2012

Enhancement of Heat Transfer Rate in Fractional Horse Power Condensing Unit

Vamsi Krishna Tiruveedhula
t.vamsikrishna@tecumseh.com

Sridevi Davuluru

Naveen Munnangi

Follow this and additional works at: <http://docs.lib.purdue.edu/iracc>

Tiruveedhula, Vamsi Krishna; Davuluru, Sridevi; and Munnangi, Naveen, "Enhancement of Heat Transfer Rate in Fractional Horse Power Condensing Unit" (2012). *International Refrigeration and Air Conditioning Conference*. Paper 1248.
<http://docs.lib.purdue.edu/iracc/1248>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

Enhancement of Heat Transfer Rate in Fractional Horse Power Condensing Unit

Sridevi Davuluru ¹, Vamsi Krishna Tiruveedhula ^{2*}, Naveen Munnangi ³

Tecumseh Products India Private Limited, CADEM Center
Hyderabad, India

+91-40-30116967, t.vamsikrishna@tecumseh.com, sridevi.davuluru@tecumseh.com

ABSTRACT

Rating & Sizing of heat exchangers are two important challenges encountered in the system design. The rating problem is concerned with the determination total heat transfer rate, the sizing problem is concerned with the determination of the total heat transfer surface area. The key factor contributing in higher heat transfer rate & minimum heat transfer area is the air flow over the condenser coil, as the system lies in indoor of a cabinet.

CFD (Computational Fluid Dynamics) is used to optimize components for aerodynamic performance in a Fractional Horse Power indoor air-cooled condensing unit. The analysis is validated with the test results and 4.5% deviation in air flow rate is observed. Structural analysis of components is also performed to analyze the deformations & stresses occurring due to pressures & temperatures of the flow. The behavior of flow over the condensing unit components is studied and necessary modifications for the improvement of flow over components are suggested that enhances the heat transfer rate to improve the system EER (Energy Efficiency Ratio).

1. INTRODUCTION

FHP Condensing units are most popular systems in commercial refrigeration because of their compact size. These units occupy less space and can accommodate easily in the indoor of cabinet. But the major challenge in these units is the rating of condenser, because the size of the unit is inflexible. The rating of the condenser is based on the temperature difference between the dry bulb temperature of the air entering the coil and the saturated condensing temperature corresponding to the pressure at the inlet. The total heat rejection or Total heat Transfer of condenser which is the product of mass flow of refrigerant and the difference in the enthalpy of the refrigerant vapor entering the condenser coil and enthalpy of the leaving refrigerant liquid is proportional to the Temperature difference. The Total heat transfer rate can be sharply increased by increasing air velocity across the coil by forced convection. The performance depends on the turbulence of the air flow into the coil and the uniformity of air distribution over the coil face. The air resistance through the coils assists in distributing the air properly at the downstream of the condenser, because air flow over compressor is also important. The air streaming out from the condenser has to pick up heat from the compressor also. Hence the surface geometries, flow arrangements, flow rates, inlet temperatures affect the aerodynamic performance. The objective is to perform CFD & Structural analysis to identify the Efficiencies or Deficiencies of the fan shroud towards aerodynamic performance and its strength in the condensing unit.

Figure 1: Layout of unit

Fig. 2: Side view of unit

2. PROBLEM SET UP

The present study is made on a FHP condensing unit which consists of condenser coil, fan, Motor, shroud to mount the fan and motor on condenser, compressor, receiver tank, piping & electrical wiring. The baseplate houses all the components of the Condensing Unit. The condenser is a four row, 8 FPI(fins per inch) finned tube. The compressor is Tecumseh Products Company's AEA4440. The Fan & Fan motor is mounted on a shroud, which in turn holds on to the condenser.

The CFD model for this study is built using commercially available software. The code solves the Reynolds averaged N-S (navier-stokes) equations. The effects of turbulence are modeled using the standard SST (shear stress transport) turbulence 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 non-dimensionlized maximum residuals are reduced to $1.0\text{e-}04$.

The baseline results obtained from the CFD analysis are compared with the test results. The design iterations are verified in commercially available condensing unit codes called R & R program.

The fluid volume consists of Condenser, Compressor, Fan, Fan shroud, Motor & base plate is considered in the analysis. A rectangular box is placed at the inlet of condenser domain to regulate the flow in front of condenser. Clips, nuts & bolts, Coil tube bends & piping are neglected in the domain consideration.

Figure 3: 3-D Cad model of the unit

Figure 4: Simplified model of the unit

Figure 5: Fan 3-D model

Figure 6: Condenser coil

Figure 7: Shroud model

3. DESCRITIZATION & BOUNDARY CONDITIONS

A convenient feature of the software is that the grids need not match at the grid interfaces. Hence it is easy to connect grids at the critical surface, where one side of the surface is smallest area and the other side is largest one. Hexahedral meshing is done for the Inlet domain & condenser coils. Tetrahedral mesh is generated for all the other domains like fan, fan shroud & downstream area along with compressor. The grid is generated such that the minimum angles are more than 20 degrees and quality is above 0.3. The grid generated for all the components are such that at least 10 nodes are placed near the wall to capture boundary layer. The mesh for the condenser coils is performed for one fin fluid volume & is copied & translated to complete the full 72 finned condenser.

Figure 8: Fluid volume of the unit

Figure 9: Discretized volume of the unit

Figure 10: Inlet domain

Figure 11: Condenser domain

Figure 12: Condenser coils

Figure 13: Condenser coils zoomed

Figure 14: Fan model

Figure 15: Shroud & compressor parts

All the components are stationary except the fan, which is rotating. A mixing grid interface between the stationary & rotating components is given as Frozen rotor interface using the software attachment facility. The exit fluxes from the rotating components are averaged efficiently near the rotating-stationary component's interface.

Air at 25⁰ C is considered with 101325 Pa as reference pressure. The density, Dynamic viscosity, Thermal conductivity & Cp are changing with the temperature variation. Total energy equations are considered for the analysis since the effects of temperature has to be simulated. Though the condenser has three phases such as desuperheating, condensing & subcooling, refrigerant fluid is not considered in the analysis. It is assumed that the temperature on the coil surface is constant to 55⁰ C.

Since the unit is placed in cabinet it is assumed that air is flowing into the domain from the atmosphere in all directions. So, inlet & outlet boundary conditions are given as opening pressure of 0 Pa & opening temperature of 25°C.

Figure 16: Domain with boundary conditions

4. EXPERIMENTAL SETUP

The unit is tested in an exclusive wind tunnel test rig at Tecumseh Product Company, North America as per the AMCA (Air movement and control association) standards and the air flow rate across the unit is measured. The data obtained from the test is used to compare the results obtained from CFD simulation. The CFD results are thus validated and further design modifications are analyzed and suggested.

5. ANALYTICAL STUDY

From the Fan Handbook by Frank P. Bleier [1] the Fig 17 is explained as follows. The fan is exhausting from the left portion of the system and is blowing into the right portion of the system. It is noted that:

- The ambient static pressure near the inlet of the system is zero again.
- As the air velocity increases from zero to the duct velocity, the static pressure decreases to a negative value equal to the velocity pressure in the duct plus a turbulence loss.
- The duct friction reduces the static pressure at a slow rate to a more negative value.
- The resistance of certain components reduces the static pressure at a steeper rate to a more negative value.
- The fan rises the static pressure from its maximum negative value to a maximum positive value.
- The duct friction reduces the static pressure at a slow rate.

Figure 17: Typical graph of variation of static pressure along a ventilating system

5. RESULTS & DISCUSSION

The flow at the outlet of the compressor is considered as the result comparable with the test. The pressure variation across the domain is an interesting phenomenon to observe. Various planes at equal distances are generated across the domain to capture the static pressure. The pressures at these planes are extracted and a graph is drawn with pressure v/s location. The Fig 18 shows the location of planes at which results are taken.

Figure 18: Planes at which results are extracted

The fig 19 shows the variation of pressure across the system from the inlet domain to the outlet domain. The trend line graph of the system resistance in Fig 17, and the static pressure variation in Fig 19 can be correlated by considering the inlet duct as Inlet domain, the Dampers as the Finned coils, duct as the shroud cone, Fan as fan domain, the duct, Dampers & last duct as Shroud ribs portion. It is observed that there is a pressure drop of 8 Pa in the condenser coils. The pressure rise in the fan is about 19 Pa. The results show 168SCFM at the outlet of unit which is 4.5% off with the test results.

Fig 20 shows the velocity variation across the domain from inlet to outlet. The blue color shows the velocity is around 0-1.9 m/s. The increase in the velocity is seen in green in color near the fan. It can also be seen that at the bottom of the compressor there is flow. From Fig 21 the static pressure is almost same except near the fan. Fig 22 shows the temperature variation along the system. The maximum temperature achieved is 327 K. Fig 23 shows the stream line plot. The smooth flow of air in the coils section can be observed here. Fig. 42 shows the velocity vector plot. The recirculation zone can be observed near the fan tip.

Figure 19: Static pressure variation across the domain

Figure 20: Velocity across the domain

Figure 21: Pressure distribution

Figure 22: Temperature distribution

Figure 23: Streamlines

Figure 24: Vector Plot

Figure 25: Vector Plot Zoomed view

6. STRUCTURAL ANALYSIS

Shroud design is verified not only in aerodynamic performance but also in the structural stability of the shroud in terms of with standing the motor weight, pressure & temperature. Shroud alone is considered for the analysis. Motor mass is applied at the center of gravity of the motor and is connected to the shroud remotely. Shroud clips are fixed in all directions. Pressures from CFD analysis are transferred on to the faces of shroud. Maximum temperature of 54°C is applied at the inner surfaces of shroud. Polyethylene material is used for analysis.

Figure 26: Shroud considered for the analysis

6.1 Results

Stresses are occurring at the clips, which are clamped to the condenser coil and are below the yield strength of the material. Fig 27 shows the deformations plots and Fig. 28 shows the VonMises stress plot.

Figure 27: Deformations

Figure 28: VonMises stress

7. DESIGN OF EXPERIMENTS

By observing the flow losses in the system following areas were identified for further study.

Figure 29: Areas of improvement

7.1. DOE-1: Coil & Fin Optimization

The number of coils is reduced to obtain less pressure drop and more air flow rate. Since the number of coils is reducing the coils are optimized to give same amount of heat rejection. Due to IP reasons the data and information on optimization cannot be published. The system EER is observed in R&R program. The CFD results show an improvement of 3% in the flow due to the reduction in the number of coils. It is observed that there is less resistance to the flow in the coils. These variations are observed in the tests conducted also. Fig.32 shows the comparison between Baseline & DOE-1. It is observed that the pressure loss in the condenser coils has reduced.

Figure 30: Pressure Plot

Figure 31: Velocity Plot

Figure 32: Pressure variation across the domain

7.2. DOE-2: Shroud Optimization

From the velocity vectors of the Base line analysis it is observed that there is flow from the discharge side to the suction side. The clearances between Fan & shroud are modified to reduce the leakages between discharge side & suction side. While calculating the clearances the material deformations due to loads & thermal effects, fan Thermal expansion, motor & shroud alignment are considered. Material deformations from the structural analysis and relief from the stack up analysis has to be considered when finalizing the design. The CFD results show an improvement in the flow by 4%. The pressure gain in the fan has increased. The recirculations observed previously have reduced.

7.3. DOE-3: Combination of DOE1 & DOE2

Both the Doe's are incorporated in this DOE and results are observed. It is observed that the flow has improved for about 8 %. This is taking the advantage of both. The pressure loss at the coils section has decreased and pressure gain in the fan domain has increased.

8. CONCLUSION

DOE-1: Coil & Fin Optimization: It is concluded that by reducing the number of coils, there is a flow improvement of 3%. The pressure loss in the coils has reduced. These variations are observed in the tests conducted.

DOE-2: Shroud Optimization: It can be concluded that the leakages between fan and shroud can be reduced by optimizing the shroud design. This has increased the flow by 4%. Recirculation near the fan tip have also decreased.

DOE-3: Combination of DOE1 & DOE2: It is concluded that by optimizing the shroud & reducing the number of coils, there is an improvement in the flow by 8%. This improvement is contributed by both fan & coils.

From the R+R program, It is observed from the R+R program that, with improvement in 2% - 5% of flow, there would be an improvement of 5%-10% in EER.

REFERENCES

1. Frank P. Bleier, *Fan Handbook*, McGRAW-HILL
2. 1996-ASHRAE Hand book. HVAC Systems and Equipment
3. M. Necati Ozisik, *Heat Transfer A Basic Approach*, McGRAW-HILL INTERNATIONAL EDITIONS