TECHNICAL ARTICLE—PEER-REVIEWED

Numerical Analysis of Hot Air Recirculation Phenomenon in a Compressor Unit and Prevention of Overheating

A. M. Khalaf · I. Barsoum

Abstract Managing the hot air recirculation and flow pattern in air-cooled systems is essential for satisfactory performance and reliable operation of compressor units. In this study, a numerical analysis of the exhaust air from an air-cooled compressor package located in oil and gas off-shore platform is presented using computational fluid dynamics. The results of the analysis reveal that hot air recirculation can have a direct impact in air-cooled units, which is associated with the installation arrangement of the mechanical packages involving air coolers and the environmental conditions around the units. The results were able to predict flow and temperature fields in the compressor area, and provided recommendation to avoid exhaust air recirculation, which if not addressed and prevented can greatly reduce the cooling effectiveness of the air coolers leading to less reliable systems.

Keywords Air-cooled · Heat exchanger · Compressor · CFD · Hot air · Recirculation

List of Symbols

ρ	Density (kg/m^3)
$\overline{u}$	Velocity vector (m/s)
g	Gravitational constant (9.81 m/s^2)
μ	Dynamic viscosity (Pa s)
c_p	Specific heat at constant pressure (J/g K)
$\dot{q}$	Rate of heat generation per unit volume (W/m^3)
K	Thermal conductivity (W m^{-1} K^{-1})
β	Thermal expansion coefficient (K^{-1})
Φ	Viscous dissipate rate (J/kg s)
R	Universal gas constant = 8.314472 J/K.mol
$\dot{m}$	Mass flow (kg/s)
T	Temperature (K)
ΔT	Change in temperature (K)
T_{in}	Cooler fan inlet temperature (K)
T_{out}	Cooler fan outlet temperature (K)
T_a	Atmospheric ambient temperature (K)
F_z	Buoyancy force (N)
R_{HAR}	Hot air recirculation ratio
$\overline{R}_{HAR}$	Average hot air recirculation ratio

A. M. Khalaf
Central Engineering Department, CED, Abu Dhabi National Oil Company (ADNOC Offshore), P.O. Box 303, Abu Dhabi, UAE
e-mail: amkhalaf@adnoc.ae

I. Barsoum (✉)
Department of Mechanical Engineering, Khalifa University of Science and Technology, P.O. Box 2533, Abu Dhabi, UAE
e-mail: imad.barsoum@ku.ac.ae

Introduction

Air-cooled heat exchangers (ACHE) are commonly used in industrial applications such as electrical power generation plants, petrochemical processing areas, and in data centers involving high-temperature IT equipment [1–4] due to economic and environmental considerations [5]. The thermal management of critical equipment including air-cooled condensers in power plants, power transformers, and gas compression systems is crucial, and the associated cooling systems highly contribute to the reliability and efficiency of the overall plant [6–8]. Although air-cooled systems may require less cost compared to water-cooled systems, their optimum performance will depend on several factors such as wind direction and speed, flow obstructions, environmental conditions, i.e., changes in ambient conditions, and hot air recirculation [1]. All these factors have a significant effect on the heat transfer ability, especially in controlling

the cooling effectiveness. Hot air recirculation phenomenon is usually not considered during the design phase and its influence on the heat transfer ability will only be observed during operation, which in many instances may lead the system to trip causing serious business interruptions [6–8]. Therefore, understanding hot air recirculation and flow patterns around air coolers is of great importance and its consideration for safe and efficient operation can be crucial.

Du Toit et al. [9] found that the heat transfer ability of ACHE is based on the local flow conditions, and the impact of inlet conditions on fan performance and air recirculation needs to be taken into account. Gu et al. [10, 11] studied the hot air recirculation on the direct air-cooled systems for a major power plant using wind tunnel simulations. Their results revealed that due to the interference of nearby buildings, the flow incident angle have a significant effect on the air-cooled condenser's efficiency. Kroger [12] found that flow interactions between ACHE and neighboring buildings and structures can considerably impact the flow patterns leading to reduced plant performance. Du Toit et al. [13] and Duvenhage et al. [14] studied the recirculation phenomenon in windless conditions and obtained an empirical correlation to predict the plume recirculation for different draft air cooler geometries. Staden and Prestorius [15, 16] numerically predicted the impact of varying ambient conditions and recirculation on fan performance and found that the cooling efficiency of air-cooled condensers is very sensitive to wind speed and direction. The influence of wind direction and hot air recirculation on fan cooler performance was also analyzed by Duvenhage and Kroger [17], who found that wind direction can significantly reduce the airflow rate into the fan leading to reduced volumetric efficiency. Wind along the longitudinal axis causes an increased rate of hot plume recirculation leading to increased air inlet temperature to the fan cooler, which in turn will subsequently reduce the coolers heat transfer effectiveness. Similarly, Rooyen and Kroger [18] numerically studied the effect of hot plume recirculation on the performance of axial fans. Ramani et al. [19] demonstrated the effect of wind direction on hot air recirculation and its impact on the inlet boundary condition of the fan coolers for an air-cooled condenser unit. The results showed that hot air recirculation increases the inlet temperature of the fans which in turn leads to reduced heat transfer capacity of the air-cooled condenser, and hence lower overall plant efficiency. Many experimental and numerical studies have been carried out to investigate the thermal flow behavior of air-cooled systems [20–23], and it was found that computational fluid dynamics (CFD) can be a useful approach to numerically simulate hot air recirculation phenomenon due to techno-economically reasons [18, 22].

The current work presents a case study of hot air recirculation phenomenon that impacted the performance of an air-cooled instrument air compressor package at an oil and gas offshore platform. The inadequate hot air management of the system caused frequent system break offs and outings due to increased temperature, which in turn led to serious operational interruptions. The paper studies the overall flow distribution and temperature field around the compressor package area by means of CFD simulations. Based on the results, proposed mitigation measures were analyzed to avoid hot air recirculation with the aim to determine a guideline and industry best practice for similar installations. Actual plant data were also collected, analyzed, and compared to CFD predictions.

Layout of the System

The simplified schematic diagram shown in Fig. 1 illustrates the airflow within the compressor package. The air compressor unit is a double-stage, gear-driven oil-free screw compressor type. The cooler fan draws fresh air to the compressor package enclosure via the suction/air intake louvers. The cooling air flows into an integral ACHE in order to cool the compressed air and lube oil within the package, before exiting the package through hot air exhaust louvers. The drawn atmospheric air will also travel into the first-stage compressor after passing through air filters. The compressed air is discharged to an intercooler, as part of the ACHE. The cooled compressed air will then enter a second-stage compressor for further compression to achieve the duty pressure. The high pressure air will finally enter an aftercooler, part of the ACHE, to cool the compressed air and will then be distributed for process use.

Computational Fluid Dynamics Simulations

Governing Equations

The governing equation of the flow is described by Navier–Stokes equation and given by the following equations in vector form

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overline{\boldsymbol{u}}) = 0 \qquad (Eq\ 1)$$

$$\rho \frac{D(\overline{\boldsymbol{u}})}{Dt} = \rho \overline{\boldsymbol{g}} - \nabla p - \frac{2}{3} \nabla \cdot (\mu \nabla \cdot \overline{\boldsymbol{u}}) + \nabla \cdot \left[\mu \left(\nabla \overline{\boldsymbol{u}} + (\nabla \overline{\boldsymbol{u}})^T \right) \right] \qquad (Eq\ 2)$$

$$\rho c_{\mathrm{p}} \frac{D(T)}{Dt} = \rho \dot{q}_g + \nabla \cdot (k \nabla T) + \beta T \frac{D(p)}{Dt} + \varphi \qquad (Eq\ 3)$$
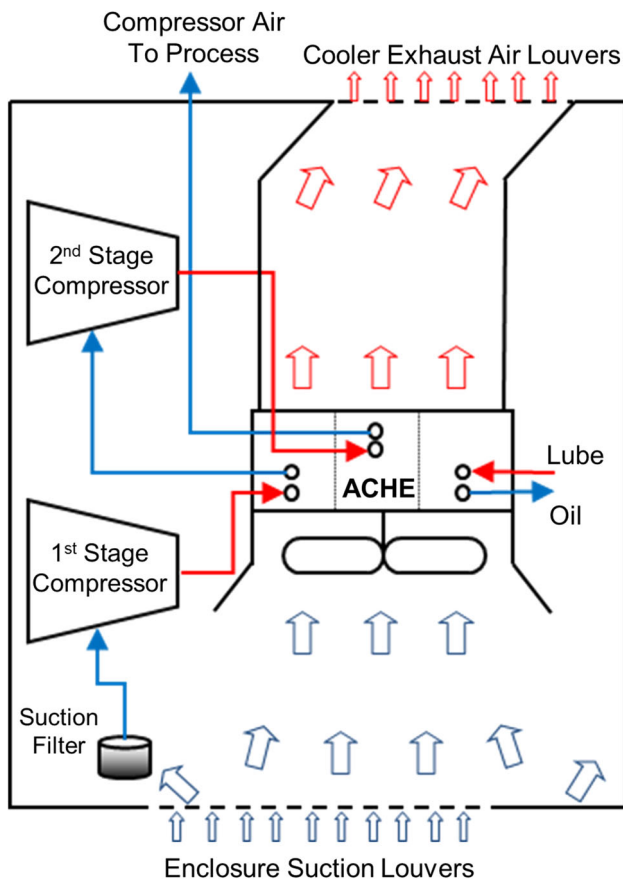
$$\rho = \rho RT \qquad (Eq\ 4)$$

Fig. 1 Schematic of the air compressor package with the ACHE

where Eq. 1 pertains to mass conversation, Eq. 2 momentum conservation, Eq. 3 energy conservation and Eq. 4 equation of state. Here, ρ and $\overline{u}$ are the density and velocity vector of air, $g = 9.81$ (m/s^2) is the gravity vector constant, ∇ is the divergence operator, μ is the dynamic velocity, c_p is the specific heat of air, $\dot{q}_g$ is the rate of heat generation, k is the thermal conductivity, β is the thermal expansion coefficient, φ is the viscous dissipation rate, $R = 8.31447$ (J/K mol) is the universal gas constant, T is the temperature and p is the pressure.

Although the presented problem is governed by laminar flow around the package, which is verified in the results section by examining the post-processing turbulent parameters such as turbulent kinetic energy, the flow type in this simulation is selected to capture laminar and turbulent flow behaviors of the air since SolidWorks flow simulation [24] is able to consider both flow regimes simultaneously. To predict turbulent flows, the Favre-averaged Navier–Stokes equations are used, in which extra terms known as the Reynolds stresses appear in the equations. Accordingly, the transport equations for the turbulent kinetic energy and its dissipation rate, using modified k–ε model [23] are employed.

The buoyancy force effect on the air can be explained by means of Boussinesq model [4], in which its effect is approximated in the momentum equation by

$$F_z = g(\rho - \rho_o) \tag{Eq 5}$$

Computational Fluid Dynamics Model

A detailed three-dimensional computational fluid dynamics (CFD) model was built using SolidWorks flow simulation [24] to simulate the phenomenon of hot air recirculation and its influence on the overall performance of the compressor unit by analyzing the airflow and temperature distribution around the unit. The geometry of the CFD model was constructed to represent the physical model of the air compressor unit and its surrounding area, as shown in Fig. 2. The size of the computational domain was determined to be sufficiently far away from the compressor unit in order to represent the boundary conditions realistically and therefore obtain accurate numerical predictions.

Internal flow analysis type was used in the model, where air is defined as the working fluid, with initial parameters corresponding to atmospheric ambient summer conditions at a pressure of 101.3 kPa, temperature of 41°C, and 50% relative humidity. To account for turbulent flow regime, the modified k–ε model employed in SolidWorks is used with the default initial turbulent kinetic energy of 1 J/kg and a turbulent dissipation rate of 1 W/kg. No slip condition is assumed at all solid walls in the computational domain and regarding air as a Newtonian fluid. Atmospheric ambient conditions are applied to open areas such as walkways and the stairs. It is worth noting that the air condition at the atmospheric boundaries is considered windless, i.e., effect of wind speed and direction is not taken into account. The enclosure walls of the compressor package are defined to be adiabatic due to the fact that insulation foam is applied on the internal side of the walls. The ACHE fan flow is defined to be 9.43 m^3/s, and the ACHE heat load is set to be 300 kW. The heat load input to the model was verified by site measurements utilizing the following the specific heat equation given by

$$Q = \dot{m}c_p\Delta T \tag{Eq 6}$$

where $\dot{m}$ is the mass flow and ΔT is the temperature difference.

Since the aim of this study is to investigate the hot air recirculation in the enclosed area, recirculation ratio R_{HAR} defined by Zhao [6, 7] will be utilized to assess the results, as given by

$$R_{HAR} = \frac{T_{in} - T_a}{T_{out} - T_a} \tag{Eq 7}$$

Fig. 2 Illustration of the computational domain: (**a**) CFD model and (**b**) actual compressor
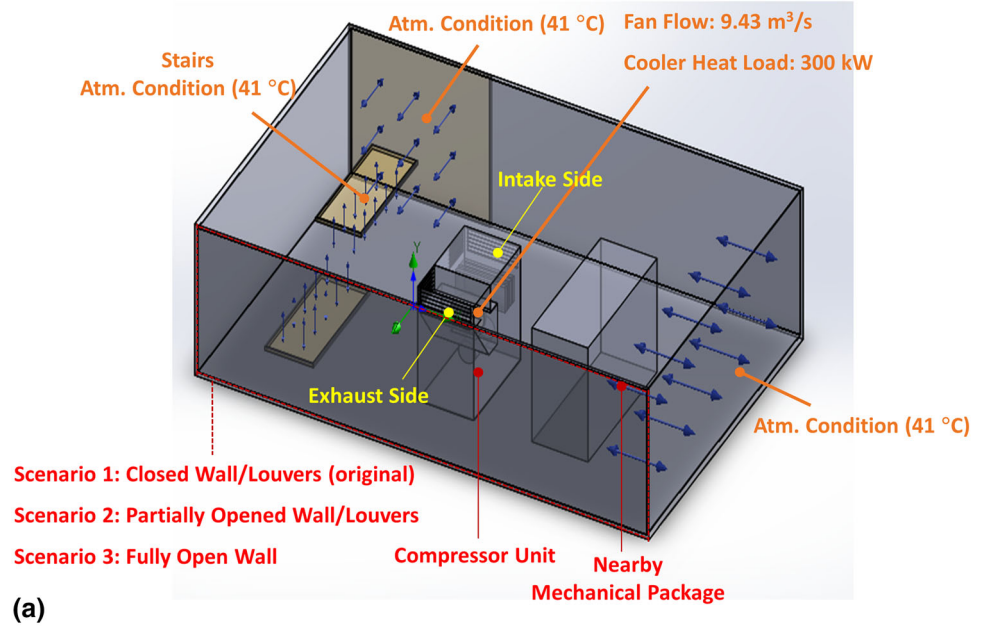

(a)

(b)

Fig. 3 Comparison between CFD model and actual site temperature measurements
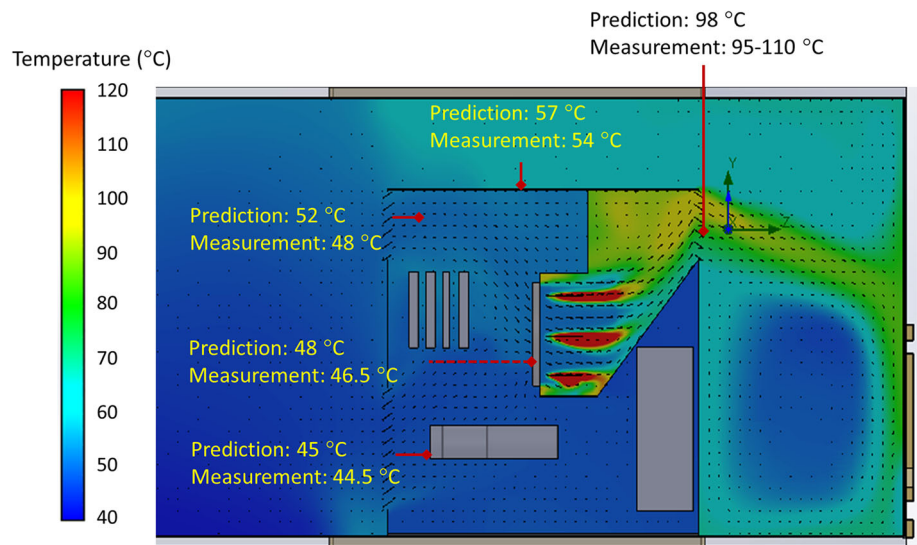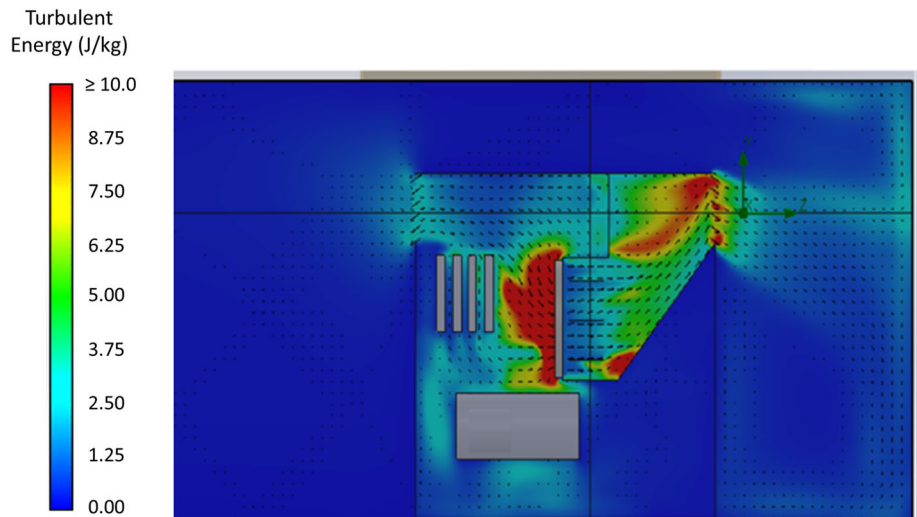
Fig. 4 Turbulent kinetic energy around the compressor unit

$$R_{HAR}^{ave} = \sum_{i=1}^{N} \frac{R}{N} \qquad \text{(Eq 8)}$$

where T_{in} is the cooler fan inlet temperature, T_{out} is the cooler fan outlet temperature, T_a is the atmospheric ambient temperature, and R_{HAR}^{ave} is the average hot air recirculation ratio.

Mesh was generated by dividing the predefined computational domain into set of rectangular cells called cuboids, with enabled fluid cell refinement and narrow channel refinement options. The basic initial mesh has 58, 15, and 39 number of cells in the X, Y, and Z direction, respectively. Level 3 global domain refinement was selected to ensure mesh convergence is achieved. In order to ensure that steady-state condition is achieved, average temperature and velocity of the air were selected as goals for convergence control.

Model Validation

The CFD model was validated against site temperature measurements at different locations around the compressor package (i.e., Scenario 2 in Fig. 5) since actual site measurements were available for this site installation. It can be seen from Fig. 3 that the CFD model is in good agreement with actual site installation measurements, in terms of average air temperature. The difference in temperature values is approximately 2 to 7 °C, with a relative error of about 5 to 9%, which is considered acceptable for the scope of this study.

Figure 4 shows the turbulent kinetic energy around the compressor unit. It can be depicted that the turbulent kinetic energy away from the package is almost zero, which means that the flow is dominantly laminar away from the package. Near the package outside walls, the turbulent kinetic energy is approximately 0.7–2.5 (J/kg) which is close to the default initial value assigned. The turbulent kinetic energy is very high inside the package which is essentially due to the significant changes in geometry and high velocity resulting from fan flow.

Results and Discussion

In this section, the CFD simulation results of the exhaust hot air recirculation for the compressor package are presented and discussed. The flow and temperature fields of three scenarios were modelled in this study under different installation configurations, as shown in Fig. 5. Scenario 1 represents the original site installation, where the platform boundaries at the compressor package exhaust side are totally enclosed by walls and louvers. Scenario 2 represents installation of the compressor package with additional suction openings while partially opening the platform louvers. This scenario is in fact implemented by site personnel to enhance ventilation on the package and its surrounding area to reduce the risk of system trips due to high temperature. Lastly, Scenario 3 is similar to Scenario 2 with the platform louvers fully eliminated as an attempt to avoid exhaust air recirculation.

In order to assess the impact of hot air recirculation on the ACHE of the compressor unit, the temperature field obtained from the CFD results is shown in Fig. 6. It is worth noting that all contour plots are displayed at the compressor unit exhaust louvers center line. As revealed from these results, Scenario 1 experiences the highest temperature field around the compressor package. This is due to the fact that hot exhaust air is restricted from dispersing as a result of fully enclosed platform walls. A noticeable improvement on the temperature field of the
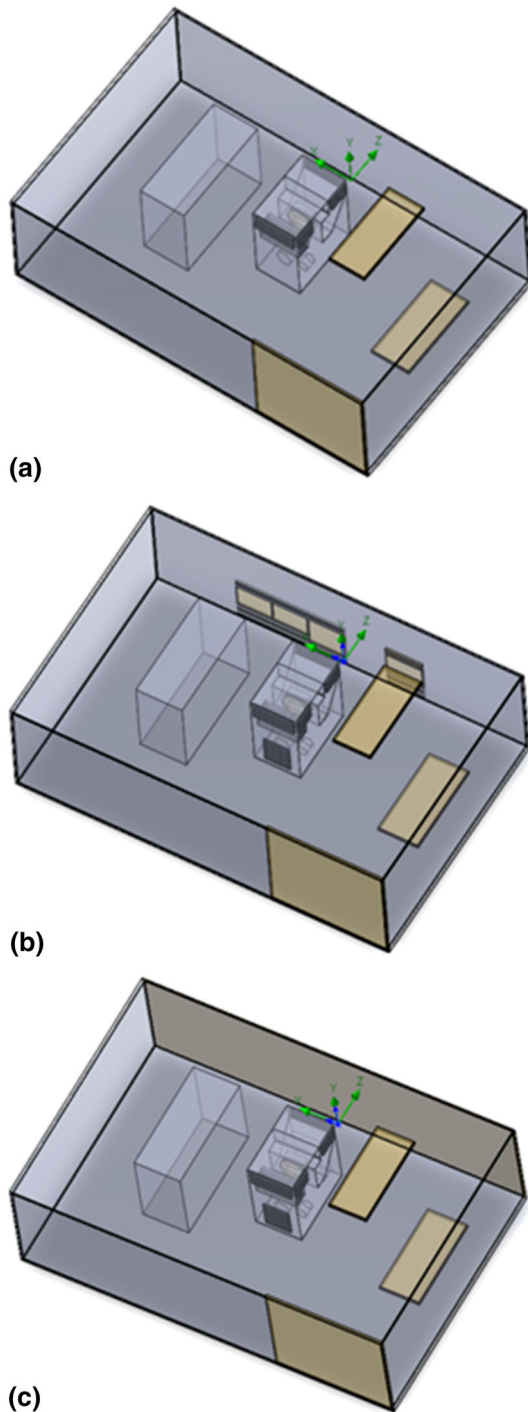
1110

J Fail. Anal. and Preven. (2019) 19:1105–1112

Fig. 5 Three scenarios considered: (**a**) Scenario 1, (**b**) Scenario 2 and (**c**) Scenario 3

compressor unit area is observed for Scenario 2. This is due to the partial opening of the platform louvers at the exhaust side, where the temperature around the compressor unit is found to be lower. Moreover, the additional suction louvers

at the intake side of the compressor unit increase the intake of relatively colder air, which in turn reduces the effect of hot air recirculation on the ACHE. Scenario 3 is a hypothetical case, which mimics the situation when the compressor unit is installed in open area, i.e., the fully opened exhaust side. It is clear that hot air will not accumulate in the area, and the overall temperature is corresponding to the assigned ambient condition.

The hot air recirculation ratio R_{HAR} is calculated using Eq. 7 and found to be 26, 21, and 0% for Scenarios 1, 2 and 3, respectively. Higher R_{HAR} indicates that higher temperature rise (i.e., above the ambient temperature) at the intake of the ACHE is taking place. This leads to a lower heat rejection rate of the cooler resulting in reduced cooling efficiency. Hence, the temperature of the cooled medium tends to rise reaching a trip set-point. Frequent system trips will result in reduced system availability and reliability leading to serious operational interruptions.

It can be seen from Fig. 6 that the air temperature is higher at the upper part of the compressor package. One main reason to this behavior is due to the buoyancy effect, where hot air tends to move upwards due to the lower density, as illustrated in Fig. 7a. This can be explained by means of Boussinesq model approximated by Eq. 5. In addition, the velocity stream lines illustrated in Fig. 7b show that due to the obstruction of airflow by the platform walls, a large region of wake flow is formed. In the wake region, part of the air deflects downward forming a vortex and hence sweeps the platform ground in a reverse direction. This wake flow and circulation generally form a negative pressure that may draw hot air downward. In contrast to this typical flow behavior, negative pressure was not experienced, which is possibly due to large flow domain associated with small pressure gradient in the area, i.e., atmospheric condition. However, these wake formation can alter the overall flow distribution leading to higher exhaust hot air recirculation.

According to the actual site observations and numerical predictions, the extent of hot air recirculation has a direct influence on the ACHE efficiency and the overall system reliability and therefore needs to be considered and controlled. This can be achieved by either placing the air cooler packages in an open area, while taking into account the wind effect in terms of speed and direction, or installing exhaust duct work to route the hot air away from the air cooler intake. Exhaust ducting was successfully implemented as a remedy at site and as guided by the results in this case study, which prevented any hot air recirculation and system trip was completely eliminated.

J Fail. Anal. and Preven. (2019) 19:1105–1112

1111

Fig. 6 Side and top view of the air temperature distribution for scenario 1, 2 and 3
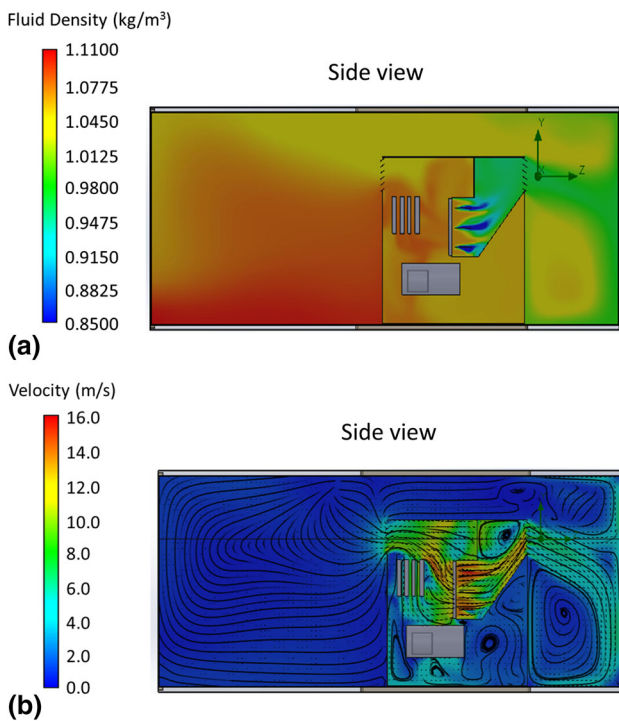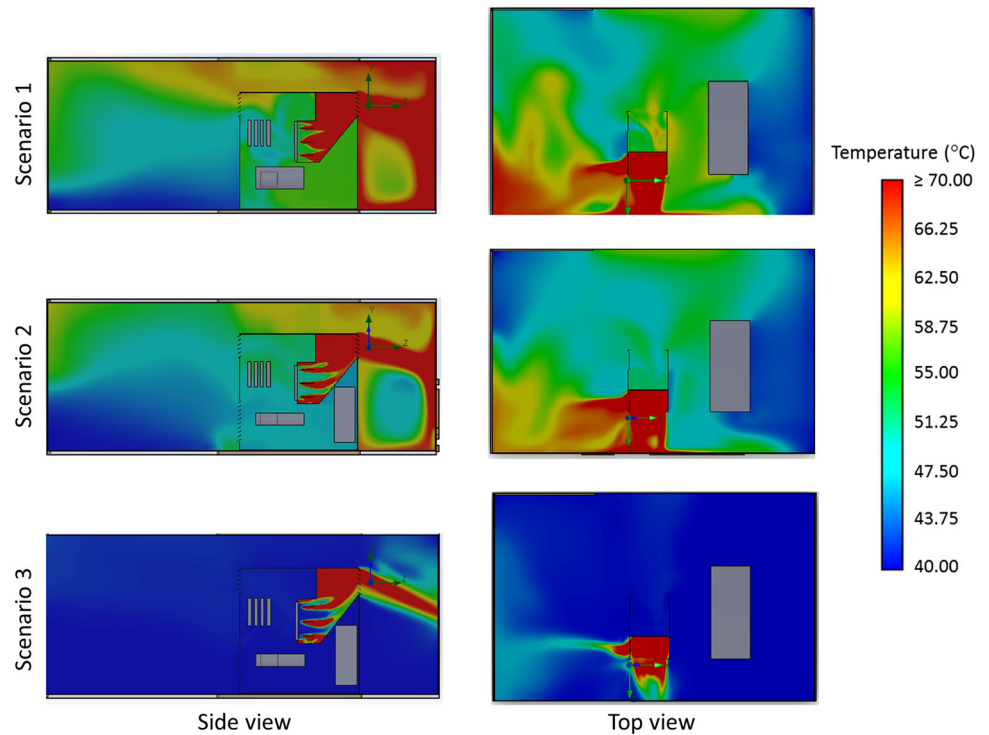

Fig. 7 Contour plots for Scenario 1: (**a**) fluid density and (**b**) fluid velocity

Conclusions

In the current study, an investigation of hot air recirculation phenomenon in an air-cooled compressor package located in oil and gas offshore platform is presented. Three different installation scenarios are analyzed using CFD simulations. Numerical predictions show that the ratio of air recirculation has a direct impact on the inlet conditions of air coolers, which can lead to reduced heat rejection rate and efficiency. The outcomes of this assessment were able to predict flow and temperature behavior in the compressor area and provided recommendation to avoid exhaust air recirculation. As a remedy motivated by the numerical results, exhaust ducts were installed to route the hot exhaust air away from the air cooler intake, which resolved hot air recirculation and eliminated system trips due to overheating completely.

References

1. A. Patel, J. Sudani, Review on wind effect and axial fan performance of air cooled condenser for a thermal power plant. Int. J. Adv. Eng. Res. Dev. **4**(7), 2348 (2017)
2. S. Park, J. Seo, Analysis of Air-side economizers in terms of cooling-energy performance in a data center considering Exhaust air recirculation. Energies **11**, 444 (2018)

1112

J Fail. Anal. and Preven. (2019) 19:1105–1112

3. W. Zhao, Q. Wang, P. Liu, The experimental investigation of recirculation of air-cooled system for a large power plant. Energy Power Eng. **2**, 291–297 (2010)

4. L. Borghei, R.M. Khoshkhoo, Wind effects on air-cooled condenser performance. New Asp. Fluid Mech. Heat Transf. Environ. (2011)

5. C. Butler, R. Grimes, The effect of wind on the optimal design and performance of a modular air-cooled condenser for a concentrated solar power plant. Energy **68**, 886 (2014)

6. W.L. Zhao, P.Q. Liu, The reasons of recirculation produce and it's evaluate criteria for air-cooed power pant, in *Proceedings of Chinese Congress of Theoretical and Applied Mechanics in memory of the Golden Jubilee of CSTAM*, Beijing (2007), p. 143

7. W.L. Zhao, P.Q. Liu, Experimental researches of the effect of environmental wind on thermal recirculation under the tower of direct air cooled system. J. Power Eng. **23**(3), 390–394 (2008)

8. P.Q. Liu, W.L. Zhao, Z.L. Xu, Simulation and experimental study of the wind tunnel thermal effect of a directly air cooled system in a thermal power plant. J. Thermal Energy Power **28**(3), 240–243 (2008)

9. C.G. Du Toit, D.G. Kroger, Analysis of recirculation in mechanical-draught heat exchangers, in *Proceedings of the 3rd World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics* (1996)

10. Z.F. Gu, H. Li, Wind tunnel simulation on re-circulation of air-cooled condensers of a power plant. J. Wind Eng. Ind. Aerodyn. **93**, 509–520 (2005)

11. Z.F. Gu, X.R. Chen, Wind tunnel simulation of exhaust recirculation in an air-cooling system at a large power plant. Int. J. Therm. Sci. **46**(5), 308–317 (2007)

12. D. Kroger, *Air-Cooled Heat Exchangers and Cooling Towers Thermal Flow Performance Evaluation And Design* (Penn Well Corporation, Tulsa, 2004)

13. C.G. Du Toit, D.G. Kroger, Modeling of the recirculation in mechanical draught heat exchangers. S. Afr. Inst. Mech. Eng. R DJ **9**, 2–8 (1993)

14. K. Duvenhage, D.G. Kroger, Plume recirculation in mechanical draught heat exchangers. Heat Transfer Eng. **16**(4), 42–49 (1995)

15. V. Staden, Numerical modelling of the effects of ambient conditions on large power station air cooled steam condensers. Am. Soc. Mech. Eng. **221**, 145–150 (1995)

16. V. Staden, L. Pretorius, Simulation of heat exchange in large air cooled condensers, heat transfer, in *Proceedings of 11th IHTC*, vol. 6 (1998), pp. 155–160

17. K. Duvenhage, D.G. Kroger, The influence of wind on the performance of forced draught air-cooled heat exchangers. J. Wind Eng. Aero **63**(2–3), 259 (1996)

18. J.A. Rooyen, D.G. Kroger, Performance trends of an air-cooled steam condenser under windy conditions. J. Eng. Gas Turbines Power **130**(2), 023006 (2008)

19. A. Ramani, B.A. Paul, D.A. Saparia, Performance characteristics of an air-cooled condenser under ambient conditions, in *International Conference on Current Trends In Technology* (2011)

20. K. Duvenhage, J.A. Vermeulen, C.J. Meyer, D.G. Kroger, Flow distortions at the fan inlet of forced draught air-cooled heat exchangers. Appl. Therm. Eng. **16**(8/9), 741–752 (2009)

21. H.W. Krus, J.O. Haanstra, R.V. Derham, B.W. Schreur, Numerical simulations of wind measurements at amsterdam airport schipol. J. Wind Eng. Ind. Aerodyn. **91**, 1215–1223 (2003)

22. W.S. Zhao, M. Lei, S.L. Wang, N. Cui, Y. Liu, Numerical simulation and analysis of the hot air recirculation phenomenon observed in direct air-cooled system. Proc. Third IEEE Conf. Ind. Electron. Appl. **1–3**, 1624–1628 (2008)

23. J.R. Bredell, D.G. Kroger, G.D. Thiart, Numerical investigation of fan performance in a forced draft air-cooled heat exchangers. J. Appl. Therm. Eng. **26**, 846–852 (2006)

24. Solidworks Flow Simulation v.2014, Dassault Systèmes SolidWorks Corp