Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

Flow Analysis of Cooling Water in the Gas Turbine Fin Fan Cooler at Utilities Plant

A. M. S. Zuan[*,1,a], *S. S. Hidayah*[1,b], *S. Syahrullail*[1,c], *M. N. Musa*[1,d], *and E. A. Rahim*[2,e]

Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia.

Fakulti Kejuruteraan Mekanikal dan Pembuatan, Jabatan Kejuruteraan Pembuatan dan Industri (JKPI), Universiti Tun Hussein Onn Malaysia, 83000, Parit Raja, Johor, Malaysia.

[a,*]*zuanazhary@yahoo.com*, [b]*hsofia2@live.utm.my*, [c]*syahruls@mail.fkm.utm.my*, [d]*mdnor@fkm.utm.my*, [e]*erween@uthm.edu.my*

Abstract – *The main purpose of this project is to study the flow analysis of cooling water in gas turbine fin fan cooler in utilities plant in Gebeng, Pahang. The focus of this project is to study the flow analysis over the implementation of the new design of the fin fan cooler, with enhancement from the aspect of geometrical dimension and fin material selection. There are two design that have been analysed which are the former old design of the fin fan cooler who later will be replaced and the new and improved design of fin fan which will be commissioned. The flow patterns are studied by using the section of the finned tube within the domain of both different design of new and old. The Ansys Fluent R15.0 is utilized to stimulate the related boundary condition such forced convection and conduction for the fin section within the domain. The model of finned tube are drawn and represented geometrically by using Solidworks 2013. Finally, the result accumulated from the flow simulation of each design have been analysed and explained in precise manner to provide the future researchers to make further and more concise study regarding the flow analysis of gas turbine fin fan cooler in utilities plant.*

Keywords: Flow Analysis, Fin Fan Cooler, Cooling Water, Ansys Fluent, Gas Turbine

1.0 INTRODUCTION

Utilities plant is the heart of industrial operation which mostly involved the utilization of heavy machinery and tremendous production capacity which rely on private utilities consumption. As for Utilities Gebeng, Kuantan under Petronas Gas Berhad, they were obligated to supply adequate amount of energy especially electricity to surrounding customer such as Petronas Dehydrogenation (PDH), BASF Petronas Chemical (BPC), Kaneka, and also Polyplastic. Fulfilling the daily demand has been a challenge since the operation of the plant need to be monitored 24-hours and even slight glitch of running requirement impacting customers' operation which leads to loss of production and costs.

Utilities Gebeng consists of four major plants producing essential commodities such as electricity, steam, nitrogen liquid and gas also demineralized water. The most important plant is cogeneration (COGEN) plant mainly focused on the steam and electricity production. This COGEN plant formed by few significantly conventional components mainly focus on gas turbine, heat recovery steam generator (HRSG), boilers and diesel engine conclude by combination of power generation system and steam generation system.

Gas turbine stressed out as main mechanism of electricity generation driven by set of internal combustion engine powered by diesel connected to upstream rotating compressor coupled to a downstream turbine with combustion chamber in between. To highlight the gas turbine practical functions, this component equipped with standalone cooling system just to maintain the temperature of lubricating system. Inconsistency in lubrication oil temperature is a serious issue due to the correlation in between the rate of cooling and bearing trip temperature. High bearing temperature eventually wage in the journal and thrust bearing dysfunctions due to particular factors and ultimately damaging the bearings by radial or axial rubbing of the rotor.

The cooling water undergo minimization of heat capacity within the fluid by the flow of the liquid into a set of air cooled heat exchanger or gas turbine fin fan (GT FiFa) cooler built with fins to optimize the rate of heat transfer to surrounding. Due to the deteriorating condition of current GT FiFa, the management of Utilities Gebeng has decided to install new GT FiFa with upgrades from the perspective of fins material and tube bundles design standard. The typical design uses a fin-fan cooler to cool combustion turbine extraction air, with no attempt to regain any heat lost to the atmosphere [1]. New materials that will be used for tubes are carbon steel and aluminium for the fins. The design is compatible with current structure including fans, motor and piping which expected to deliver as per current design flow, pressure and temperature. Many research and methods has been done to study the fin fan cooler usage [2] [3-6]. Changes of the design has been evaluated, reviewed, and verified prior to their implementation.

Behaviour of the new GT FiFa is under the impression that this new equipment will behave better than the current one. The related parameters will be identified and the utilization of Computation Fluid Dynamics (CFD) specifically ANSYS ICEM allow the simulation of the fluid flow within the finned tube bundles inside the GT FiFa. The utilization of this software purposely defines the air flow pattern. Analysis of the air flow pattern enables us to study the air distribution within tube bundles and determine the effectiveness of new GT FiFa design upon the standard operational requirement. The enhancement of the new GT FiFa can be evaluated by comparing the collected performance data to the previous GT FiFa performance.

2.0 METHODOLOGY

The flow analysis of the finned tube bundles in the gas turbine fin fan cooler can be done analytically by using Computational Fluid Dynamics (CFD) software developed by ANSYS, well known as Ansys Fluent. This software is notoriously recognized as a tool to multiple models such as fluid flow and heat transfer, turbulence, volumetric reactions, multiphase flow and few others. This chapter significantly highlights procedures and the flow of work from development of three dimensional drawing up until the boundary condition identification for the flow simulation which focus on the correlation between the variables to achieve our main objectives of the study.

The procedures of utilizing Ansys Fluent programming obtained from user manual of the software. Referring to user manual can act as a guide for user to use the software especially when any trouble emerges from the utilization of the meshing and simulation of the model. This also can spare the right path for the user in performing the simulation.

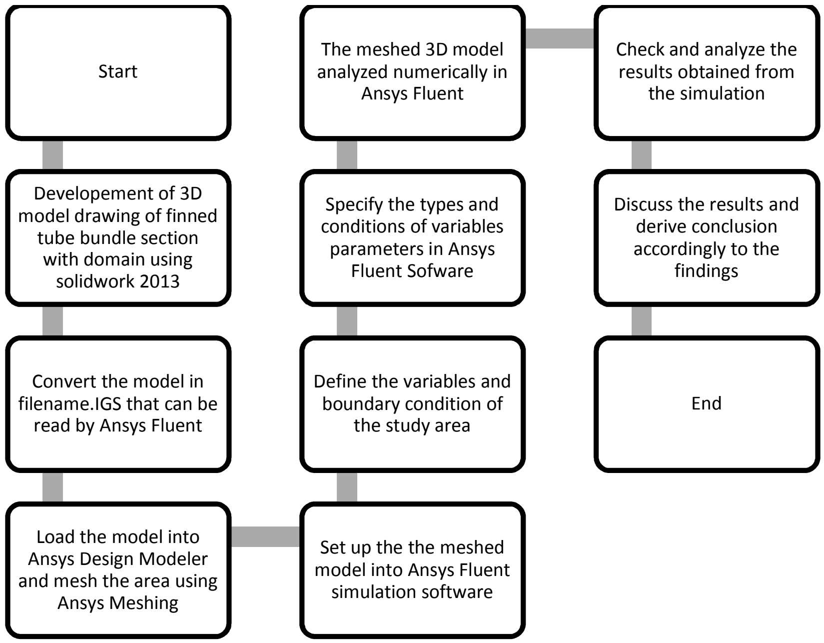


Figure 1: Flow chart of finned tube section simulation procedures

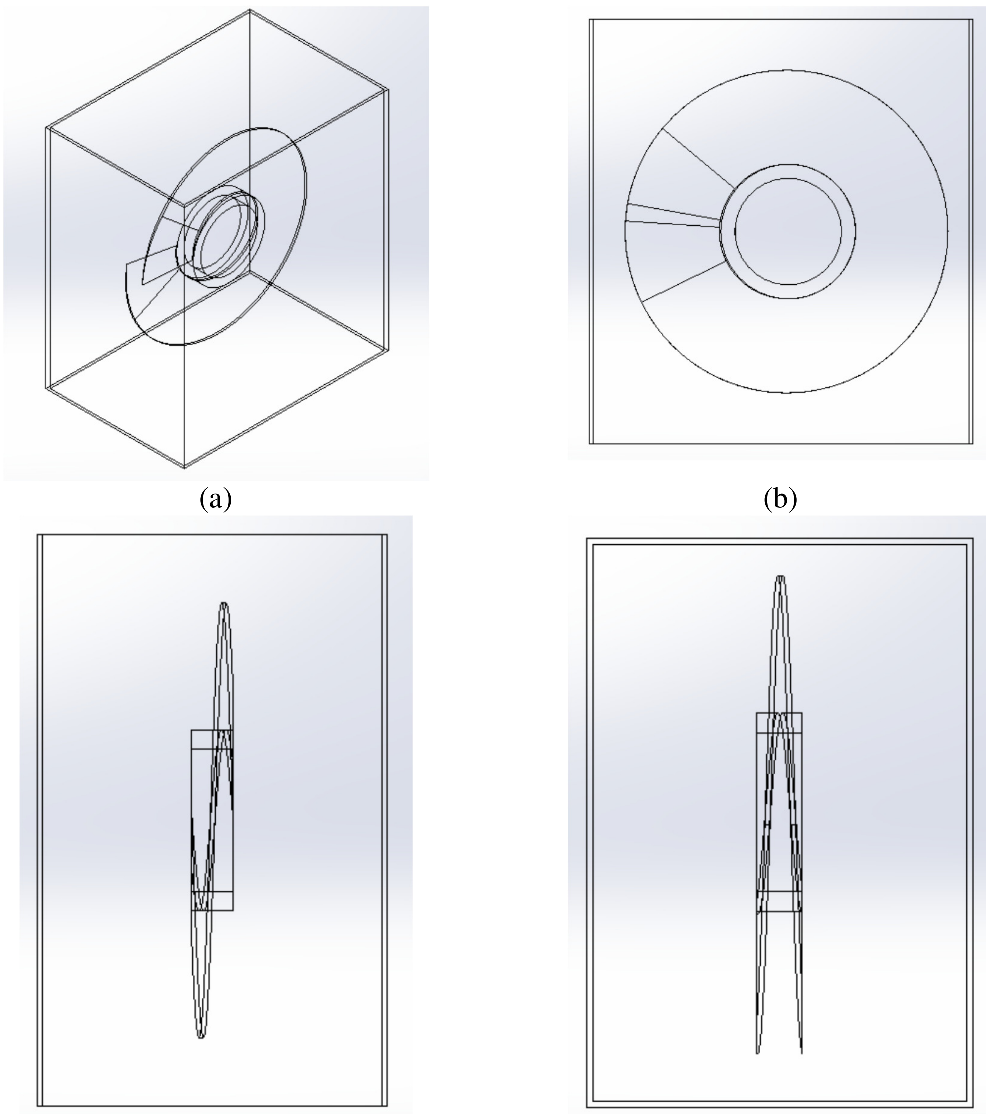
Simulation of flow in Ansys Fluent requires the identification of boundary for the theoretical fluid to swift represented by the three dimensional model drawing of the finned tube geometry with their surrounding domain. As starter, the models of the finned tube were drawn accordingly to their scale for both new and old finned fan coolers. The main tool to illustrate the model is by using Solidwork 2013 as shown in the figure 2.

Since our objective highlighting fin condition as main focus, the fin fan cooler u section will be sectioned more to one piece of fin with the tube of significant length.

Figure 2: Front and side view section of the finned tube

Penerbit Akademia Baru
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

Ansys Fluent possessed a feature that enables the user to mirror symmetry analysis in order to simplify the analysis. So, the model dimension that will be set up into Ansys Fluent is sectional dimension of finned tube with a rectangular domain. Figure 3 show the wireframe view of the fin fan tube.



(a) (b) (c) (d)

Figure 3: (a) isometric (b) side (c) front (d) top wireframe view of fin tube section with rectangular domain

To maximize the analysis for fins focus on material, the fin length and thickness are considered the same for both new and old design. The comparison between both old and new design for fin fan are shown in Table 1.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

Table 1: Data Comparison between old design and new design

		Old Design		New Design	
Tube	Length (mm)	120.97		124.94	
	Diameter (mm)	12.7	13.35	14.224	15.875
	Thickness (mm)	0.325		0.8255	
Fin	Length (mm)	11.112		11.125	
	Thickness (mm)	0.4		0.4	
Domain	Length (mm)	25		25	
	Wide (mm)	5		5	
	Height (mm)	50		50	
Cooling Water	Inlet Temperature (℃)	49.3		49.3	
Fin/Tube	Material	Pure Copper/Pure Copper		Aluminium/ Pure Copper	
Driven Fan	Air Velocity (ms^{-1})	8,182		10.87	

The Ansys Fluent 15.0 was being utilized in this project to stimulate the airflow pattern aftermath in the gas turbine fin fan cooler tube finned tube section after the meshing and boundary identification by using ansys meshing application. First, it is compulsory to set the simulation into 3D version because the model has been drawn in three dimensional models. There are several requirements that need to be dispensed before the software can simulate our case study.

The mesh file will be read, check and scale in the Ansys Fluent and by calculating the Reynolds number of the air flow, we can determine the type of flow to be decided in the vicious model. Since the value of the Reynolds number of the airflow exceeding 2,300, the type of flow for this airflow is turbulent. The velocity of the inlet will be defined as well in the boundary condition.

3.0 RESULTS AND DISCUSSION

Based on the outcome from the simulation by using Ansys Fluent 15.0, it basically appears as both different design exhibits diverse result which distinguished the results. This enables the comparison between the effectiveness of the modification to be made from the former design to the new design of fin fan tube.

The results from simulation illustrated that the vector of the air flow from the fan behaves differently according to their speed and the fin plate region it flow through. This discrepancy in results leads by the enhancement in the context of the materials of the fins, geometrical dimensions, pressure and temperature gradient, also the fan velocity variation for each design. But, putting the study within a perspective, our main purpose of this study inclination is to analyze the air flow pattern in the domain over the fin plate only.

The factors that govern the patterns of the air flow denote by the velocity of the inlet blower and the geometrical representation by the finned plate, upon the air flow travelling distance within the domain. In this analysis, there are three types of differences illustrated during the simulation which are the differences in velocity inlet of the blower, geometry dimensions and fin materials. Figure 4 below describe the similar pattern that happened over the finned plate within the enclosure.

Penerbit Akademia Baru
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

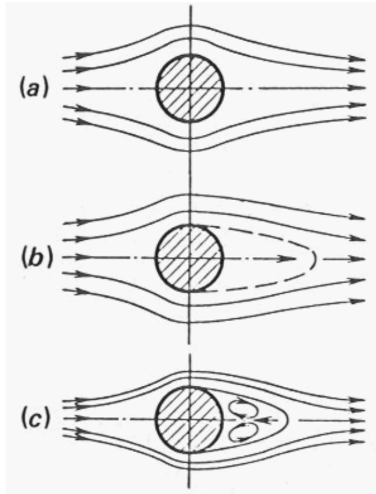


Figure 4: Examples of air flow pattern through several object

As we can see from the figure 4, the flow over flat plate creates a developed velocity boundary layer as a result of the fluid layer adjacent to the plate surface. Velocity boundary layer delineate as the area in which the variation of fluid velocity started to act out from zero to 0.99V. Since the air flow in this case originated from a blower, forced convection need to be considered which enacted as the mechanism of heat transfer channeling through a solid or quiescent fluid. Forced convection physically describes how the motion of the fluid was initiated. Forced convection is circling around the two dimensionless Prandtl and Nusselt number, which significant to the amount of heat transfer within the analyzed system. Fig. 5 illustrates the examples of streamlines and layers in forced convection mechanism.

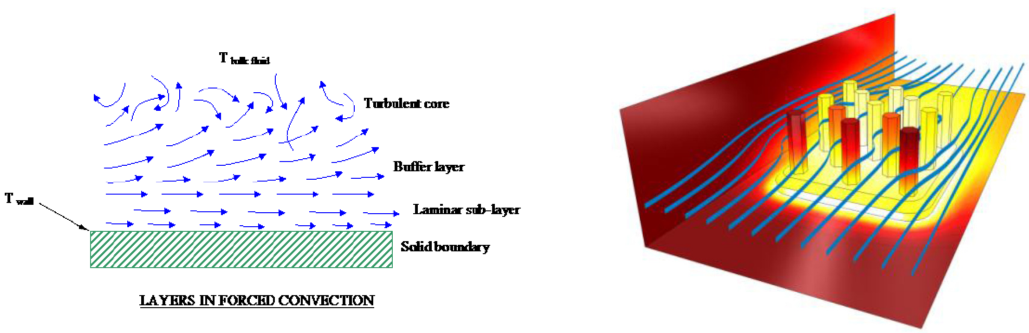


Figure 5: Layer and streamlines in forced convection

3.1 Effects from Different Velocity Inlet

The modification upon the tube finned fan leads to variation in inlet velocity for the new design, which the increment is substantial for the heat flow requirement within the system which dictating the effective heat flow out from the cooling water from the gas turbine oil lubrication system.

Velocity inlet is the initial representation of velocity vector, along with all relevant scalar properties of the flow, at flow inlets. As for the former design of the finned tube, the velocity constantly blows from the inlet at 8.182 m/s which it effectively transferring the heat out the

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

system for certain reliable period of time. As the lifespan of the finned tube grew longer, the air flow inside the finned tube somehow influenced not only the external circumstances but also internal limitations which constrained the heat exchange in between cooling water and air flow. Under these conditions, by revamping the current geometry of finned tube with designated velocity inlet, the heat exchange in between two fluids medium are expected to rise and outgrow the former finned tube. Figures below illustrating the result of velocity vector from the simulation based on velocity magnitude for both old and new design of finned tube along with the comparison in between both outlines.

3.1.1 Old Design of Finned Tube

Figure 6 (a) is the illustration of the result by the simulation from the standpoint of velocity vector based on velocity magnitude recommendation by the manufacturer. The flow of air depicted by the yellow region moving down vertically through the slice of finned tube plate section.

From the figure 6 (a), we can see that the flow of air projected steadily from the inlet but once the flow entered the region of sudden alteration in geometry which basically portrayed the finned tube section, the flow started to shift from yellow to green region. The alteration of graphic representation exhibits that the flow steadiness started to drop to lower value, which generates the direction disorientation of the velocity vector of the air flow. This inconsistency cause a whirlpool to form once it passes through finned tube geometry consequently from the velocity initiated by the fan blower. This whirlpool can be classified as rotational or forced vortex which can be described as the swirling motion of the fluid whereas the motion of fluid is rotated about its axis with specific angular velocity.

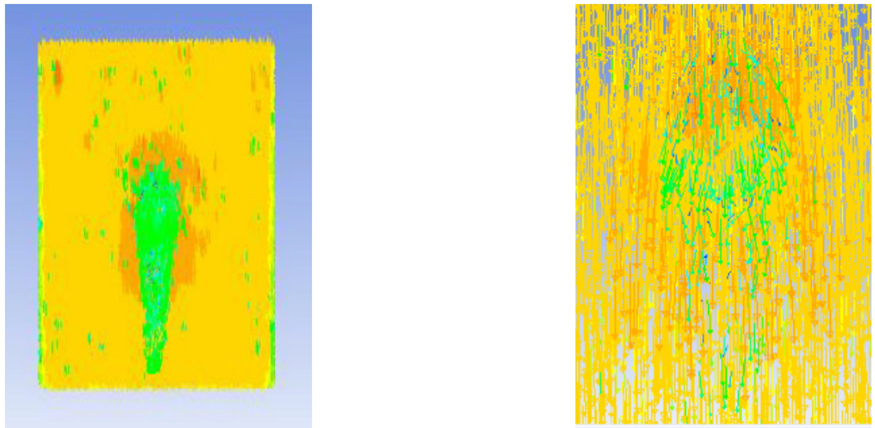
Based on asymmetry geometry of the finned tube which the fins swept around the tube, it creates a gap in between the swirling fins as shown in figure 6 (b) upon the slicing operation. This gap upon the flow of air over it experiencing the pressure drops which enable the pressure from atmosphere to push the flow of air into the gap. This causes the flow to change its orientation and whirl around the geometry which leads to decrement of velocity and less heat exchange. This causes the cooling effect to be less effective and affecting the cooling process of the lubrication oil of the gas turbine bearings.

Overalls, the velocity vectors of the flow outside the finned tube region are smooth since it is a free air flow without any obstacles straining the movement. The wide area of enclosure surround the sliced fins encourage the smoothness of the flow outside the fins region as it pushes the hot element from the finned tube out of the system.

3.1.2 New Design of Finned Tube

The new design of finned tube undergoes major modifications from the geometry and materials which drifted the velocity requirement higher that the old design in order to meet the requirement of the substantial cooling effect. The velocity for this new design is 10.87 m/s which establish the temperament on the velocity vector compared to the old design of fin fan tube. This temperament however still incline to the existence of vortex at the fin gap since the size of it can be considered as significant and affecting the flow, as well as the velocity vector.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015



(a) (b)

Figure 6: Velocity vector of old finned tube section

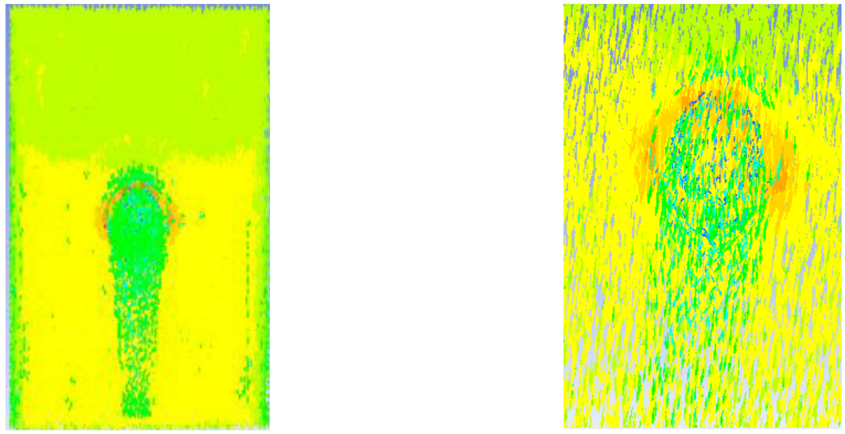
Figure 7(a) illustrated the velocity vector of the air flow from the inlet sinking to the outlet of the new fin fan design. We can see that the flow at the inlet is more consistent since it is pretty straightforward from to outlet compare to the condition the flow leaving the system through the outlet. The flow from inlet move straight and steady to the finned tube section as no any obstacle within the path and the flow stability about to be blown when the flow entering the region of temperature and geometry discrepancy. This flow non-uniformity act out as the mechanism of heat transfer confined in the system as it represents the heat exchange upon thermal equilibrium when the flow of air and plate conduction responds to their contact state simultaneously.

As can be reviewed in figure 7(a), the yellow region of the enclosure representing the air flow from the inlet blower and the intensity of the region is less than figure 6(a) which show that the discrepancy of the inlet velocity lead to different representation in the simulation. The same scenario of flow disorientation of happened at the fin gap and this created pressure drop which leads to velocity drop which can be assessed in figure 7(b). But this pressure drop enables the outside pressure to settle into the gap and pushes the heat out of the system. This ensures the cooling effect to be at vital level for the system to operate ideally. The rotational vortex existed at the region of low pressure gap which separated it from the main stream, neutralize it to normal flow by the steady irrational flow and formed a wake downstream away from the body. This wake includes a region of slow random moving fluid which later exit the system out to surrounding area.

3.2 Velocity Vectors Result Comparison

The velocity vectors from both simulation of new and old design appear to be no much different and it is difficult to discuss the discrepancy, we will compare the result by using the dimensionless parameters which contributes significantly to convective heat transfer within the system. Nusselt number viewed as the dimensionless convection heat transfer coefficient as in this study, forced convection governs the flow of air from the external source fan blower.

Penerbit Akademia Baru
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015



(a) (b)

Figure 7: Velocity vector of new finned tube section

Heat transfer through fluid layer is by convection when the fluid involves some motion, either naturally or by additional forces. Nusselt number represents the enhancement of heat transfer through the air layer as a result of convection relative to conduction across the same fluid layer. The larger the Nusselt number, the more effective the convection will be. The differentiation between the Nusselt number of both new and old design flow can be seen in table 2.

Table 2: Nusselt and Prandtl number from old and new fin fan simulation

	Old Fin Fan	**New Fin Fan**
Nusselt Number	6,254.95	10,312.95
Prandtl Number	0.7889-0.8457	0.7532-0.8487

From the Nusselt number comparison in between old and new design, new fin fan exhibit larger value which resort forced convection is more vigorous compare to the old design. This value leads us to believe that the heat transfer of the new design of fin fan is greater compare to the former design performance. The convection heat transfer anywhere along the surface is related to the temperature gradient in between the plate conduction originated from hot cooling water from the gas turbine lubrication system and flow of air by the fan blower.

On the other term, Prandtl number significantly related to the air flow and velocity vector since this dimensionless parameter best described the relative thickness of the velocity and thermal boundary layer. The Prandtl number for gases range from 0.19 – 1.0, which both of new and old design range fall within the same value (0.7532-0.8487) , since the flow of air establish from natural standard surrounding conditions and it still lies within the range which make total sense of the established results.

3.3 Effects from Different Type of Fin Materials

Effects from fin materials upon the heat transfer are studied via the conduction of the medium under steady conditions and surface temperatures. Most materials possessed thermal resistance properties which vary accordingly to their design consideration. By increasing the heat transfer using different type of fin material which more conductive from the previous one, cooling effects of the new material design of the system can be enhanced exceedingly the old finned tube material.

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

3.3.1 Old Design of Finned Tube

The old design of the copper tube increases their surface area by attaching the tube with extended fins surface from the same material as the tube. These fins surfaces are manufactured either by extruding, welding or wrapping thin metal sheet onto the curve surfaces of the copper tube. These fins enhance the heat transfer from a surface by exposing a larger surface area to mechanism of heat transfer, as for this case forced convection driven by the fan blower.

The copper tube conditioned to undergo conduction as a result from flowing hot cooling water leaving the heat exchange system in between the cooling water and high temperature of lubrication oil. The cooling water acts as a working fluid where it transfers the heat out from bearing of the gas turbine to ensure the friction and heat of the system does not influence the performance of the gas turbine and eventually lead to worst case scenario such gas turbine shutdown or bearing replacement.

The heat transfer via conduction for this finned tube can be scrutinized from the static temperature contours generated from the simulation software. The temperature distributions of the old finned tube section portrayed by figure 8(a), where the distribution depicted by the color variation of the contours. It can be seen in the figure that the exploitation of copper as fins material for this former design is really effective from the perspective heat transfer since copper possessed 352 – 401 W/m.K which can be considered high thermal conductivity and very practical as an extension for transferring heat out under given circumstances. The heat conduction through the copper fins assumed to be constant and uniform over entire surface for the convenience in this analysis.

The amount of heat transfer usually much lower at the fin base compare to the tip area since the fluid is surrounded by solid surfaces near the wall. This manifestation can be spotted in figure 8(a), the area of copper tube thickness at boundary condition is set to 49.3^{o}C and the temperature which has been set as wall in the boundary condition conducted through the wall where the air flow from outside at 25^{o}C flow over the heated finned tube section. The area turning lighter blue after the simulation indicated that the heat has been transferred out of the region by the flow of cooler air. Meanwhile, the tip of the fin started to heat up since the heat has been conducted uniformly to the edge of the fins.

The area where undergo velocity inlet flow first hand also experience more heat transfer which underlined by the blue area of the half section of fin compared to other half which in yellow and green contours. The temperature of the surface for the fin is 318 K or 44.85^{o}C in average which lies within the range of copper tube and air flow, which 49.3^{o}C and 25^{o}C respectively. The small reduce in temperature is less effective to supply the system with cooling effect necessary to maintain the sufficient heat transfer, as keeping the temperature of bearing in gas turbine at par with ideal values. This leads to the decision of replacing this old design to the new one with the expectation of the heat transfer and cooling effects will be higher and effective.

3.3.2 New Design of Finned Tube

The new design of finned tube subjected to few changes as the refinement of the model aims to increase the practical requirement and effectiveness of the system as a whole. This is important since the systems connected to each other as whole gas turbine operation supported by other small units which very cardinal to the whole plant performance.

The revamped new design material has been changed from copper to aluminum. Theoretically, aluminum thermal conductivity is less compared to copper but practically, aluminum is more in favors from the aspect of material cost and physical properties. Aluminum is well known as it appears as material with lower cost of procurement and undemanding process of handling since it physically lighter than copper. To bounce this disadvantage of low thermal conductivity of aluminum, the copper tube has been designed specifically in definite size and length so that the gap for heat transfer will be more open for the heat flow to leave the system. This geometrical enhancement will be discussed in details in section 4.3.

The same mechanism of heat transfer by heat conductivity occurred in the new design followed by cross flow of forced convection of the air flow. This direct contact heat exchange with built-in aluminum fins enables the system to operate at a lower temperature difference and reduce the weight of the equipment. There are few advantages of this type fin and plate heat exchangers are larger transfer area, five times lighter if compared to shell and tube heat exchanger and it can withstand high pressure constraints. The disadvantages of this heat exchanger that could be underlined are clogging cooling water pathways and aluminum is susceptible to brittle failure after certain period of time.

The representation of temperature contours from the simulation provides us with the behaviors of the model when heat was applied to the system. From figure 8(b), we can see that the fins experiencing high degree of heat transfer from the temperature distribution of the heat. The tip of the fin appears to have higher temperature intensity than the base plate near the copper tube. This draws out that the heat started to accumulate at the aluminum-built area where the occurrences of this phenomenon in certain period of operation duration will lead to deterioration in the structure microscopically which eventually will upshot in obvious macroscopic disfiguration of the structure.

From the perspective of heat transfer efficacious, the temperature distribution contours converge at only at the tip spot of the fins as in figure 8(b) undergo the induced cross flow occurred driven by the blower fan leads to more heat transfer. The surface temperature of the fins simulated to be 308 K or 34.85^oC which within the ideal range of temperature gradient of air flow and copper tube temperature which respectively at 25^oC and 49.3^oC. This portrays the effectiveness of the finned tube has been enhanced with the replacement of the fins material.

3.3.3 Material conduction from temperature contours result comparison

The temperature contours of both design exhibited a very distinct and specific pattern for each which makes it possible to interpret and compare the outcomes. By comparing these two results, this will provide the sufficient information in making judgmental decision whether or not to upgrade the design and forecasting the temperature of cooling water leaving cooling unit.

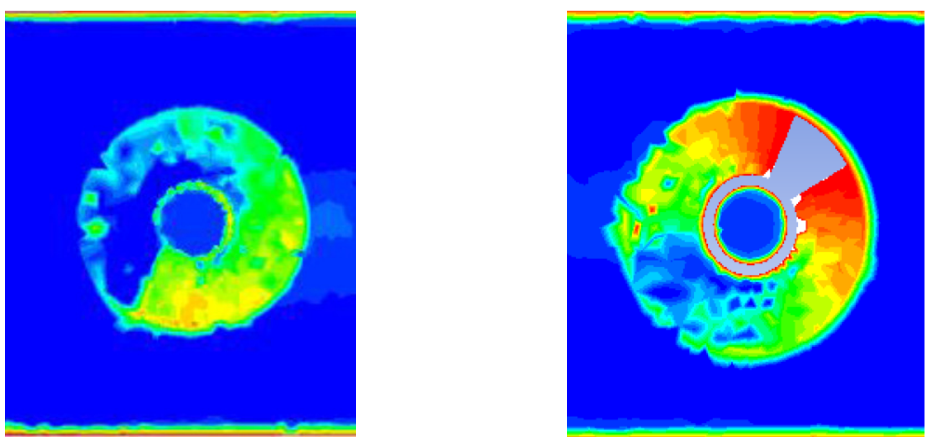
As we can see from figure 8(a) and 8(b), the temperature contours enacted very prominent discrepancy. The temperature contours from old design of fin fan in figure 8(a) simulated to be having more evenly distributed heat contours throughout the geometry. This is very different to be compared to the new design of fin fan which in figure 8(b) when the distributed heat seems to be focus at the edge of the fin. This deviation cause by the variation of the material used for the fins, which resulted in different thermal conductivity coefficient, k. Higher value of k will gave substantial effects of the heat transfer rate since the value depended on k values This observation justify that in terms of fins material of thermal conductivity, the old design fin which produced from copper have higher efficiency compared to the new design, aluminum.

As a whole system, the finned tube will be cross flow with air from the surrounding which will be force convection by fan blower. In order to counter the effect of low heat transfer in the new aluminum finned tube design, the velocity inlet for the new design is set to higher value compared to the old one. The setup results in more lower surface temperature in new design of finned tube compared to the old one despite their values of thermal conductivity coefficient, k. The surface temperature and velocity of the design are tabulated in the table below.

Table 3: Air velocity and surface temperature data

	Old Fin Fan	New Fin Fan
Air Velocity (m/s)	8.182	10.87
Surface temperature (°C)	47.85	34.85

The new design of the fin fan acted out more effective than the old design due to the surface temperature reading. This proven that the new design of fin fan enhances the performance of the system as a whole from the perspective of heat transfer, cost effectiveness and structure reliability.



(a) (b)

Figure 8: Temperature distributions of finned tube

3.4 EFFECTS FROM DIFFERENT GEOMETRY DIMENSION

The last enhancement that we will discuss in this chapter is the alteration of tube geometry. The alteration has been made at the thickness of the copper tube. This thickness alteration initiated by the modification of tube inner and outer diameter. Different dimension of model geometry most likely to create different streamlines of the flow which can be a way of evaluating the efficiency of the designs. As for this analysis, the new design of fin fan appears to have augmentation for their dimension to be compared to the old design

3.4.1 Old Design of Finned Tube

The old design of the fin fan has the inner diameter, D_i and outer diameter, D_o of 12.7 mm and 13.35 mm respectively. This measurement sums up the thickness of the tube as 0.325 mm as it

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
ISSN (online): 2289-7879 | Vol. 14, No. 1. Pages 8-23, 2015

acts as a channel to supply the cooling water into and out the lubrication system. Streamlines are the lines that tangential to the velocity vector throughout the flow field. Streamlines depends on the boundary conditions and dimensionless parameters assigned and obtained from the simulation. The streamlines are kept at constant by structure stability which by means majorly defined reciprocates by convective instability. Convective stability occurred when adiabatic cooling or heating take place which basically cover the basis of rising and descending of the air.

The streamlines of the flow in old design finned tube when viewed from y-axis seems to establish pattern consistency which basically originated from the geometry of the model itself. The presence of the specific copper tube create a boundary for the streamlines to get through and this leads to air flow vertical displacement. This is more likely to happen especially when the convective instability commonly appears to be occurred in turbulent flow and convective activity. The streamlines obviously to have symmetry pattern if viewed from velocity inlet flow.

3.4.2 New Design of Finned Tube

The new design of finned tube has been modified to have larger values compared to the old fin fan by having the inner diameter, D_i and outer diameter, D_o of 14.224 mm and 15.875 mm respectively. This alteration leads to the thickness of 0.8255 mm of the copper tube. The magnification of the tube size leads to bigger area of finned surface which increases the area of heat dissipation in the system. This finned surface are extended to different length as in this new model, the fins are made from aluminum.

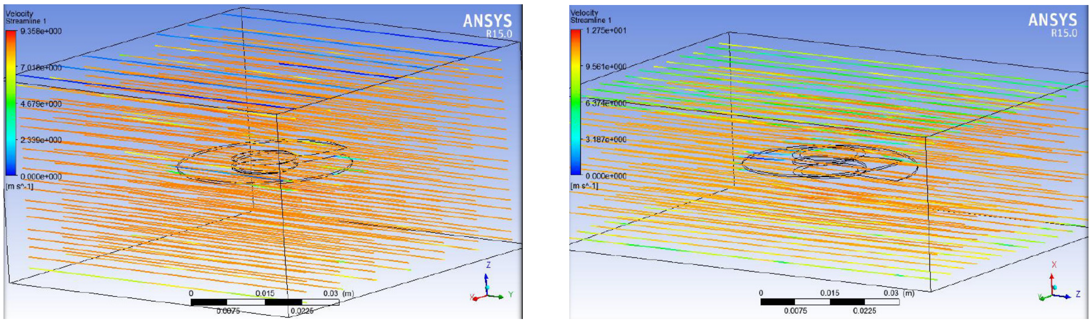
In the analysis of fins, the fins are considered to be in steady state operation and no heat generated within the fin, as their thermal conductivity, k remained constant. The convective heat transfer coefficient, h also recognized to be inconsistent along the fins according to the length of the circumference. Other than that, the value of the it also represented as a influential function of the fluid motion at the corresponding point.

From figure 9(b), the streamlines of the flow in new fin fan design graphically represented as asymmetry, which completely contrast the streamlines of the old design represented in figure 9(a). This contrast is basically denoted by the difference in tube geometry when the conduction in between the tube and fins happened. As the conduction creep on the model, it is theoretically given that the tip of the fin is usually hotter compared to the base of the fin or the copper tube. The area with higher temperature has higher velocity as the air flow over the fine, so the higher velocity have the tendency formed more tangled streamlines once it flow over the plate, causing the streamlines to behave erratically. This difference is an upshot from the modification of the tube geometry, which we really need to consider in the analysis.

3.4.3 Flow Streamlines within Geometry Result Comparison

As we can see and compare graphically from both figure 9(a) and 9(b), the streamlines of both old and new design exhibit significant variance in their patterns. This might be caused by area of surface exposed to the forced convection.

As stated in boundary condition, the copper tube inside fin area opted to undergo wall conduction as the temperature set to 49.3^oC and subjected to conduct throughout the fin. The heat mostly conducted up until to the tip of fin so the heat can be extended as far as it can from the heated base. So, the streamlines of the new finned tube section mostly pivoted at the tip of the fin and the copper tube area, as the area are more heated and the velocity is higher when exposed to hot area.



(a) (b)

Figure 9: Streamlines of the flow in fin fan design

In conjunction with that, the increment of the tube new geometry creates turbulence for the streamlines and lead to erratic behavior of streamlines patterns causes the streamline to be not equivalence graphically. This is what differentiates the old and new design where the size of old finned tube is smaller and the distribution of the hot area are smaller. The smaller area distribution means the hot area somehow cannot be properly distinguish by the simulation which leads to more obvious and organized streamlines.

This discrepancy can also be examined by using the rate of heat transfer from any surface at any temperature, T_s to the ambient medium with T_∞ which denoted by Newton's law of cooling such

$$\dot{Q}_{conv}=hA_s(T_s-T_\infty);$$

where A_s is the heat transfer surface area and h is the convection heat transfer coefficient. From the equation we can see that, with the increment of A_s with fixed value of heat transfer coefficient and temperature gradient, the value of heat transfer will theoretically increase to any significant amount. This proves that the changes and modification of the geometry produced the significant effect to the performance of the finned tube as a unit.

4.0 CONCLUSSION

From the research, we stumbled upon ways to identify and analyse the new and old design of finned tube which enable us to understand the system as a whole. The first step to identify the modification of the new fin fan design followed by the analysis of the alteration and lastly, compares their significant effects to the effectiveness of the design and the system itself. The presence of the crucial parameters for the simulation of flow such as temperature and velocity govern the flow representation from different standpoint such as velocity profile, temperature contours, dimensionless parameters, and streamlines with cover the basis of different enhancement of the system.

Simulation was chosen as the ultimate tool of the analysis and through it, we can generate accurate solution with the utilization of less cost. The critical constraints of doing flow simulation is the recognition of necessary and suitable boundary condition as each case of the analysis requires specific set and values of the constraints. This reliable boundary condition is really important to visualize our case into simulation that resembles the pragmatic and accurate real condition. Instead of this long and elaborated explanation about simulation, this flow analysis also can be analysed by using calculation or experiments which respond accordingly to our scope of study and method of selection.

Besides that, within this study, we might find few inconsistency in analysis especially in the replacement of fin material from copper to aluminium as the changes can be misunderstand as the downgrading the material efficiency but the external factors such the material cost and structure durability appears to be the limitation that need to be included in the analysis and discussion. The inconsistency has been bounced by the adjustment of the related boundary condition such as increased velocity and appropriate material selection.

Furthermore, the boundary conditions and changes of the model influenced the several of temperature contours, velocity profile and streamlines obtained. We can spot various results regarding the responds of the systems to the boundary condition. Some changes can be smooth and organized and some other can be erratic and unstable, which lead to the formation of low pressure stagnation point and whirlpool. This inconsistency in system responds lead us to believe that the modification of the design, material selection for the fins and velocity inlet variety influence the flow pattern inside the finned tube section within the domain, as well as the analysis of the flow.

As a conclusion, this project successfully achieved the objectives proposed in the beginning of the study. The objectives are achieved steps by steps with structured solution, analysis and recommendation for the system to behave more independently and effective. But, the main objective which to study the flow of cooling water in gas turbine fin fan cooler in utilities plant is fulfilled and finally came out with several recommendations that can be suggested after analysing the results from the simulation.

ACKNOWLEDGMENT

The authors wish to thank the Faculty of Mechanical Engineering at the Universiti Teknologi Malaysia for their support and cooperation during this study. The authors also wish to thank Research Management Centre (RMC) for the Research University Grant (02G34, 02G35, 09H64) from the Universiti Teknologi Malaysia and Fundamental Research Grant Scheme (4F610) from the Ministry of Higher Education for their financial support.

REFERENCES

[1] M. Colleen, Design of a combustion turbine heat recovery rotor air cooling system, Society of Mechanical Engineers (1992).

[2] G.N. Xie, Q.Y. Chen, M. Zeng, Q.W. Wang, Thermal design of heat exchanger with fins inside and outside tubes, Proceedings of the ASME Turbo Expo (2006).

[3] Y.J. Kim, M. Kim, M.Y. Ha, J.K. Min, Numerical study on surface air-oil heat exchanger for aero gas-turbine engine using one-dimensional flow and thermal network model. Transactions of the Korean Society of Mechanical Engineers 38 (2014) 915-924.

[4] A.D. Al Dakhil, J.A. Al Sudairy, I.A. Al Buraiki, Failure analysis at steam condenser fin fan cooler tubes, Ras Tanura Refinery, American Society of Mechanical Engineers. Pressure Vessels and Piping Division 3 (2012) 483-488.

[5] M. De Paepe, A. Willems, A. Zenner, Experimental determination of the heat transfer coefficient of a plate-fin heat exchanger, Heat Transfer Engineering 26 (2005) 29-35.

[6] G. Lozza, U. Merlo, An experimental investigation of heat transfer and friction losses of interrupted and wavy fins for fin-and-tube heat exchangers. International Journal of Refrigeration 24 (2001) 409-416.