International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-08, Issue-05, Aug 2022

Analysis of Air-Cooled Finned Tube Condenser using ANSYS

*Nomit Kumar Sahu, #Vinay Kumar

*Research scholar, #Assistant Professor, Department of Mechanical Engineering, Shri Shankaracharya Group of Institutions, Junwani, Bhilai, Chhattisgarh, India.

Abstract: **Condenser is one of the most important components of an air conditioning system and exchanges heat from surroundings. The current research is investigating the application of R410 refrigerant on air cooled condenser. The CAD model of condenser is developed in Creo design software and CFD analysis is performed in ANSYS CFX. The temperature profiles, velocity profiles and pressure profiles are generated for three different diameters of tubes i.e., 8mm, 9mm and 10mm. The heat rejection of R410 refrigerant is found to be higher than that of the conventional R134 refrigerant and maximum heat rejection is observed for 10mm tube diameter but minimum heat rejection is observed for 8mm tube diameter. The turbulence model used for the analysis is RNG k-epsilon.**

Key Words: Condenser, CFD, refrigerants

I. INTRODUCTION

Performance of a compressor-based cooling system significantly depends on the condenser. The condenser is located at the rear of the appliance in conventional refrigerators and cooling of the refrigerant in the condenser occurs by natural convection flow of the air over the condenser surface. On the other hand, compressor and condenser are located together in a chassis in built-in refrigerators. "Cooling of condenser is fulfilled by forced convection with a fan or blower as natural convection flow is not enough for cooling of the condenser. The air velocity varies between 2 m/s to 3.5 m/s for economic design. When sufficient heat transfer is not provided, compressor overworks and energy consumption increases. Furthermore, refrigerant temperature at compressor outlet exceeds critical value for safe work of compressor. Therefore, condenser design parameters should be carefully specified" [7].

II. LITERATURE REVIEW

S. C. Walawade [1] has worked on utilization of waste heat recovery from domestic refrigerator which could enable to reduce energy requirement. The set up cost was found to be economical with lower maintenance and heat can be used for both commercial and domestic purposes.

Hongfang Gu, Zhang Zhe, Haijun Wang, Chen Q [2] has investigated the performance of direct air cooled condenser (DACC) using computational fluid dynamics. The numerical results have shown that optimal performance of air cooled condenser is obtained at inclined length L_2 = 5m, and inclination angle α = 60.

Xuelei Zhang, Haiping Chen (2015) [3] has investigated the air-cooled steam condenser using numerical testing methods. The findings of the research has shown that by using rectangular configuration, the cooling efficiency is much higher as compared to other design configurations.

ShuyangCao, JinWang, JinxinCao, LinZhao, XuChen, [4] has worked on cooling tower used in commercial buildings. The wind direction, wind velocity and temperature has significant effect on cooling of rooms.

Weifeng He, Dong Han, Chen Yue, Wenhao Pu, Yiping Dai [5] has worked on performance of axial fan inlets used in condensers. The findings have shown that blade curvature angle and RPM has highest highest effect on performance of air-cooled condenser.

Parag Mishra, Manoj Arya. (2015) [6] has worked on air cooled condenser and the investigation has shown that by proper synchronization of fan, motor and pump the efficiency of condenser can be improved which could reduce energy requirement also.

III. OBJECTIVE

The current research investigates the use of R410 refrigerant in air cooled condenser. The investigation is conducted using techniques of Computational Fluid Dynamics and software used for analysis is ANSYS CFX. The temperature plot, velocity plot and heat flow is determined for diameter of tubes.

IV. METHODOLOGY

The CFD analysis involved CAD modeling, meshing, applying loads and boundary conditions. The condenser model is developed using Creo design software with

 DOI : 10.35291/2454-9150.2022.0409

appropriate dimensions. The dimensions of condensers are.

1. Diameter of tube =10mm
2. Length of the condenser =300mm
3. Height of the condenser =30mm
4. Pitch value = 15mm
5. No. of turns =12

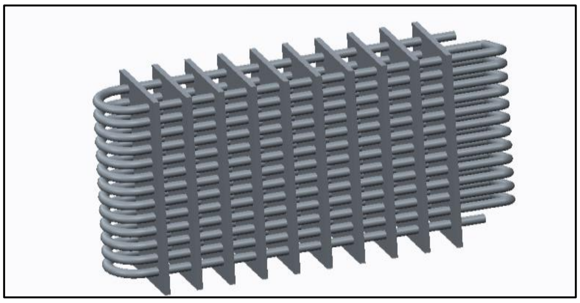

Figure 1: Condenser tubes assembled

Dimensions of fins:
1. Height of the fin = 300mm
2. Thickness of the fin = 1mm
3. Width= 50mm
4. No. of fins =10

The model of condenser is shown in figure 1 above which shows sequence of tubes arranged in parallel configuration. The model of condenser is converted in .iges file format which makes it compatible with other software. The condenser CAD model is then imported in ANSYS design modeler as shown in figure 2 below. The model checking is conducted here which involves correction of hard edges, smoothening of surfaces.

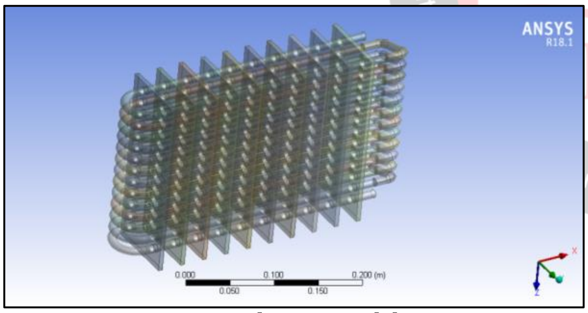

Figure 2: Imported CAD model in ANSYS

The discretization of condenser model is done in ANSYS mesher using fine relevance setting and 1.2 growth rate. The smoothing is set to medium and transition is set to slow. The number of elements generated is 101152 and number of nodes generated is 159625 as shown in figure 3.

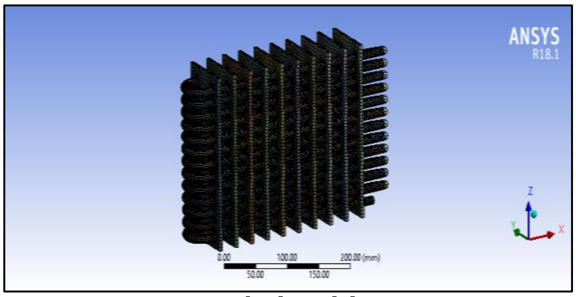

Figure 3: Meshed model in ANSYS

The computational domain is defined with fluid and solid materials. The fluid in the analysis is R134a refrigerant and R410 refrigerant and reference pressure is set to 1 atm. The turbulence model used in the current analysis is RNG k-epsilon which considers swirl flow and turbulent flow.

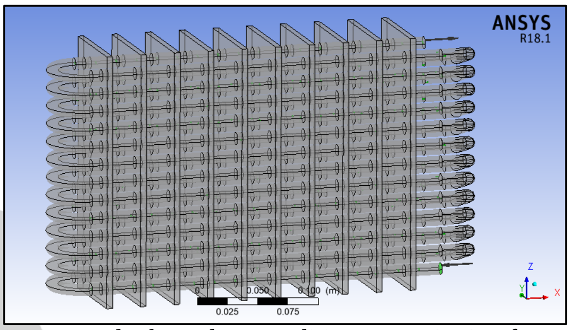

Figure 4: Inlet boundary condition in ANSYS software

The boundary conditions are defined which includes inlet boundary condition, outlet boundary condition and wall condition. The mass flow rate and initial temperature is defined which is 325K.

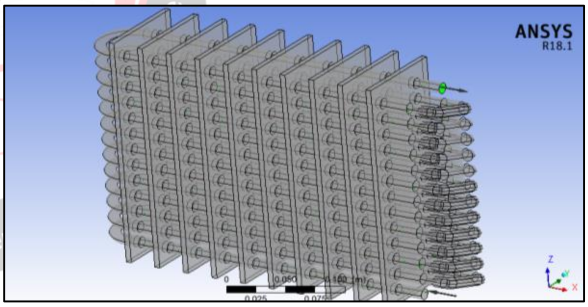

Figure 5: Outlet boundary condition in ANSYS software

The outlet boundary condition is shown in figure 5 above. The outlet boundary condition is pressure based with relative pressure difference assigned is 0Pa.

V. RESULTS AND DISCUSSION

The temperature profile and pressure profile of condenser for different diameter of condenser tubes are generated for both refrigerants i.e., R134 and R410.

 DOI : 10.35291/2454-9150.2022.0409

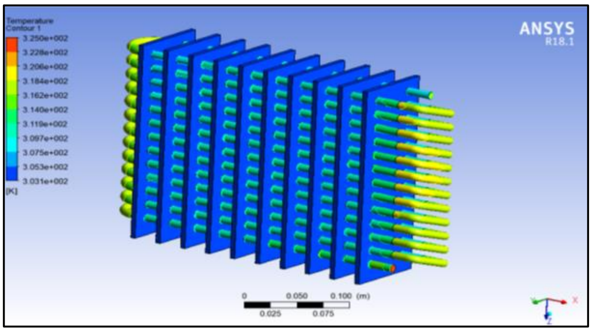

Figure 6: Temperature plot using R410 refrigerant and 8mm tube diameter

The temperature plot of condenser with R410 refrigerant is shown in figure 6 above. The outer surface of copper tubes reaches up to 314K and the zone near refrigerant inlet has higher temperature magnitude.

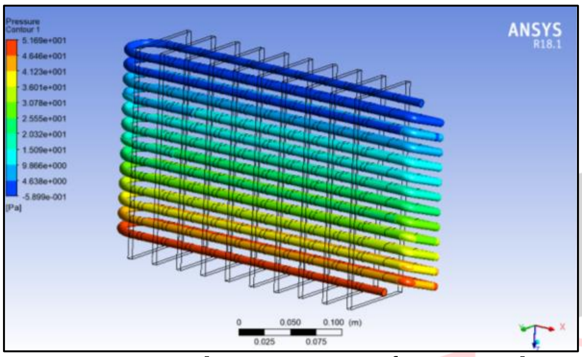

Figure 7: Pressure plot using R410 refrigerant and 8mm tube diameter

The pressure distribution profile of condenser is shown in figure 7 above. The zone near refrigerant inlet has higher pressure magnitude of nearly 44.88 Pa. The gradual reduction of pressure magnitude is observed on moving towards condenser outlet and reduces to 40.39 Pa. The pressure magnitude in last coil is nearly 35.90 Pa.

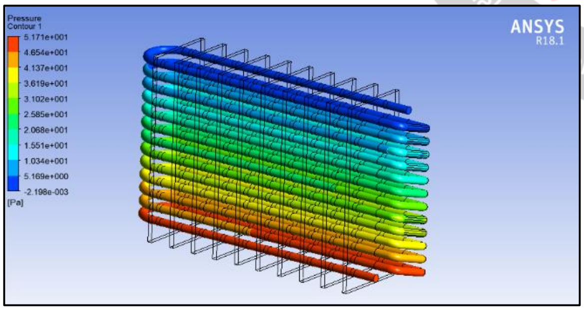

Figure 8: Temperature plot using R410 refrigerant and 9mm tube diameter

The temperature plot of condenser with R410 refrigerant is shown in figure 8 above. The outer surface of copper tubes reaches up to nearly 310K and the zone near refrigerant inlet has higher temperature magnitude.

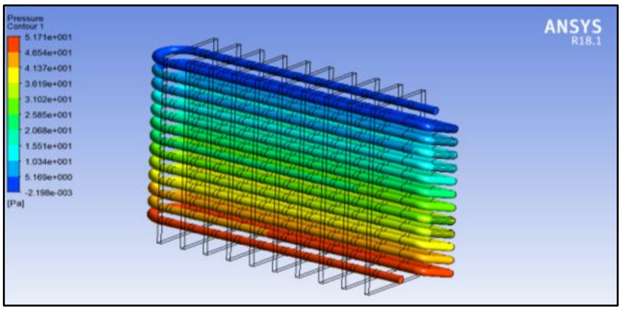

Figure 9: Pressure plot using R410 refrigerant and 9mm tube diameter

The pressure distribution profile of condenser with R410 refrigerant is shown in figure 9 above. The zone near refrigerant inlet has higher pressure magnitude of nearly 52.5Pa. The gradual reduction of pressure magnitude is observed on moving towards condenser outlet and reduces to nearly 46Pa. The pressure magnitude in last coil is nearly 41Pa.

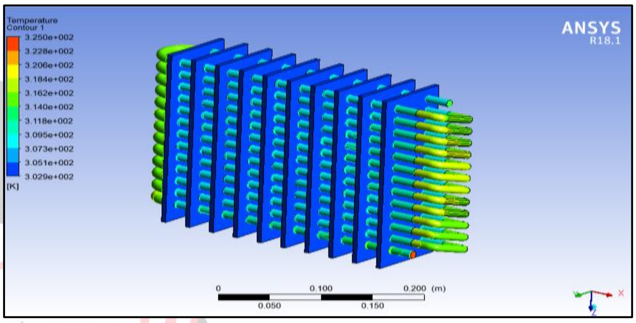

Figure 10: Temperature plot using R410 refrigerant and 10mm tube diameter

The temperature plot of condenser with R410 refrigerant is shown in figure10 above. The outer surface of copper tubes reaches up to nearly 307.1 K and the zone near refrigerant inlet has higher temperature magnitude.

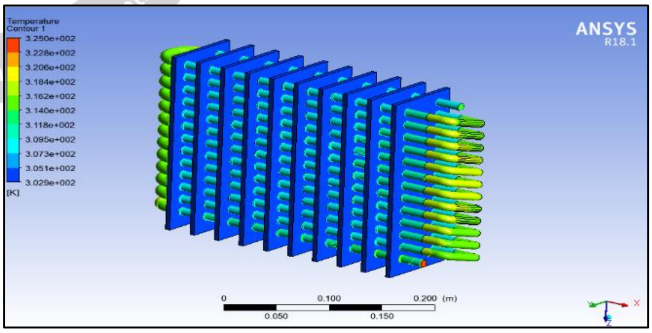

Figure 11: Pressure plot using R410 refrigerant and 10mm tube diameter

The pressure distribution profile of condenser with R410 refrigerant is shown in figure 11 above. The zone near refrigerant inlet has higher pressure magnitude of nearly 43.81 Pa. The gradual reduction of pressure magnitude is observed on moving towards condenser outlet and reduces to nearly 39.4 Pa. The pressure magnitude in last coil is nearly 35.05 Pa.

 DOI : 10.35291/2454-9150.2022.0409

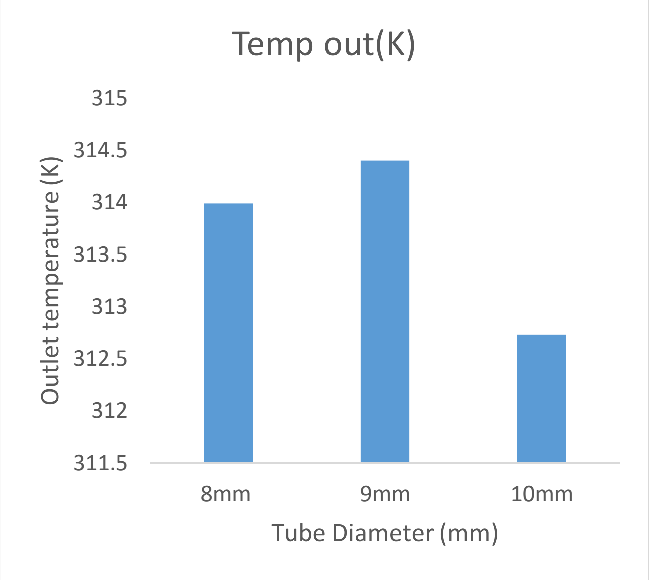

Figure 12: Outlet temperature vs tube diameter for R410 refrigerant

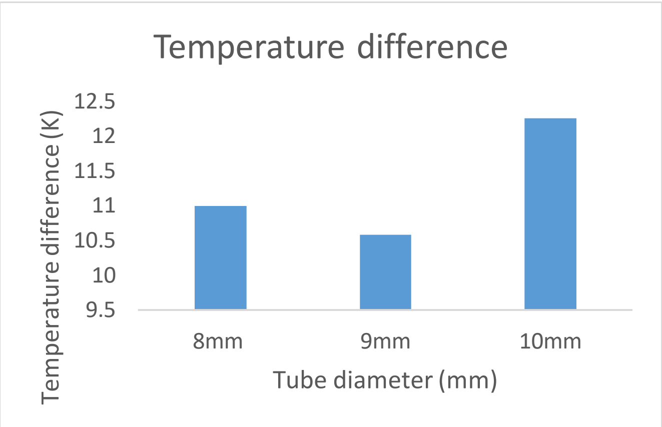

Figure 13: Temperature difference vs tube diameter for R410 refrigerant

The temperature difference between inlet and outlet is minimum for 9mm tube diameter and maximum for 10mm tube diameter. The maximum temperature difference obtained is 12.25K for 10mm tube diameter.

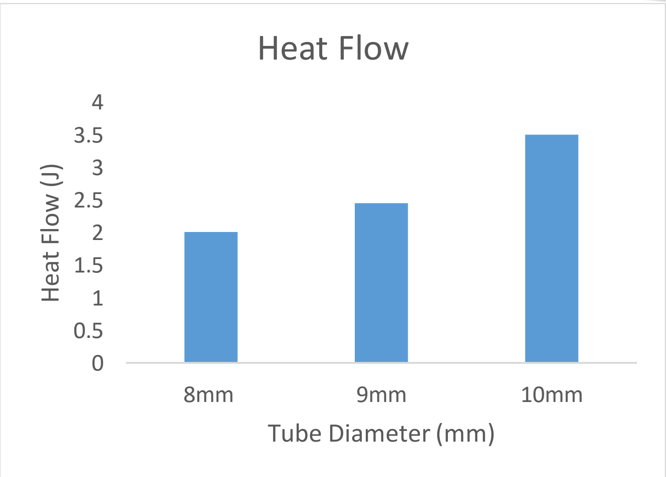

Figure 14: Heat flow vs tube diameter for R410 refrigerant

The heat flow is maximum for 10mm tube diameter i.e. 3.502 J and minimum heat flow is observed for 8mm tube diameter i.e. 2.008 J while 9mm tube diameter has 2.452 J.

VI. CONCLUSION

The current research investigates the effect of tube diameter and refrigerant properties on heat rejection and temperature drop characteristics of condenser. The CFD technique employed for analysis has proved to be viable option for substituting conventional experimental techniques which are costly and time consuming also. The findings from analysis are discussed below.

1> For R410 refrigerant, the maximum heat rejection is shown by 10mm tube diameter.
2> For R410 refrigerant, the minimum heat rejection is shown by 8mm tube diameter.
3> For R410 refrigerant, the maximum temperature difference is observed using 10mm tube diameter.
4> For R410 refrigerant, the minimum temperature difference is observed using 9mm tube diameter.

REFERENCES

[1] S. C. Walawade, B. R. Barve, P. R. Kulkarni," Design and Development of Waste Heat Recovery System for Domestic Refrigerator" IOSR Journal of Mechanical and Civil Engineering, PP: 28-32, 2012.

[2] Hongfang Gu, Zhang Zhe, Haijun Wang, Chen Qi, "A numerical study on the effect of roof windbreak structures in an air-cooled system" Applied Thermal Engineering (Science Direct), (2015)

[3] Xuelei Zhang, Haiping Chen, "Effects of windbreak mesh on thermo-flow characteristics of air-cooled steam condenser under windy conditions" Applied Thermal Engineering (Science Direct), (2015)

[4] ShuyangCao, JinWang, JinxinCao, LinZhao, XuChen, "Experimental study of wind pressures acting on a cooling tower exposed to stationary tornado-like vortices" Journal of Wind Engineering and Industrial Aerodynamics (Science Direct), (2015)

[5] Weifeng He, Dong Han, Chen Yue, Wenhao Pu, Yiping Dai, "Mechanism of the air temperature rise at the forced draught fan inlets in an air-cooled steam condenser" Applied Thermal Engineering (Science Direct), (2015)

[6] Parag Mishra, Manoj Arya, "A Review of Literature on Air Cooled Steam Condenser (A Heat Exchanger used in Thermal Power Plants)" International Journal of Research in Aeronautical and Mechanical Engineering, (2015)

[7] Singh, S. (2012). Condensers and Evaporators. R&AC Lecture 22, Version 1 ME, IIT Kharagpur

[8] Madhuraj Naik, S. N. Tande, "Analysis and Design of Steam Turbine Generator Building with R.C.C. Deck" International Journal of Innovative Research in Science, Engineering and Technology, (2016)

 DOI : 10.35291/2454-9150.2022.0409