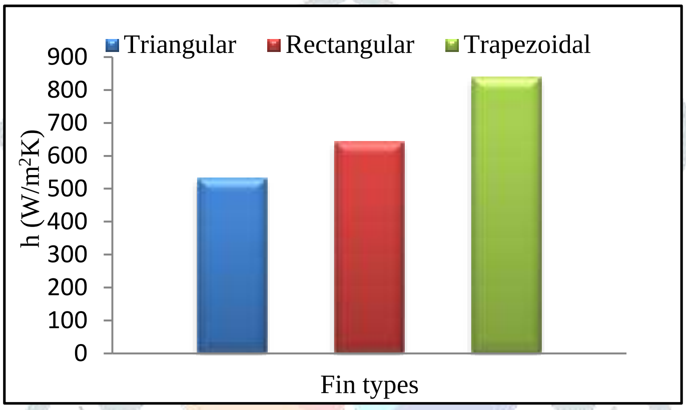


Fig 20: Heat transfer coefficient Vs fin types

Maximum heat transfer coefficient for the trapezoidal fin, due to maximum vortices form in the trapezoidal shape fin due to which effective mixing of air happens and turbulence is increased more in the case of trapezoidal fin that's why energy transfer is more, comparison with other fins.

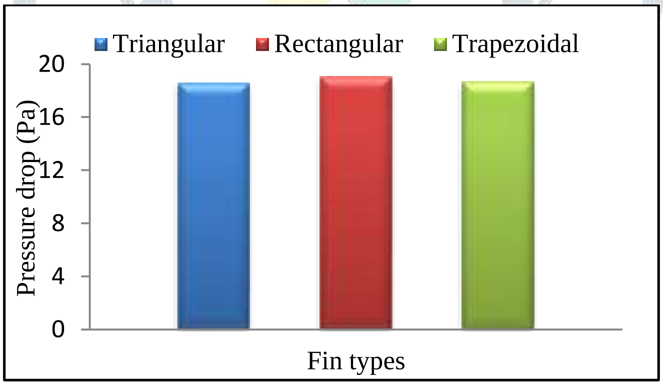


Fig 21: Pressure drop Vs fin types

From simulation we got the maximum pressure drop in the Rectangular fin shape geometry. Minimum pressure drop seen in triangular fin shape. Maximum pressure drop occurs in rectangular fin because of more obstruction to fluid flow, due to sudden geometrical changes head loss occurs.

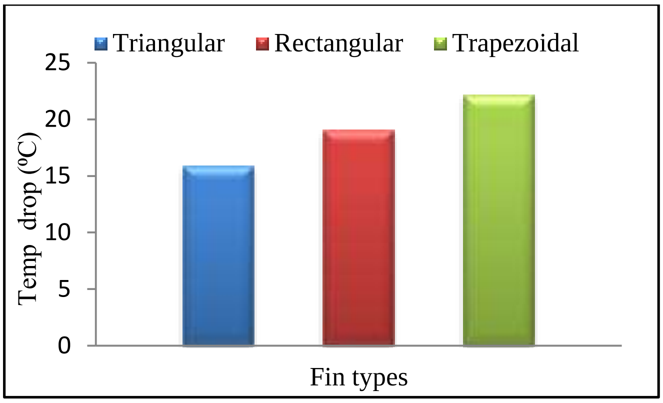


Fig 22: Temperature drop Vs fin types

From the CFD simulation we got the maximum Temperature drops in the Trapezoidal fin hence we select Trapezoidal shape geometry for achieving maximum heat transfer.

IX. CONCLUSION

Heat transfer coefficient is seen to be higher in Trapezoidal fin than other fins due to increased interfacial turbulence which will increase heat transfer. Maximum pressure drop seen in the Rectangular fin geometry, because of more obstruction to fluid flow and due to sudden geometrical changes head loss occurs. For non-fin model temperature and pressure drop variation between CFD and experimental is 15% and 13.5% respectively. While for trapezoidal fin model temperature and pressure drop variation is 17% and 11% respectively. Only CFD simulation to be used for analysis of triangular and rectangular fins with the same strategy which saves experimentation cost.

X. ACKNOWLEDGMENT

Authors would like to thanks the whirlpool corporation for their financial support and giving the access of Creo software during design stage of project and CFD STARCCM software for the analysis. Also Authors would like to express their gratitude to Lab Technicians of FAB Lab in Italy for providing experimental data as per requirement of project work.

REFERENCES

[1] Pradeep Bansal, Karishma Sharma, Sumana Islam, "Thermal analysis of a new concept in a household clothes tumble dryer", *Applied Energy*, *Elsevier*, Vol 87 (2010),pp.1562-1571.

[2] Bansal PK, Braun JE, Groll EA. "Improving the energy efficiency of conventional tumbler clothes drying systems", *International Journal Energy Research*, Vol 25 (2001), pp.1315–1332.

[3] C. Junior, G. Chen, J. Koehler, "Modelling a new recuperative thermoelectric cycle for a tumble dryer", *International journal of heat and mass transfer*, Elsevier, Vol 55 (2012), pp.1536-1543.

[4] V. Yadav, C.G. Moon, "Modelling and experimentation for the fabric –drying process in domestic dryers", Applied Energy, Elsevier, Vol 85 (2008), pp.404-419.

[5] J.Deans "The modeling of a domestic tumbler dryer", *Applied Thermal Engineering "*Vol 21(2001), pp.977-990.

[6] J.E. Braun, P.K. Bansal and E.A. Groll, "Energy efficiency analysis of air cycle heat pump dryers", International journal of refrigeration, Vol 25 (2002), pp.954-965.

[7] Yeongjin Do , sangho park, jaekeun lee," An experimental study on the performance of a condensing tumbler dryer with an air to air heat exchanger", Korean Journal of Chemical Engineering. 30 (6), 1195-1200(2013).

[8] B.B.Kuchhadiya and P.P. Rathod. "Experimental investigation of Thermal behavior of cross flow plate fin heat exchanger with offset strip fin",Procedia Technology 23(2016) pp.423-430.

[9] Rafat Al waked, Mohammad shakirnasifand Graham Morrison, "CFD Simulation of air to air enthalpy heat exchanger", Energy conversion and management 74(2013) 377-385.

[10] Weiping Wang, Jinxing Guo, Shuwen Zhang and Jian Yang, "Numerical study on hydrodynamic characteristics of plate-fin heat exchanger using porous media approach", computers and chemical engineering 61 (2014) 30-37.

[11] Conde, Manuel. R., Energy conservation with tumbler drying in laundries, *Applied Thermal Engineering*, Elsevier, 17 (1997), 1163-1172.

[12] Erek, Aytunc. Ozerdem, Baris. Bilir, Levent. And Ilken. Zafer, "Effect of geometrical parameters on heat transfer and pressure drop characteristics of plate fin heat exchanger", *Applied Thermal Engineering* 25 (2005) 2421-2431

[13] Manglik, R.M. and Bergles.A.E, "Heat transfer and pressure drop correlations forthe Rectangular offset strip fin compact heat exchanger", *Experimental Thermal and fluid science 10* (1995) 171-180.

[14] Jose Fernandez-seara, Ruben Dizand Francisco J.Uhia. "Pressure drop and heat transfer characteristics of a titanium brazed plate fin heat exchanger with OSF".*Applied Thermal engineering* 51(2013) 502-511.

[15] Yang Yujie, Li Yanzhong, Si Biao, Zheng Jieyu. "Performance evaluation of heat transfer enhancement in plate-fin heat exchangers with offset strip fins". Physics Procedia 67 (2015) 543-550.

[16] Yonghan Kim and Yongchan Kim. "Heat transfer characteristics of flat plate finned tube heat exchanger with large fin pitch". International Journal of Refrigeration 28 (2005) 851-858.

[17] Joshi, H.M. and Webb, R.L. "Heat Transfer and Friction in the Offset

[18] Strip-fin Heat Exchanger", *International Journal of Heat and Mass*

[19] *Transfer*, 30 (1987) 69-84.

[20] Praveen Pandey, Rozeena Praveen, S.N.Mishra. "Experimental investigation on the effect of fin pitch on the performance of plate type fins". *IJRET* 01 (2012) 382-388.

[21] L.Z. Zhang, "Flow maldistribution and thermal performance deterioration in a cross-flow air to air heat exchanger with plate-fin cores", *Int. J. Heat Mass Transfer* 52 (2009) 4500 - 4509.

[22] A.G. Kanaris, A.A. Mouza, S.V. Paras, "Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code", Chem. Eng. Technol. 8 (2006) 923- 930.

[23] ASHRAE Standard 84, "Method of Testing Air-to-Air Heat/Energy Exchangers," American Society of Heating, Refrigerating and Air Conditioning Engineers Inc., Atlanta. 2013.