CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong

ISBN 978-88-95608-30-3; **ISSN** 2283-9216

DOI: 10.3303/CET1439226

Computational Fluid Dynamics Investigation of Air Cooled Heat Exchangers

William G. Osley*[a], Peter Drögemüller[a], Peter Ellerby[a], Ian Gibbard[b]

[a]Cal Gavin Limited, Minerva Mill Innovation Centre, Station Road, Alcester, Warwickshire, B49 5ET, UK
[b]Progressive Thermal Engineering, 58 Alcester Road, Studley, B80 7NP, UK
william.osley@calgavin.com

Cooling of process streams is a standard operation in many industries, and where possible water cooling is the most cost effective solution. However in areas where water supply is limited Air Cooled Heat Exchangers (ACHE's) are often the only alternative. The overall coefficient maybe limited by the air side heat transfer. To improve this coefficient the air flow should be evenly distributed and at as high velocity as possible over the whole bundle. The limits are set by the fan power consumption and the generated noise level. The required fan power forms a significant part of running costs. In order to keep these costs as low as possible knowledge concerning the flow distributions within the bundle can be important.

Fluid bypass within the tube-side can be of similar importance. In design calculations an equal fluid flow velocity per tube is typically assumed from the total inlet mass flow. However if some of the fluid bypasses the tube bundle due to poor header design lower than expected fluid velocities in the tubes will be found, leading to underperforming ACHE's, since heat transfer and fouling behaviour are influenced by the fluid velocity.

Computational Fluid Dynamics (CFD) has been found to be a powerful tool to investigate how fluid flows through a defined geometry. In this paper CFD techniques will be used to investigate the tube side and air side flow of ACHE's. The main focus will be on finding areas of fluid maldistribution within the air side and the effect of bypass on the tube-side of ACHE's.

As a benchmark the pressure drop results of CFD simulations were compared to correlations available in the public domain and agreed well, giving confidence in the fluid flow patterns produced by the CFD simulations.

The effect of the tube-side bypass was found to reduce the duty of the ACHE by 27 % in the worst case scenario that was tested. Maldistribution of the air-side flow was found in the ACHE, with the number of fans and depth of the plenum influencing the amount seen.

1. Introduction

In this paper two different Air Cooled Heat Exchanger (ACHE) case studies will be presented. ACHE's typically consist of a rectangular bundle of tubes, with the tubes arranged in rows. Ambient air is forced in cross flow over the outside of the tubes by axial fans, cooling the tube side fluid. Computational Fluid Dynamics (CFD) will be used to investigate fluid flow around ACHE geometries. CFD has been found to be a powerful tool to investigate how fluid flows through a defined geometry.

The first case study investigates bypass flow within the header of a two pass ACHE bundle caused by an oversized vent hole in the pass partition. (The sizing of the heat transfer area is based on thermal design calculations assuming that all the flow entering the exchanger flows through the bundle). However if due to poor header design there is bypass flow less mass will flow through the tube bundle. (This has consequences in terms of fluid velocity, heat transfer and driving temperature differences this can lead to severe underperforming of the ACHE, since heat transfer and fouling behaviour are influenced by those parameters). CFD has been used to investigate tube-side flow maldistribution in ACHE by Habib, et *al* (2009). They investigated the effect that the nozzle size and position had on the fluid distribution within the

header. They found that the number of nozzles had the greatest influence on maldistribution. Long Huang et al. (2014) used CFD to investigate maldistribution in micro channel heat exchanger headers.
The second case study investigates the air-side distribution within an ACHE's plenum. Often the overall heat transfer is limited by the air side heat transfer. To maximise the outside heat transfer the air flow should be evenly distributed and at as high velocity as possible over the whole bundle. However in reality it may not be possible to achieve an even velocity distribution and this paper will use CFD to investigate the air distribution after the fan inlet for possible maldistribution within plenum and tube bundle in several ACHE geometries. Maldistribution could lead the ACHE to underperform as it is not normally taken into account in the design calculations. Bredell et al. (2006) have used CFD to investigate air flow into ACHEs. They found maldistribution of air flow into the fan inlets, resulting in poor fan performance. Chu et al. (2013) used CFD to investigate air-side flow through a finned tube bundle using a small section,

2. CFD Models

The ACHE geometries for both case studies were drawn on to the computer using ANSYS DesignModeler software and then meshed using ANSYS meshing. Mesh independences studies were carried out to find an accurate mesh for the each geometry. The CFD solver used was ANSYS CFX.

3. Case Study 1 – Tube-side Header Bypass

3.1 CFD simulation scenarios

This ACHE is operating as a lube oil cooler with laminar flow conditions due to the high viscosity of the lube oil. The geometry of the ACHE is a two pass arrangement. The tubes are equipped internally with hiTRAN wire Matrix elements in order to boost the tube side heat transfer in laminar flow conditions. The bundle consists of 54 high fin tubes per pass; the base tube has 25.4 mm outside diameter, 22.1 mm internal diameter and is 8,700 mm long arranged in four staggered rows of transvers pitch 66.6 mm and longitudinal pitch 57.8 mm.
For high operation pressures ACHE are often designed as plug head type exchanger. The thickness of the header plates are typically in the region from 10 mm to 30 mm. In order to prevent fluid bypassing the pass partitions are welded where possible. Not every part of the header inside is accessible for welding. Different sealing methods are used in order to prevent fluid bypassing. In the worst cases the partition plate is just sitting flush on the header casing without seal, for those designs small longitudinal gaps are possible. The other possibilities for bypass are vent holes between the passes. These holes are used in order to vent the headers and when draining the tube side fluid. They should be designed in such a way that under normal operating conditions minimum liquid bypasses the tubes. When sizing these holes pressure drop across the pass partitions and liquid viscosity are the main variables. Manufacturers often use only one common hole size, not sized for individual process conditions. Bypassing can cause severe underperformance, because only a fraction of the liquid is exposed to the heat exchange area. There is also the danger that the reduced flow causes a pinch in the temperature difference between hot fluid and cooling air.

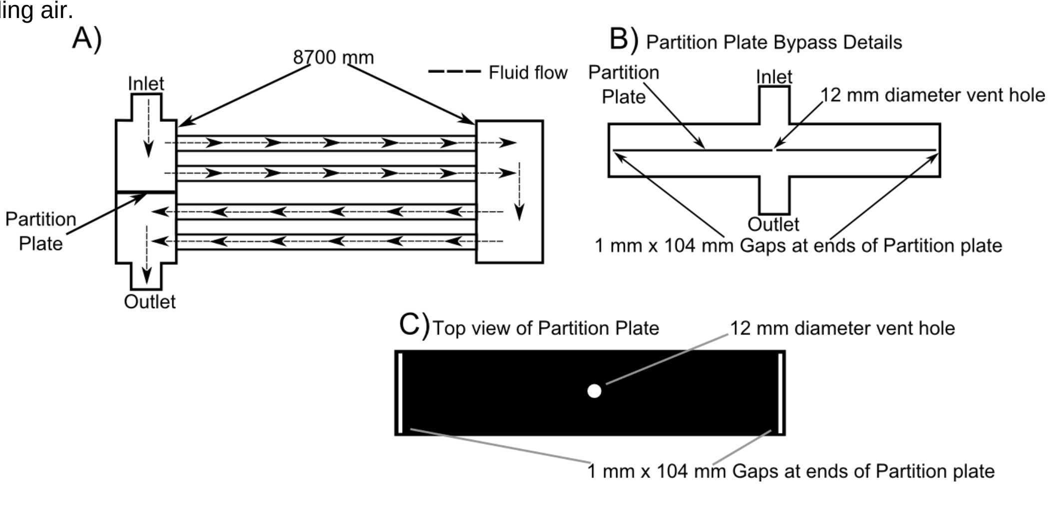

Figure 1: Tube-side geometry, A) pass arrangement B) location of bypass in inlet/outlet header and C) view of partition plate from above showing location of bypass (not to scale)

Table 1: Case simulation conditions

Case	Bypass	Mass Flow [kg/s]	Inlet Temperature [°C]	Outside Air Temperature [°C]	Air Face Velocity [m/s]
1	No	3.4	27 (isothermal)		
2	No	5.5	86	35	3.3
3	12mm vent	5.5	86	35	3.3
4	12mm vent + 1 mm gaps	5.5	86	35	3.3

For this study (which is based on a real case) three scenarios are investigated. The first is an ideal situation where there is no bypass of the tubes. The second scenario is where there is a 12 mm diameter vent hole in the centre of the pass partition plate. The third (worst case) scenario is where there is a 12 mm diameter vent hole in the centre of the pass partition and 1 mm wide (104 mm long) gaps have been left between the header wall and the ends of the partition plate. Details of the vent hole and gap positions can be seen in Figure 1. These geometrical conditions mirror a real case where bypassing was a contributing factor for underperformance.

Bypassing is increased with increasing pressure drop between the corresponding passes but an optimised design often requires significant pressure drop. In order to optimise the heat transfer in this design the pressure drop has been fully utilised by using hiTRAN Matrix Elements. As the heat transfer and pressure drop characteristics of hiTRAN are different then empty tubes they have been modelled using porous media.

3.2 CFD model verification and Results

By using CFD to model these three scenarios with the same inlet conditions (see Table 1) the effect of the bypass can be seen. This ACHE operates with an oil inlet temperature of 86 °C and under design conditions the oil is cooled to 59 °C. Fluid properties at inlet temperature are: density – 829 kg/m^3, viscosity 7.8 cP, therefore the Reynolds number of the flow was below 1,000 and a laminar model was used for the CFD simulations.

The isothermal tube side pressure drop of the lube oil flowing through the ACHE (case 1) was used to check the accuracy of the CFD results. It is calculated by adding the frictional pressure drop inside the tube bundle the nozzle pressure losses and the pressure losses from the header tube sheet interface. The pressure drop in the tube bundle was calculated with the software package hiTRAN.SP which is available on request from the insert manufacturer Cal Gavin Ltd. Calculations for the remaining pressure losses can be found in literature, (Sinnott, 2009).

Isothermal test Case 1 (no bypass) CFD calculates a pressure drop of 137 kPa, when using correlation based calculation as outlined above the pressure drop is calculated as 149 kPa. The small deviation of about 8 % is within the expected error and shows that the model is accurately modelling the fluid flow in the header and through the bundle with hiTRAN Elements.

Heat transfer can now be added to the model by applying a heat transfer coefficient in the porous media and on the tube walls. The heat transfer conditions are the same for all three scenarios the only change is the amount of bypass. Using Eq(1) the tube-side duty can be calculated for each case and compared. Q is the tube-side duty (W), $\dot{m}$ is the mass flow through the tubes (kg/s), Cp is the specific heat capacity (J/kg °C) and T is temperature (°C) at the inlet and outlet of the exchanger.

$$Q = \dot{m} * ((C_p out * Tout) - (C_p in * Tin)) \tag{1}$$

The pressure drop and duty results from Case 2 (no bypass) will be used as base case in order to compare the two bypass cases. Results for Case 3 indicate that 20 % of the inlet mass flow is bypassing the tube bundle. This resulted in a pressure drop decrease of 35 % when compared to Case 2. As less fluid is flowing though the tube bundle and not contacting with the heat transfer surface this leads to an 11 % reduction in duty. When Case 4 results are compared to base Case 2 there is an even greater reduction in mass flow of 42 % through the tube bundle. This results in a 64 % reduction in the pressure drop and a 27 % reduction in exchanger duty.

The velocity profiles in the inlet/outlet header of the ACHE are shown in Figure 2 A). The jet of fluid caused by the 12 mm vent hole can clearly be seen and shows the extent of bypassing fluid. In B) the increase fluid bypass caused by the 1 mm gaps is visible along the side and bottom of the outlet side of the header. It can be noticed that fluid is drawn into this gap since it provides less flow resistance compared to the tube

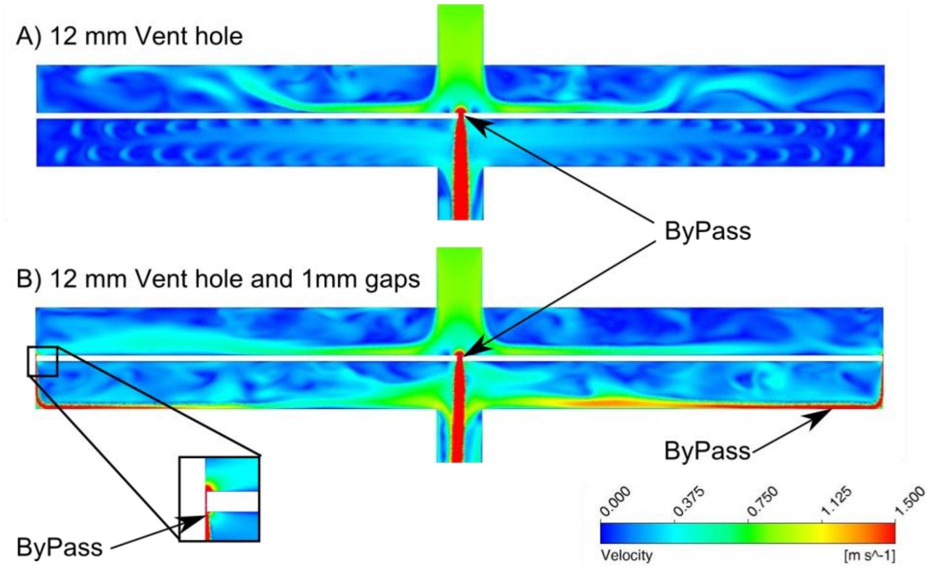

Figure 2: Header Velocity profile for ACHE1 A) 12 mm vent hole in partition plate and B) 12mm vent hole and 1 mm gaps at ends of partition plate

bundle. This clearly shows great care has to be taken when sizing the required vent hole. If possible it is recommended to avoid all together or design alternatives which prevent bypassing.

4. Case Study 2 – Air-side Maldistribution

4.1 CFD simulation scenarios

The purpose of these simulations is to investigate whether fan arrangements for ACHE allowed by the American Petroleum Institute (API) specifications can lead to air flow maldistribution across the tube bundle. API recommends 40 % fan coverage of the bundle in order to maintain sufficient airflow. In this specific case the coverage can be achieved by selecting either three or four fan arrangement see Figure 3. The three fan coverage was used as a base case to investigate the influence of three plenum depths of 500 mm, 720 mm and 1,000 mm had on air distribution.

The tube bundle consist of 70 tubes with 25.4 mm outside diameter and 8,700 mm long, arranged in three staggered rows, with a transvers pitch of 63.5 mm and a longitudinal pitch of 55 mm. The tubes have high fins of 57.15 mm outside diameter (15 mm fin high), 0.4 mm fin thickness and 394 fins per meter. In the CFD model a porous media was used around the tube bundle to mimic the pressure drop induced by the finned tubes. The bundle sits in a bay of width 1,550 mm and length 8,700 mm. A fan diameter of 1,500 mm was used for the three fan case and 1,300 mm diameter for the four fan case, each giving the recommended 40 % fan coverage, Serth and Lestian (2014).

All the simulations are carried out isothermally at 25 °C, using the k-ε turbulence model, with compressible air as the fluid (density changes due to the ideal gas equation) and total air volume flow of 48.5 m^3/s, that equates to a face velocity of about 3.6 m/s. It was assumed that the fan hub takes up 25 % of the fan inlet area and that the fans are spinning at 800 rpm. By using expressions for each inlet velocity component the velocity profile of an axial fan can be implemented without having to explicitly model the fan.

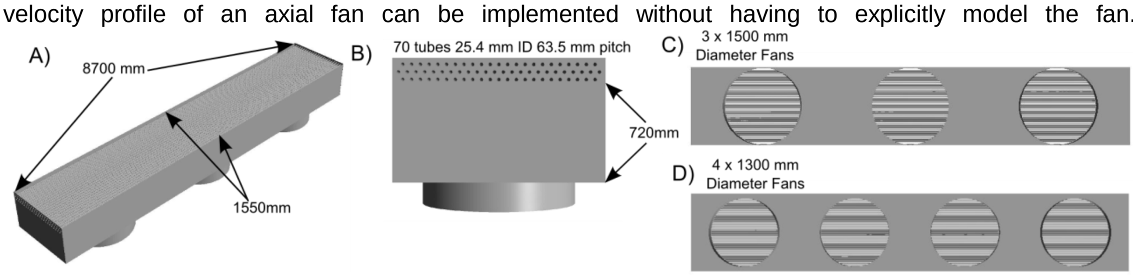

Figure 3: Air-side geometry, A) bay length and width, B) plenum height, C) three fan layout and D) four fan layout

4.2 CFD Model Verification and Results

Due to the lack of experimental data, the air-side pressure drop through the tube bundle was predicted by open literature calculations and used to check the accuracy of the CFD results. For this purpose Eq(2) was used, where Nr is number of tube rows (-), f_b is the friction factor of the bundle (-), ρ is the fluid density (kg/m^3) and V_{max} is the maximum air velocity through the tube bundle (m/s). From Serth and Lestian (2014)

$$\Delta P_A = \left(2 * f_b * Nr * (\rho * V_{max})^2\right) / \rho \tag{2}$$

V_{max} is calculated from Eq(3) where V_{face} is the face velocity (m/s), P_t is the tube pitch (m), D_r is outside diameter(m), n_f is the number of fins per meter (#/m), b is the fin height (m) and τ is the fin thickness (m).

$$V_{max} = P_t * V_{face} / P_t - D_r - (2n_f * b * \tau) \tag{3}$$

Eq(4) was used to calculate f_b the friction factor through the tube bundle developed by Ganguli, et al (1985) where a is equal too $((P_t - D_f)/D_r)$ (where D_f is the outer fin diameter (m)) and Re_{eff} is the effective Reynolds number though the bundle equal two bay's Reynolds number multiplied by fin spacing divided by fin height, Serth and Lestian (2014).

$$f_b = \left\{1 + \frac{2e^{-(a/4)}}{1+a}\right\}\left\{0.021 + \frac{27.2}{Re_{eff}} + \frac{0.29}{Re_{eff}^{0.2}}\right\} \tag{4}$$

For the air conditions investigated with Eq(2) a pressure drop of 86.4 Pa is calculated. When applying CFD simulation a pressure drop of 88.4 Pa is calculated, indicating that the CFD model is accurately predicting the air flow. Figure 4 and Figure 5 show the air v velocity (velocity component in the upward direction) on a plane 10 mm below the tube bundle for the three and four fan arrangement. Shading towards dark indicates low air velocities and light shading indicates high air velocities. There are two distinctive regions of low air velocity, the centres of each fan caused by the fan hub and the area between two adjacent fans.
Line 1 is used to compare the velocity distributions in the three fan and four fan geometries. This is done by calculating the percentage of velocity along the length of the bundle that is within 10 % of the mean velocity on the line. Therefore a higher percentage will mean a more even distribution. For the three fan simulation 5.5 % is within 10 % of the mean v velocity of 3.2 m/s. For the four fan simulation 5.7 % is within 10 % of the mean v velocity of 3.5 m/s. This shows that the four fan design gives a slightly improved air flow distribution. This air maldistribution is not taken into account when applying air cooler design correlations; they are based on an ideal distribution of airflow over the whole bundle. The ideal situation would be for the profiles shown in Figure 4 and Figure 5 to show constant grey shading.
The depth of the plenum of the ACHE was change to investigate what effect this had on the air velocity distribution. Three depths where simulated the original 720 mm, 500 mm and 1,000 mm. When compared to the original geometry in Figure 4 the 500 mm depth plenum shows worse distribution and the 1,000 mm depth plenum shows improved fluid distribution below the tube bundle.

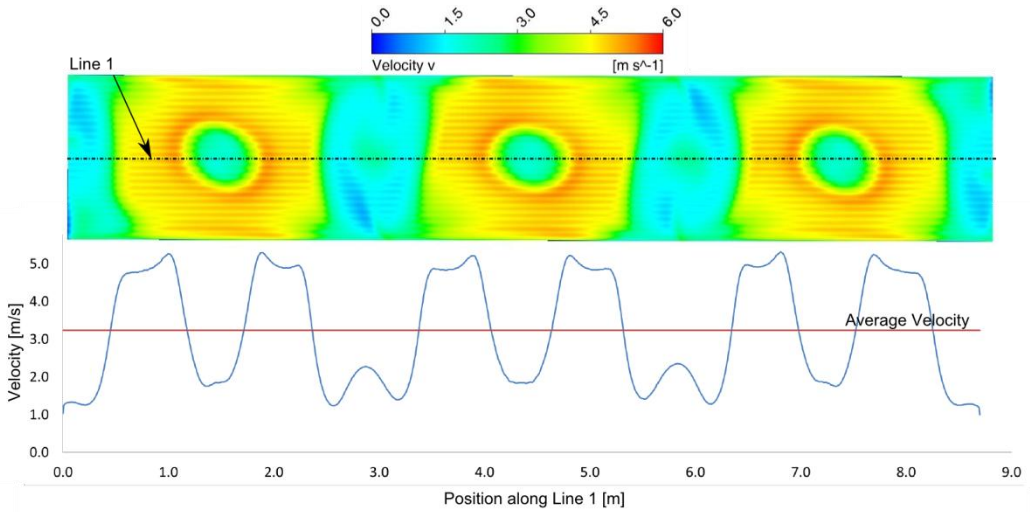

Figure 4: Three fan arrangement velocity profile and graph of V velocity along line1

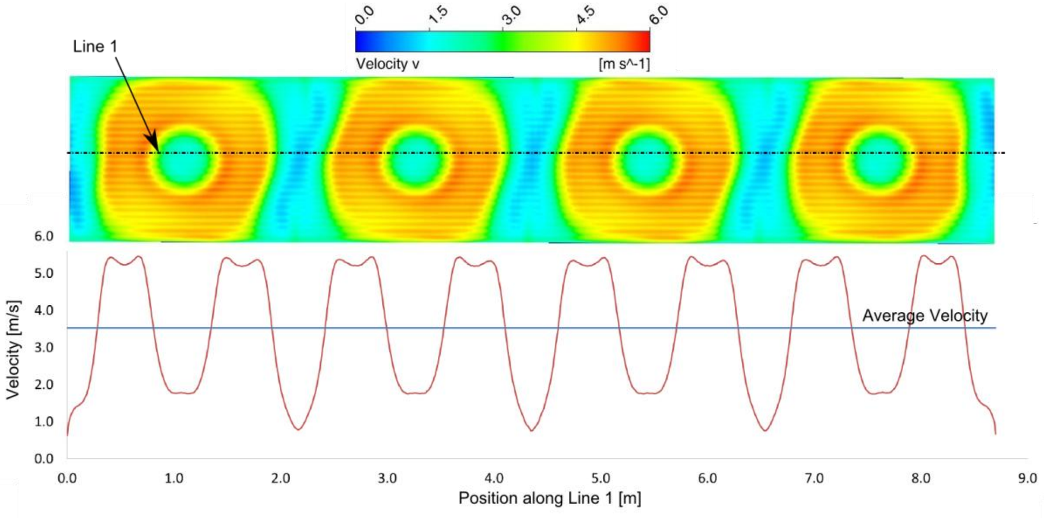

Figure 5: Four fan arrangement velocity profile and graph of V velocity along line 1

5. Conclusions

CFD has been successfully used to visualize the fluid flow on the tube-side and air-side of ACHE's. From the CFD results problem areas within the ACHE geometries have been identified along with the effects of bypass due to the poorly placed vent hole and non-welded partition plate ends, giving a 27 % reduction in duty for this worst case scenario.

It was found that the four fan arrangement and increasing the depth of the ACHE plenum had a positive effect on the air-side flow distribution as a plenum depth of 1,000 mm gave a more even velocity distribution than one of 500 mm depth.

Future CFD work will be carried out to investigate the effect of air-side maldistribution on overall heat transfer in ACHE's.

References

Habib M.A., Ben-Mansour R., Said S.A.M., Al-Qahtani M.S., Al-Bagawi J.J., 2009, Evaluation of Flow Maldistribution in Air-Cooled Heat Exchangers, Computers & Fluids, 38(3), 677-690.

Huang L., Lee M.S., Saleh K., Aute V., Redermacher R., 2014, A Computational Fluid Dynamic and effectiveness-NTU based co-simulation approach for flow mal-distribution analysis in microchannel heat exchanger headers, Applied Thermal Engineering, 65, 447 – 457.

Chu W., Yu P., Ma T., Wang Q.W., 2013, Numerical analysis of plain fin-and-oval-tube heat exchanger with different inlet angles, Chemical Engineering Transactions, 35, 481-486 DOI:10.3303/CET1335080

Bredell J.R., Kröger D.G., Thiart G.D., 2006, Numerical Investigation of Fan Performance in a Forced Draft Air-cooled Steam Condenser, Applied Thermal Engineering, 26(8-9), 846-852.

Serth R.W., Lestian T., 2014, Process Heat Transfer: Principles, Applications and Rules of Thumb (2nd Edition), Academic Press/Elsevier, Oxford, UK, Elsevier.

Sinnott R.K., 2009, Chemical Engineering Design (5th Edition), Oxford, UK: Butterworth-Heinemann.

Ganguli A., Tung S.S., Taborek J., 1985, Parametric Study of Air-cooled Heat Exchanger Finned Tube Geometry, American Institute of Chemical Engineers Symposium Series, 81-245, 122-128.