HEFAT2011
8th International Conference on Heat exchange, Fluid Mechanics and Thermodynamics
26 June – 1 July 2011
Pointe Aux Piments, Mauritius

NUMERICAL INVESTIGATION OF FLOW FIELDS OF AIR-COOLED CONDENSER CELL INCLUDING FAN BLADE PASSAGES

Hui Zhang*, Yujin Yue, Xiaoze Du, Lijun Yang and Yongping Yang
*Author for correspondence
Energy & Power Engineering School,
North China Electric Power University,
Beijing, 102206
China
E-mail: fluentfluid@gmail.com

ABSTRACT

The primary purpose of this numerical investigation is to analyze the characteristic of inside and outside flow fields of air-cooled condenser (ACC) cell in power plant. The fin-tube bundles are modeled using porous regions, and fan blade passages are modeled in details to gain accurately fan exhaust flows. Results of numerical investigation indicate that rotational fan exhaust flows lead to inside rotational symmetry flows and distributions of outflows from fin-tube bundles in A-frame ACC cell. When rotational flows are in same direction of ACC outflows in the area, high pressure increases the ACC mass outflows. In the under corner near the ACC inlet with inconsistent interfaces, less ACC outflows appear because of air flows paralleling to the ACC outflow side, influenced by the rotational flows.

INTRODUCTION

Air-cooled condensers (ACCs) have also been successfully employed in power plants. From the aspect of the whole air-cooled “island”, the influence of environmental wind on the performance of ACCs was investigated numerically [1]. Situations of cross winds and hot plume air recirculation were presented. From the aspect of ACC cell, the air-cooled heat exchanger and the forced draught fan were investigated firstly and individually. To air-cooled heat exchanger, plenum chamber flow loss [2] and inlet flow loss [3] were experimentally investigated. The effects of Heat exchanger characteristics, plenum chamber geometry, air flow maldistribution, inlet flow loss on heat exchanger thermal performance were analyzed. To forced draught fan, researches focused on effects of fan inlet flow distortions on the ACC cell performance. The distance between the fan platform and the ground level [4, 5], the walkway along the edge of fan platform [4], fan inlet shrouds [5], edge-proximity and wind-induced cross-flow [6, 7] were analyzed basing on effects of the ACC cell performance.

In researches mentioned above, axial fans were modelled as an actuator disc, where the actuator disc forces are calculated using blade element theory. In general, although good agreement is obtained between the numerical results and experimental data, velocity downstream of the fan blades, as well as the fan power consumption, is under-predicted by the fan model [8].

Recently, fan performances were numerically investigated including characteristics of fan blades. It is found that a change in the angle at which the fan blades are set in the fan hub as well as the volume flow rate at which the fan operates affects the plenum chamber aerodynamic behaviour. It is further shown that the maximum possible recovery of kinetic energy within the plenum chamber does not necessarily coincide with the point of maximum fan static efficiency [9]. Hub-tip-ratios of fans also were brought into investigations of effects of inlet flow distortions on the ACC cells. Numerical simulations show all-round superior performance in terms of volume flow rate for the fan with a hub-tip-ratio of 0.4 [10].

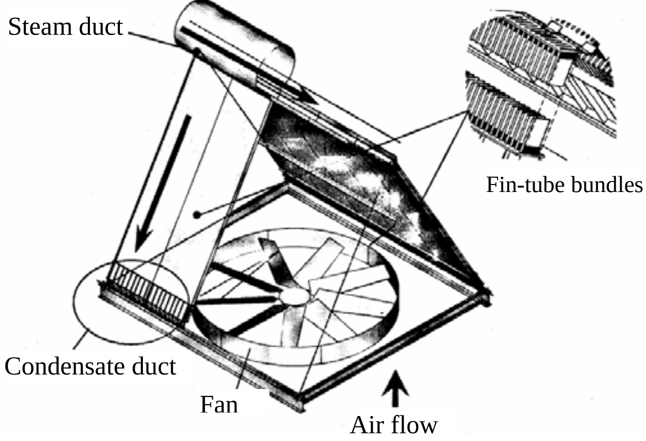


Figure 1 Air-cooled condenser model cell with a fan

This paper aims the air-cooled condenser (ACC) cell for 600MW power plant, showed in Figure 1. In ACC unit with side length of 10m, the cross-section size of the tube and fin is 219×19mm, 190×19mm respectively. The fin thickness is 0.25mm, and fin clearance is 2.3mm.

A 5×6 model cells rig was found to investigate flow characteristics in cells including fan performances. The size of a model cell is about 1/10 of the fact size of ACC cell for 600MW power plant. In this paper, a model ACC cell was numerically simulated in which fan blade passages were modelled. Velocity downstream of the fan blades is predicted more accurate in this way than in actuator disc way. Flow fields inside and outside model ACC cell were especially analyzed.

OBJECT FOR NUMERICAL SIMULATION

To gain accurately flow fields in inner ACC cell, fan blade passages is modelled in detail to give more accurate flow at exit of the fan. Based on this paper's research purpose, fin-tube bundles is simulated using the porous model rather than be modelled in detail. In addition, the steam duct is also modelled in outline size, and all heat exchange is ignored in whole simulation.

It is GAMBIT code that is used to generate the mesh for the inside and outside of A-frame ACC cell, and HEXPRESS code for fan blades and air inlet zone. The total number of node points is about 600,000. Two grid domains are illustrated in Figure 2.

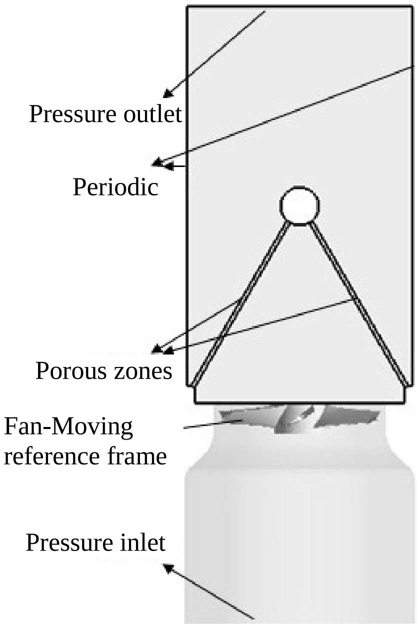


Figure 2 Numerical domains and boundary conditions

NUMERICAL METHOD

The numerical solution was finished using the FLUENT CFD code, in which detail setups are solver of pressure based, implicit formulation, steady, S-A turbulence model, SIMPLE model algorithm for pressure correction.

The flow field characteristics inside of ACC is the aim for research, and mass flow rate past the fan is decided by the fan performance and the resistance of the finned tube bundles. With a "moving reference frame" in Fluent, the main flow character can be gained as a steady-state problem. Authors also used the "slipping mesh" to do some unsteady calculation, and the unsteady couple-character is not obvious and subversive to main flow character.

Setups of boundary conditions are showed in Figure 2. The total pressure at inlet and static pressure at outlet are both set to zero. Two surfaces contacting the fan domain and the ACC domain are set into "interface" boundary condition. The flow mass is adjusted by changing the fan rotation speed and parameters of porous zones.

In FLUENT, the porous media model can be used for a wide variety of problems, including flows through packed beds, filter papers, perforated plates, flow distributors, and tube banks. In this paper, the porous media model is applied to fin-tube bundles and the pressure loss in the flow is determined via inputs based on actual air-cooled fin-tube bundles.

The boundary of ACC domain is set to periodic condition. Considering the row layout of multi A-frame ACC cells and rotational fan exhaust flow, it is an error to set the boundary to symmetry condition.

VELOCITY CHARACTERISTIC AT EXIT OF FAN

Table 1 Numerical results at exit of fan

Fan total pressure rise	126.72 Pa
Fan flow mass	6.14 m^3/s
Fan maximum circular velocity	40.0 m/s
Fan average axial velocity	10.8 m/s
Average circular velocity at exit of fan	7.6 m/s

Table 1 lists some numerical results at exit of fan. Average circular velocity at exit of fan is about 70% of the value of fan average axial velocity, so there will be an error if the circular velocity is ignored at inlet flow field of A-frame ACC cell. In FLUENT, "fan model" can be used, simplifying the fan into a plane, and an empirical fan curve can be input, which governs the relationship between pressure rise and flow rate across a fan element. And radial and tangential components of the fan swirl velocity can be specified. But the fan model does not provide an accurate description of the detailed flow through the fan blades showed in Figure3, because axial velocity and circular velocity at exit of fan, is changing along with radial position.

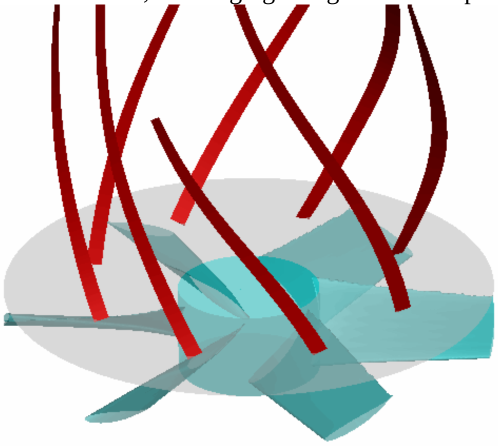
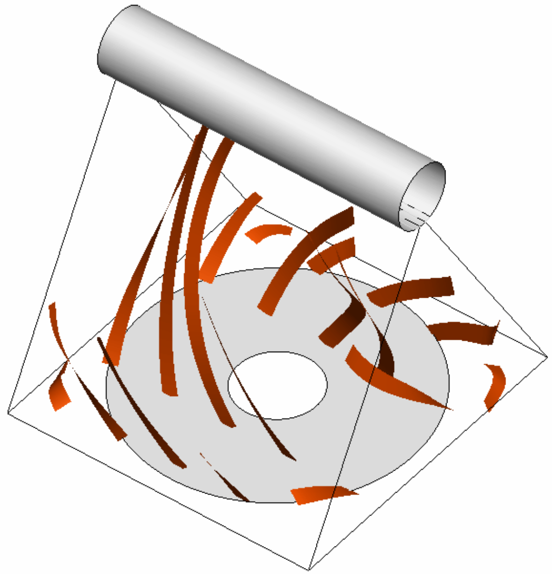


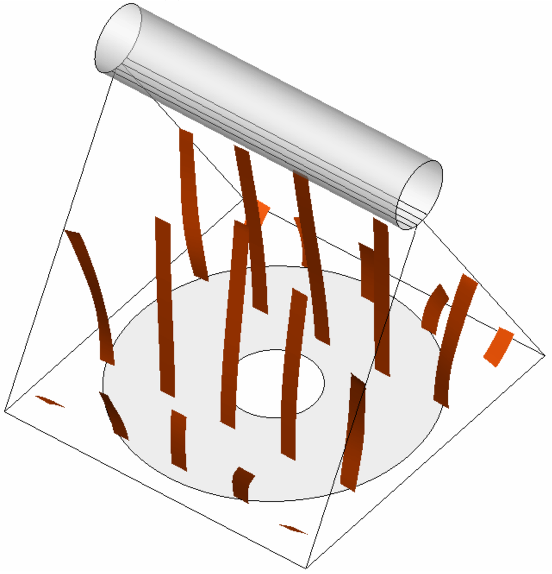
Figure 3 Streamlines at exit of fan

SIMULATION RESULTS

The fan blade passages are simulated, and fan rotational outflows are obtained at inlet of ACC. In same mass flows, flow fields with fan irrotational outflows are simulated by using simply fan model.



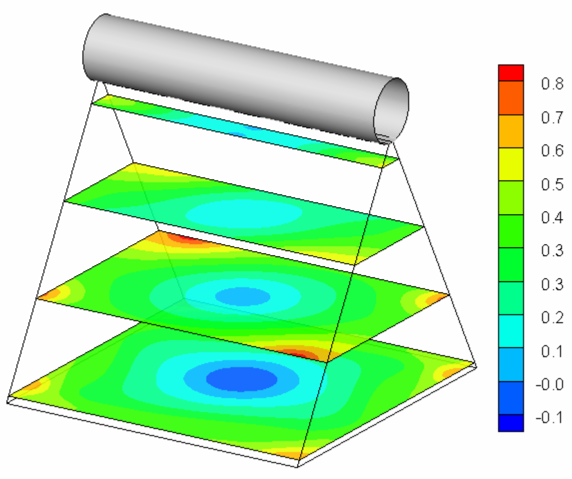
(a) Fan rotational outflows



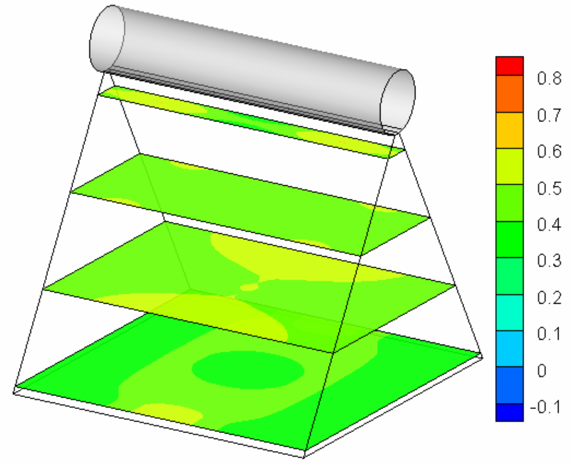
(b) Fan irrotational outflows

Figure 4 Streamlines inside ACC

Affected by fan rotational outflows, rotational flows are also the main flow character in A-frame ACC cell showed in Figure 4(a). In same fan mass flows and the same resistant loss, rotational outflows need fan supply high total pressure rise. And the rotational flows will results in very different pressure distributions with irrotational flows in ACC cell showed in Figure 4(b). Another important influence factor is the flow passage contracting of the A-frame construction. In addition, the fan outflow section is annulus, and the ACC inflow section is rectangle, so there are inconsistent interfaces. All these factors will affect the total outflow rate and outflow distributions from ACC.



(a) Fan rotational outflows



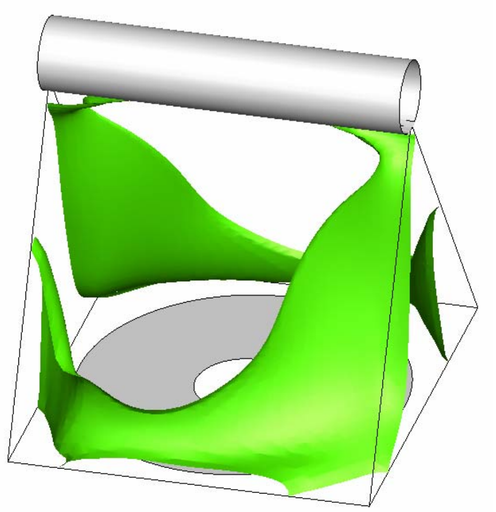
(b) Fan irrotational outflows

Figure 5 Relative pressure contours inside ACC

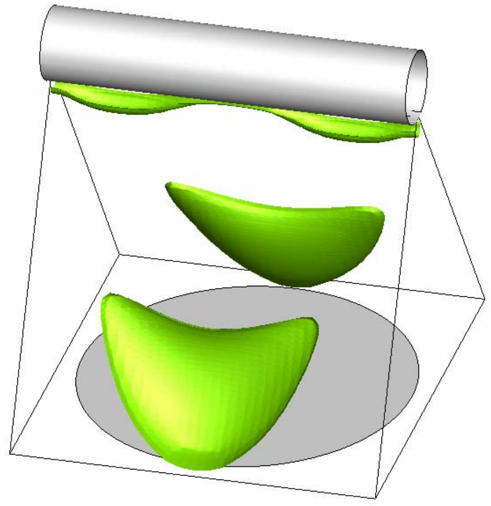
In swirling flows, pressure increases from the axis to the periphery. Figure 5(a) shows this style of pressure distributions of rotational flows inside ACC cell. Relative pressure is equal to the ratio of the real pressure and the fan total pressure rise. It is not disadvantaged in terms of ACC outflow, because high pressure zones close to ACC sides are advantage to air exhausting. Figure 5(a) also shows the rotational flows holding from inlet of ACC to approaching steam duct, even though it weakening.

Because the air only exhausts from one set of opposite sides of ACC, rotational flows result in higher pressure in a set of opposite angle area, and lower pressure in the other set of opposite angle area. In the higher pressure area, it is visible that the flow velocity direction is same with ACC side outflow direction, being showed in Figure 5(a). This indicates that more air flows into the area, but they does not discharge enough from the adjacent ACC side, and partial air accumulates in the area making higher pressure.

When fan outflows is irrotational, showed in Figure 5(b), pressure distributions inside of ACC are bilateral symmetry, rather than rotational symmetry. And pressure diversities and velocity magnitudes are less than that while fan outflows is rotational in same mass flows.



(a) Fan rotational outflows



(b) Fan irrotational outflows

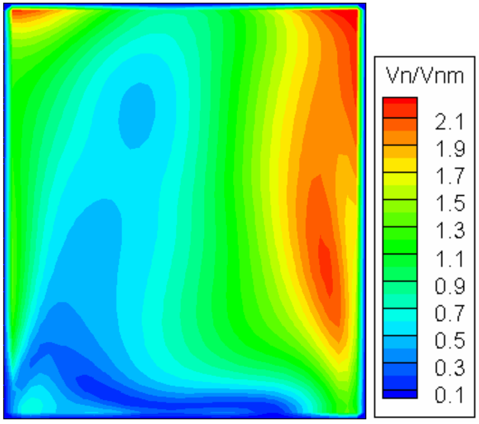
Figure 6 Pressure iso-contours inside ACC

In keeping with 2D pressure distributions showed in Figure 5, 3D pressure distributions are showed in Figure 6, the extracted parts indicate the zones in which the pressure is above a half of the fan total pressure rise. ACC outflow distributions are showed in Figure 7, on what Vn is exit normal velocity of fin-tube bundles sides and Vnm is the mass-weighted average of Vn.

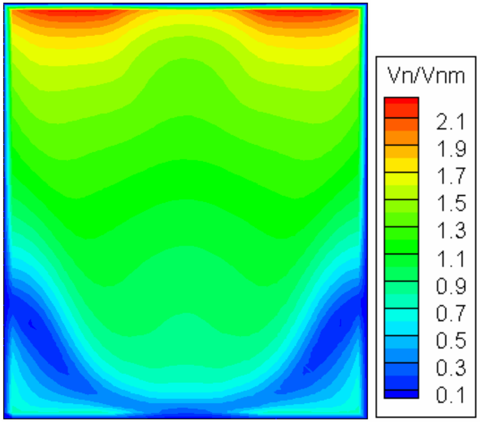
In Figure 6(a) and Figure 7(a), pressure distributions match that of the ACC outflow except for the under area close to the inlet of ACC, with High pressure leading to much outflows. It is similar while fan outflows are irrotational, showed in Figure 6(b) and Figure 7(b). Rotational flows dominates flow fields in A-frame ACC cell, and results in rotational symmetry non-uniform outflows.

Whether fan outflows are rotational or irrotational, high pressure appears both sides, and low pressure appears near the centre of the area near steam duct. In the triangular prism of ACC cell, the flow passage is changing from quadrate into narrow rectangle. On the other hand, the fan outflow section is annulus, and the velocity is higher in the centre area than in around area. Two factors result in high velocity in the centre area near steam duct, and the air branches on both sides, and make high pressure. In Figure 7(b), sharp high ACC outflows appear near steam duct, and this indicates the current flow rate is perhaps a little high if considering the efficiency of flow delivering.

In Figure 6(b), the high pressure area in the middle-lower of the ACC cell, is also related with the flow passage contracting of the triangular prism. Flow passage contracting causes that the velocity in sides decreases, with the velocity in centre holding higher value. It is indicates that ACC outflows in the area does not change the trend of flow velocity decreasing and pressure increasing, though ACC outflows advantage weakening increasing of pressure.



(a) Fan rotational outflows

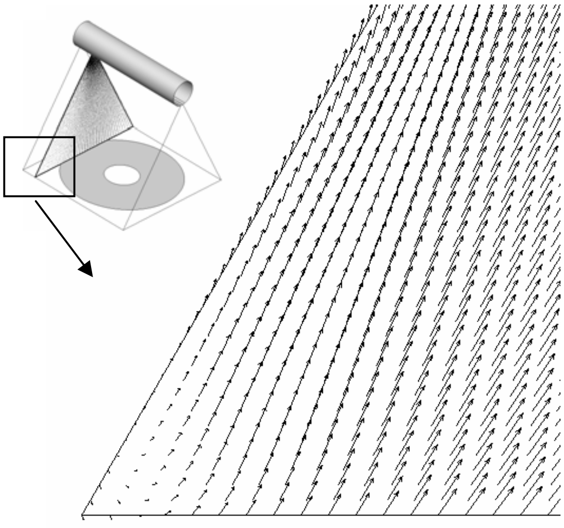


(b) Fan irrotational outflows

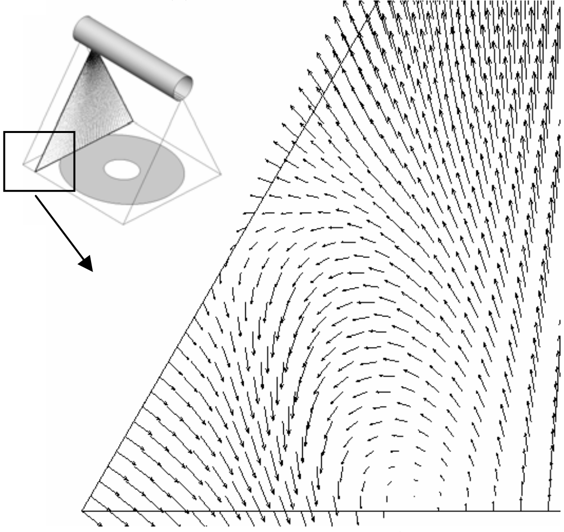
Figure7 ACC outflow distributions

The remarkable low outflow area appears in under corners and under edge area near inlet of ACC cell, showed in Figure 7. Inconsistent interfaces are the main reason for that, because of differences between the annulus fan outflow and the rectangle ACC inlet. Fan outflows do not blow directly on the under corners and under edge area near inlet of ACC inlet, and fan outflows of high velocity in the boundary will disturb the air flow of this area. When fan outflows is irrotational, the pressure in under corners and under edge area is not visibly higher than the other area, but the pressure is high when fan outflow is rotational, in Figure 6(a), because of pressure distribution of swirling flow. In fact, when rotational flows are in same direction of ACC outflows in the area, high pressure increases

ACC mass outflows, in Figure 7(a), and contrarily, when the rotational flows is in reverse direction of ACC outflows, high pressure has no remarkable advantage to ACC outflows.



(a) Fan rotational outflows



(b) Fan irrotational outflows

Figure 8 Velocity vector feature in corners

Figure 7 shows that the lowest outflow velocity is approaching zero, and in fact in Figure 8, the velocity vector feature shows back flows in the area with lowest outflow velocity which is the under corner near ACC inlet. But with fan rotational outflows, the back flow area is smaller than that with fan irrotational outflows, and the main reason for the less ACC outflow is that the air flow parallels to ACC outflow side showed in Figure 8(a). The reverse flow direction between inlet rotational flows and ACC outflows results in paralleled flows near the area. With fan irrotational outflows, the main reason for low ACC outflows is just back flows, showed in Figure 8(b).

For inside rotational symmetry flow fields of A-frame ACC cell, the mass outflow from one side of ACC is more than the other side. In outside flow fields of ACC, flows will lean on more mass outflow side. Generally, fans have same rotational direction, so the direction of flow leaning is same for multi ACC cells running in rows, showed in Figure 9. Along the stream duct, the flow leaning direction reverses because of the rotational flows. And opposite flows will induce eddy in the centre area of two flow zones, showed in Figure 10.

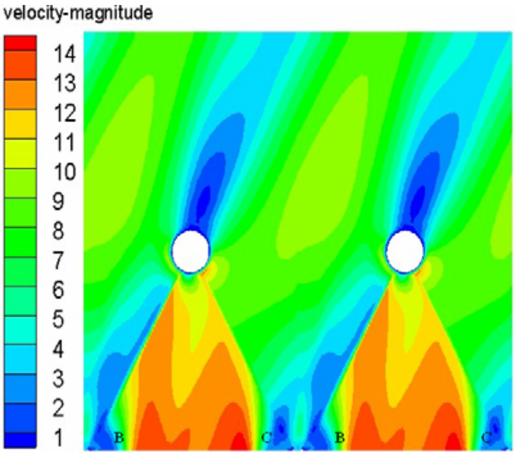


Figure 9 Velocity contour of two cells

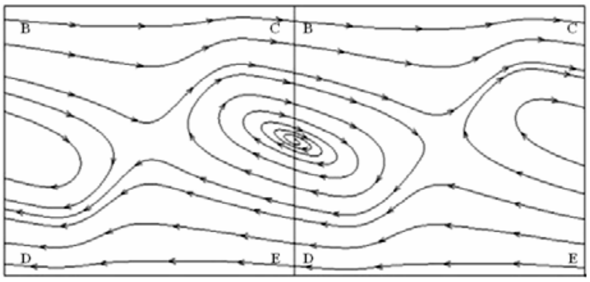


Figure 10 Streamlines on cross-section up team ducts

CONCLUSION

It is accepted that the influence of the fan on ACC cell heat exchange performance is not only relatively strong but also very difficult to predict. This paper analyses inside and outside flow fields of A-frame ACC cell under the influence of fan rotational exhaust flows only from air flow view rather than heat exchange view. Correlation between flow characteristic and heat exchange is main content of follow-up research.

Results of numerical investigation indicate that rotational fan exhaust flows lead to inside rotational symmetry flows and non-uniform distributions of outflow from fin-tube bundles in A-frame ACC cell. When rotational flows is in same direction of ACC outflows in the area, high pressure increases ACC mass outflows, and less mass outflows when rotational flows is reverse direction of ACC outflows.

Inconsistent interfaces appear when air flows from annulus fan outflow section to rectangle ACC inlet section. And then, in the under corner near ACC inlet, less ACC outflows are caused for air flows paralleling to ACC outflow side, influenced by rotational flows. If fan outflows are irrotational, back flows in the area is the main reason for less ACC outflows.

In outside flow fields of ACC, flows will lean on more mass outflow side, and an eddy appears in the middle area of two adjacent ACC cells up steam ducts.

ACKNOWLEDGEMENTS

This work was funded by National Basic Research Program of China (973 Program): 2009CB219804.

REFERENCES

[1] Duvenhage K., Kroger D.G., The influence of wind on the performance of forced draught air-cooled heat exchangers, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol 62,1996,pp. 259-277.

[2] Meyer, C.J., and Kroger, D.J., Plenum chamber flow losses in forced draught air-cooled heat exchangers, *Applied Thermal Engineering,* Vol. 18, 1998, pp. 875-893.

[3] Meyer, C.J., and Kroger, D.J., Air-cooled heat exchanger inlet flow losses, *Applied Thermal Engineering,* Vol. 21, 2000, pp. 771-786.

[4] Salta, C.A., Kroger, D.J., Effect of inlet flow distortions on fan performance in forced draught air-cooled heat exchangers, *Heat Recovery System & CHP*, vol. 15, No. 6, 1995,pp. 555-561.

[5] Duvenhage K., Vermeulen J.A., Meyer, C.J., and Kroger, D.J., Flow distortions at the fan inlet of forced-draught air-cooled heat exchangers, *Applied Thermal Engineering,* Vol. 16, 1996, pp. 741-752.

[6] Stinnes W.H. and Backstrom von T.W., Effect of cross-flow on the performance of air-cooled heat exchanger fans., *Applied Thermal Engineering*, Vol. 22, 2002, pp. 1403-1415.

[7] Meyer, C.J., Numerical investigation of the effect of inlet flow distortions on forced draught air-cooled heat exchanger performance, *Applied Thermal Engineering*, Vol. 25, 2005, pp. 1634-1649.

[8] Meyer, C.J., and Kroger, D.J., Numerical simulation of the flow field in the vicinity of an axial flow fan, *International Journal for Numerical Methods in Fluids*, Vol. 36, 2001, pp. 947-969.

[9] Meyer, C.J., and Kroger, D.J., Numerical investigation of the effect of fan performance on forced draught air-cooled heat exchanger plenum chamber aerodynamic behavior, *Applied Thermal Engineering*, Vol. 24, 2004, pp. 359-371.

[10] Bredell J.R., Kroger, D.J., and Thiart G.D., Numerical investigation of fan performance in a forced draft air-cooled steam condenser, *Applied Thermal Engineering,* Vol. 26, 2006, pp. 846-852.