RESEARCH PAPER | Engineering | Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

Performance evaluation of computational procedure and investigational performance of cooling system used in oil flooded screw compressor system.

KEYWORDS: Computational Fluid Dynamics, Fluid flow calculation, Oil injected screw compressor, compressor, Air cooling system design, Limiting ambient temperature [LAT], Cold fluid temperature difference [CTD]

Prof. Rathod Trupti Y	Prof. Jigar Suthar
Assistant Prof. Department of Mechanical Engineering, Saffrony Institute of Technology, Mehsana	Assistant Prof. Department of Mechanical Engineering, Saffrony Institute of Technology, Mehsana
Dr. P.K. Brambhatt	**Nilesh Vora**
Associated Prof. Government Engineering College, Modasa	Assistant Manager, Engineering Ingersoll-Rand (India) Limited

ABSTRACT Oil injected twin-screw compressors are widely used for medium pressure applications in many industries. Screw compressor is one of the most common types of machine used to compress gases & air. The demand of compressed air at ambient temperature for lowering the size of accessories used in air system has made the positive displacement compressors inevitable. Rotary screw compressors is a positive displacement compressor comprises of a pair of intermeshing rotors with helical grooves machined on them, they may operate without internal lubrication or with oil injected. Screw compressors do not rely on suction and discharge valves to regulate the flow of gas through the compressor, to main compressed air temperature near to ambient temperature cooler design and performance play vital role. Heat exchange is an important unit operation that contributes to efficiency and safety of many processes. Design, manufacturing and validation of coolers for a mixture of operating conditions by experiments are very complicated, expensive as well as time intriguing procedure. With help of simulations and use of CFD the development time and efforts can be minimized easily. The present work is aimed towards generation of geometry and meshing of those to evaluate performance of cooling scheme. The Simulation of working by using moving reference frame and dynamic mesh model is done. The objective of this study is to understand the impact of a variety of parameters such as air flow pattern, suction and discharge air angel, cooling air velocity etc.

The work carried in CFD been also verified in industrial laboratory to evolutes actual conditions based results and gap was observed in both consequences. An attempt is made in this paper to present solutions for identify the reason for dissimilarity in both approach.

II. Introduction

Oil injected screw compressor is an assembly of many sub systems likes air end, drive system, separation system, cooling sub system, enclosure system, control electrical system and miscellaneous sub system. A schematic diagram of rotary compressor is shown in Figure 1. Air-cooled cooling subsystem consist of air cooler, lube cooler, cooling fan thermal bypass valve, lube manifold, and filter as main system components.

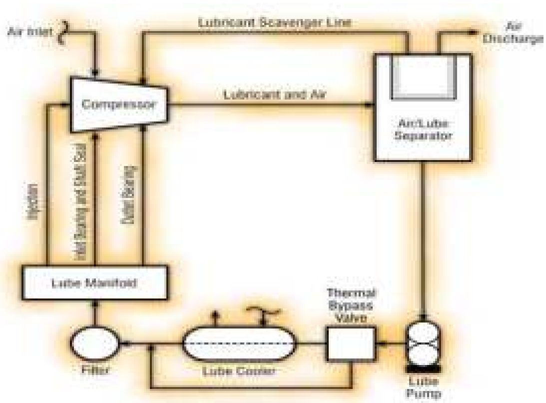

Fig.1 cooling subsystem of screw Compressor

Atmosphere air is entered in compression chamber via suction filter and inlet valve mechanism and compressed in compressor chamber with help of male female rotor pair, results in higher discharge pressure with respect to suction pressure as well as increased temperature. Compressor consists of mainly two circuits. Air cycle as an open system and Oil cycle as closed system. The reduction in temperature of open cycle is directly impacting on sizing of downstream equipments, whereas same in open cycle is resulting in improved thermal efficiency and overall efficiency of compressor. Temperature controlled of oil in oil cooler has better control in oil injection temperature. Which has direct impact on heat rejection in air end, bearing temperature controller, hence it is most important to have exact prediction of cooler performance in all operating condition. The experimental validation for variety of operating condition is not cost effective solution so use of technology like computational fluid dynamic is the final approach to handle such of challenges.

III. Nomenclature

Q_{Lat}: Latent Heat load (kW)
Q_{Sens}: Sensible Heat load (kW)
Q_{da} : Heat load of dry Air (Kw)
Q_{v} : Heat load of wet Air (Kw)
C_{pair}: Specific heat of Air at constant pressure. (KJ/ (kg-°K))
Cp_{v}: Specific heat of Air at constant pressure. (KJ/ (kg-°K))
T_{air1} : Temperature of air inlet (˚C)
T_{air2} : Temperature of air outlet (˚C)
T_{amb}: Measured ambient temperature (˚C)
T_{cd}: Measured compressor discharge temperature (˚C)
Q_{H2O} Condensate (kW)
v: Flow velocity (M/sec)
Q: Heat load (kW)
U: Overall heat transfer coefficient

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

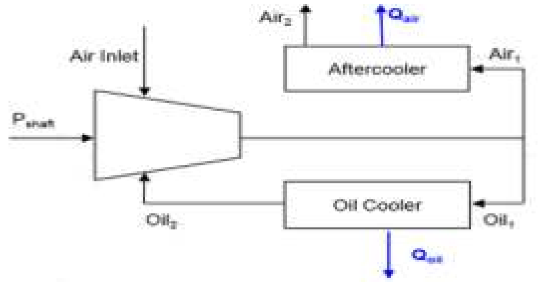

Fig.2 Schematic of heat generation and rejection in screw Compressor

IV BASIC CALCULATIONS

Design of a heat exchange as a component is to a large extent an engineering art despite high sophistication in heat exchanger thermal modeling. Some of the final decisions are based on qualitative judgments due to non quantifiable variable associated with exchanger manufacturing and other evaluation criteria. Still analytical modeling a very valuable tool to overcome these challenges. A screw compressor operated normally at 2800 rpm for 7 bar discharge pressure, where air is constant in contact with the oil.

The development of cooler starts from basic calculations of heat load to be handle or rejected from fluid (air / oil) to be cooled. Considering energy balance in a differential segment of a single-pass heat exchanger shown schematically in Fig 2. The rate of heat transfer in this segment is the rate of heat transfer in this segment is

$$dq(x) = U\Delta T(x)dA(x).$$

Where U is the overall heat transfer coefficient, ΔT is the local temperature difference between the hot and cold fluids, and dA is the contact area in the differential segment. The overall heat transfer coefficient is inversely proportional to the total resistance R_{tot} to the heat flow. The latter is the sum of (1) resistance $R_{conv,h}$ to convective heat transfer from the hot fluid to the partition between the fluids, (2) resistance R_p to thermal conduction through the partition, and (3) resistance $R_{conv,c}$ to convective heat transfer from the partition to the cold fluid therefore

$$U = \frac{1}{R_{tot}} = \frac{1}{R_{conv,h} + R_p + R_{conv,c}},$$

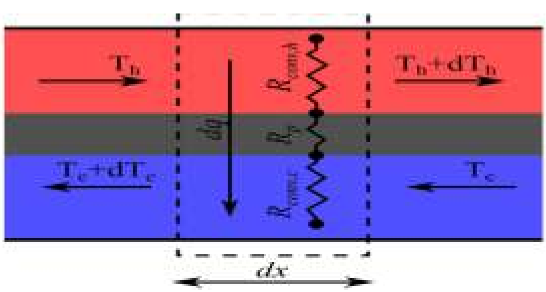

Fig.3 Energy balance in a differential element of a single pass heat exchanger

$$\dot{Q} = \frac{T_H - t_{co} - (T_{ho} - t_i)}{\left[\frac{1}{h_i \cdot A_i} + \frac{L_W}{k_W \cdot A_W} + \frac{1}{h_o \cdot A_o}\right] \cdot \ln\left[\frac{T_H - t_{co}}{T_{ho} - t_i}\right]}$$

Toil1=(QA/E+moil*cpoil*Toil2+mda*cpair*Tamb+mv*cpv*Tamb) / (moil*cpoil+mda*cpair+mv*cpv)

After cooler heat load can be derived from following equation:
Qsens = Qda + Qv

Qda = mda * cp,air * (Tair,1 - Tair,2)

Qv = mv,1 * cp,v * (Tair,1 - Tair,2)

QLat = QH2O = QH2O = mH2O * hfg

CTD and LAT can be calculated from following equations:

$$\varepsilon_{CTD} = 1 - \exp\left[\frac{1}{C_{r,CTD}} \cdot NTU_{CTD}^{0.22} \cdot \left(\exp\left(-C_{r,CTD} \cdot NTU_{CTD}^{0.78}\right) - 1\right)\right]$$

LAT $_{\text{compressor oil}}$ = (Limiting temperature – Tcd) + T_{amb}

V. Computational Analysis

The aim of this study is to investigate the temperature distributions with help of sophisticated tools like CFD. The CFD of air cooled cooling subsystem has been carried in ANSYS software in present work with following assumption and boundary conditions.

The selection of mesh elements, numbers of nodes are based on CFX mesh tool method. Finite number of Tetrahedral Elements of all parts as shown in fig 3 & number of elements & nodes are shown in the Table.1, In this set of analysis, Inertial and Viscous resistance coefficients are been calculated using Hazen Dupuit Darcy Model [5] (porous media approximation) for oil and air coolers pressure drop as inputs. The pressure drop estimation is done by wind tunnel method with three different volumetric flow. Inertial term is calculated from given mass flow, pressure drop average value and viscous term is chosen by literature survey [8]

Domain	Nodes	Elements	Tetrahedral	Hexahedra
Domain A1	32781	136967	13267	3902
Domain A2	8936	42184	43124	1200
Domain A3	12372	63829	64920	1347
∑ A	54089	242980	121311	6449

Table.1 Meshing elements and elements used in Domain.

V.I Assumption

While doing simulation of cooling system in CFD certain assumptions were pre defined in system and those are as listed here.

- Viscosity of lubricating oil has min negligible change with change in temperature.
- Rate of condensation is zero irrespective of humidity presence in atmospheric air
- Limiting fluid external for air cooler and internal for oil cooler.
- Ambient temperature is constant.
- Oil and Air coolers are modeled as solid blocks treated as porous media based on the pressure drop calculated.[6]
- Enclosure is having not allowing air leakages through compressor package.

V.II Boundary Conditions

The boundary conditions of the mathematical model are discussed as below.

- The time-dependent properties of the compressed air and the performance of the oil-injected screw compressor are calculated by a lumped parameter analysis.
- The errors of the calculated volumetric and isentropic efficiencies are both less than 2%.
- Maximum operating ambient is limited to 45 ˚C
- Heat Load carried out from Oil: 40 kW / air 15 kW
- Oil flow mass flow: 60 LPM
- Cooling air average velocity : 5.5 M/sec

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

- Blower flow 2480 CFM @ static pressure drop 1.38" Wg

V.III System Analysis for Temp / Pressure & Velocity Predictions:
The entire system consists of cooler, enclosure for pre defined passage of cooling medium and dynamically balanced centrifugal blower are model and used for meshing. The velocity stream line, temperature variation and heat distribution in all domains due to hot and cold fluid are shown in following figures. The results of same are displayed in table format for better clarity and comparisons:

V.IV CFD Observation:

• The CFD results indicated the total cooling air flow generated by blower mechanism is 3400 CFM with static pressure rise of 0.06 PSI while same was expected as 3250 CFM @ static pressure drop 0.07 PSI.

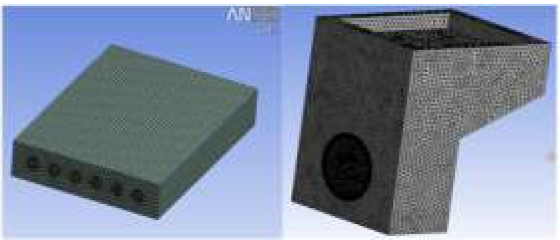

Fig.4 Meshing of Air cooler and its subsystem

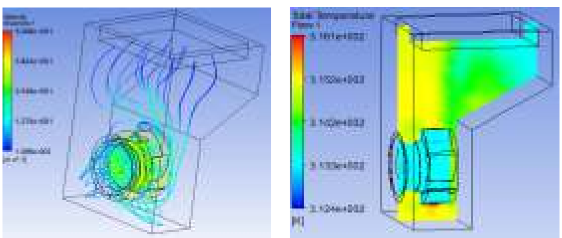

Fig.5 Velocity Streamlines & Temperature variations

•The average velocity over the cooler was predicted as 5.3 m/ sec while cooler were design based on 5.5 m/s over cooler grid.

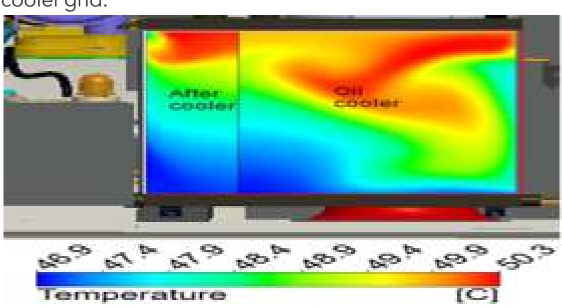

Fig.6 Temp distribution in coolers

• Out of total flow of 3400 CFM the distribution towards the air and oil cooler for cooling the internal substance was predicted as 73 % and 23 % while 4% of air flow is not passing through cooler due to turbulence generated in cooling chamber which indicated the loss of power due to non effective utilization of flow generation by blower while same is expected as 76 %and 24 % form oil and air cooler respectively from cooling airflow of 3250CFM
• As we have defined cooler as porous media with al alloy as material the pressure drop across the width of cooler was predicted 0.03 PSI Vs expected value of 0.05 PSI.

VI. Experimental setup:
The actual system simulation is carried under standard laboratory condition meeting all ISO requirements. Results were taken under steady state conditions [±2] °C Variation for 15 min].Fig:06 -07-08 described the experimental setup used for testing the cooler performance such as LAT / CTD and face velocity at inlet – out let of subsystem, flow generation through centrifugal blower.
The technical specifications of wind tunnel used for thermal performance testing are as listed here:

• Air Velocity 1 to 30 M / Sec
• Water / Coolant Flow up to 225 LPM
• Water / Coolant heating capacity up to 130°C

Fig.7 cooler testing in wind tunnel

Fig.8 Cooler leak and performances testing

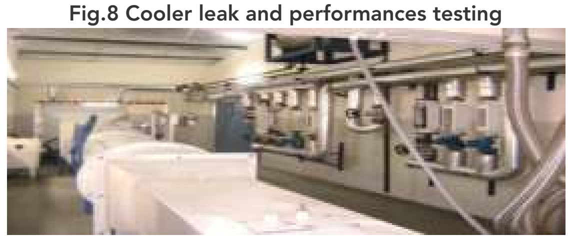

Fig.9 Rotameter & heater used for lubrication temp and flow management.

• Ambient raising capacity up to 55 °C
• Computerized data acquisition & control system
• Oil heating capacity up to 200°C
• Hot air temperature (air cooled): 130 °C

VII Experimental Observation:
There is no ISO or any other relevant standards which specify the guideline on how to measure the cooler performance hence experiment was conducted under laboratory having calibrated instruments and measurement method as per manufacture's own standard. The results of same are display in below table.

Experimental Observations	Design expectations
Cold fluid temperature differences : 8° C	Cold fluid temperature differences : 10° C
Limiting ambient temperature : 50 ° C	Limiting ambient temperature : 46 ° C
Total flow generated by blower is 3275@ static pressure rise 0.05" PSI	Expected flow 3250 CFM @ static pressure drop 0.07 PSI
Total flow of 3275 CFM, oil cooler takes 72% & remaining 25% passes through after cooler & 3% is leakages in system	For a total flow of 3250CFM, oil cooler takes a flow 76% & 24% passes through after cooler
Velocity of air flow over coolers is 5.8 m/s	Velocity of air flow over coolers is 5.5 m/s

Table.2 Experimental Performance of coole

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

VIII Result and analysis:

• The design requirements for airflow was set as 3250 CFM while actual flow generated is 3275 CFM and same is predicted as 3400 cfm from CFD , which indicated the 0.7692 % of deviation in design and experimental values while it is 4.6153 with respect to CFD which is same for diff in CFD and experimental value

• Air flow disruption was set as 72 & 27 % for oil and air cooler respectively which is having 5.2631 % and 3.9473 % different in experimental Vs design and CFD vs. design out came. While same is observed as 0.3888 % of gap for oil cooler in both these approaches.

• Cooling air velocity is having delta of 5.4545 % & 3.6363% in experimental and analytical solution. The differences in these approaches for cooling air velocity is noted as 8.6206 %

• The pressure drop across the cross sectional area of cooler is exactly matching with design expectation and same was predicted as 0.047 PSI which shows 6% of deviation in computational analysis.

X. Conclusion:

In current work the numerical method described in the above section is adopted to solve mathematical model the variation in results are observed in the range of 4 – 6 %. The prime factors due to which we observed differences are.

• The Oil path and air path in cooler are having stream line flow while in actual case it is not.

• The pressure drop varies from point to point in actual study while for simplification same is made as constant throughout cooler or an individual component.

• Temperature increases with respect to time due to repetitive nature of compression cycle and same is define as max constant temp in coolers while solving thermal performance in computational analysis,

• Voltage variations to electric cooling fan speed variation are not considered during computation study.

REFERENCE

[1]Arbon, I. M. The Design and Application of Rotary Twin-Shaft Compressors in the Oil and the Gas Process Industry, Mechanical Engineering Publication Limited, London (1994) | [2] Singh, P.J., Bowman, J.L. Effect of Design Parameters on Oil Flooded Screw Compressor Performance, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1986), Vol.I, 71-88 | [3]Stosic, N., Smith, I., Kovacevic, A. Screw Compressors: Mathematical Modeling and Performance Calculation, Springer (2005) | [4]Sjoholm, L. Important Parameters for Small Twin Screw Refrigeration Compressors, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1986), Vol.I, 1129-1144 | [5]H.F. Li, J.Y. Tu, and A. Subic. The 3-Dimensional CFD Analysis of Flow Behaviors in a Complex Twin Screw Supercharger | [6]Ahmed Kovacevic., Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors | [7]Mccreath, P., Stosic, N., Smith, I.K., Kovacevic, A. The Design of Efficient Screw Compressors for Delivery of Dry Air, International Conference, Compressors and their Systems, London, (2001) | [8]Chanukya Reddy, CFD studies on flow through screw compressor, PhD thesis, NIT Rourkela, 2007. | [9] Nagam Seshaiah, Experimental and Computational Studies on Oil Injected Twin-Screw Compressor, PhD thesis, NIT Rourkela, 2006.