

Computational Model Fouling of Air Cooled Condensers on The Air Side

M. Francis¹, K. Durga Prasad²

¹PG Scholar, Dept of Mechanical Engineering, V.K.R, V.N.B & A.G.K College of Engineering, AP, India.

²Assistant Professor, Dept of Mechanical Engineering, V.K.R, V.N.B & A.G.K College of Engineering, AP, India.

Abstract - As the electrical power demand increases and water resources become more limited, fouling on the air side of Air Cooled Condensers (ACC) is a growing concern. ACC's are widely used as a method to exhaust waste heat from power plants to the environment while using very little water. Generally fouling on the air side is neglected but with wider implementation of ACC, demands need to be considered to maximize efficiency. Air fouling is partially due to the following: pollen, dust, insects, leaves, and large debris. Fouling limits the convection heat transfer coefficient of the condenser and can cause an increase in backpressure to the turbine or incomplete condensing. Either case will result in a decrease in the plant efficiency and power output. The objective of this study was to experimentally and computationally calculate the convection heat transfer coefficient for both a clean and fouled condenser. Bee pollen was selected as the experimental fouling particle, and engineering data for similar particles were used for the computational model in ANSYS Fluent. Both the experimental and computational results have similar trends showing pollen has a negative impact on the heat transfer. The experimental results show between a 15% and 20% reduction in the coefficient of heat transfer while the computational results show between a 6% and 9% reduction. Suggestions for future work are included to further improve upon this research.

I. INTRODUCTION

1.1 Rankine Cycle

The burning of coal, natural gas, and nuclear fission generally follows a steam cycle, where the heat released by these fuels boils water resulting in steam. A simple steam cycle known as the Rankine Cycle is shown in Figure 1 (Çengel). The heat into the boiler, q_{in} , is the heat released by the fuel and added to the system as shown in Figure 1, the Rankine cycle contains four components: Pump, Boiler, Turbine, and Condenser. Each of these components has a complex thermodynamic process associated with them; however for a simple Rankine cycle the processes will be simplified. Each of these components and their processes are explain in the sections following. It is common practice in the power industry to have more than one of each component to help boost efficiencies;

however one of each component is a good simplified model. As a further simplification of the power plant model only water is used as the fluid which is not the case for actual power plants. The simple Rankine cycle also neglects effects pressure drops and heat losses in the pipes between the components. It is important to keep in mind that a perfect model of a power plant does not exist, and that the Rankine cycle is the best approximation for maximizing component efficiency. This research focuses on the condenser and how its efficiency degrades over time.

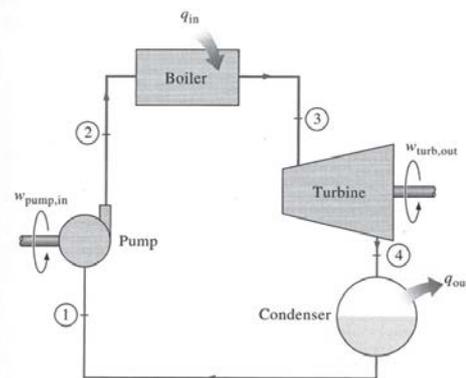


Fig.1. Simple Rankine Cycle

1.2 Condenser

Between stages 4 and 1 is the condenser is where latent heat is rejected to the atmosphere allowing the steam to condense back to liquid water. This process is conducted while holding temperature and pressure constant. The condenser allows for a complete cycle, and allows the pump to operate effectively since pumps cannot pump steam to high pressure at large flow rates required by power plants. There are many condenser types that will be discussed in the next section. As shown in Figure 1 q_{out} , the rejected heat, of the condenser can be expressed by:

$$q_{out} = h_4 - h_1$$

Where h is the enthalpy of the water at the corresponding stages. Traditionally the condensing process uses more water than any other power generation processes combined. With laws limiting water usage for the power generation industry, it will be important to find a way around using so much water for the condensing process.

Figure 1 shows how much water is consumed by process for a range of power generation types (Power Mag). New condenser designs dramatically reduce the amount of water needed for the condensing process and thus are becoming more prevalent.

1.3 Condensers Designs

As stated in the previous section, the condenser rejects heat to the atmosphere, allowing the steam to condense back to a saturated liquid. In the ideal case this should be done at constant pressure. In reality this is not possible as most condensers consist of large lengths of pipes. Due to friction in pipes a pressure drop, ΔP , exists and is defined by:

$$\Delta p = \frac{f \rho l v^2}{2D}$$

Where f is the Darcy-Wiesbach friction factor obtained from the moody diagram, D is the diameter of the pipe, L is the length of the pipe, ρ is the density of fluid at an average temperature, and V is average velocity. The moody diagram yields friction factors based on Reynolds Number, Re , and relative roughness equation, r , where in addition to those defined previously μ , is absolute viscosity and e , is surface roughness, both based on material of the pipe.

$$Re = \frac{\rho v D}{\mu}$$

$$r = \frac{e}{D}$$

Understanding that it is impossible to keep the pressure constant, the turbine and the pump are designed or selected to have a difference in pressure equivalent to the pressure drop in the condenser. If the pressure drop in the condenser is greater than the difference between these devices, the steam will not be expanded completely, decreasing the efficiency of the turbine and consequently limiting the electrical power output.

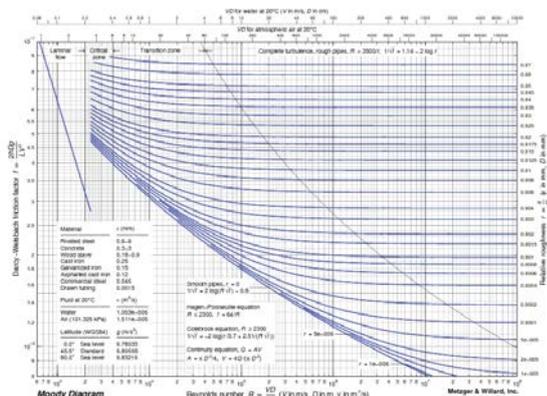


Fig.2. Moody Diagram

In order to increase efficiency of the overall plant, it is important to design the condenser to have the smallest possible pressure drop. This can be achieved by material selection as well as by building the condenser as small as possible, decreasing length.

Various types of condensers are used by the power industry including: shell and tub heat exchangers, cooling towers, and air cooled condensers (ACC). Each of these has their own unique application, along advantages and disadvantages.

1.3.1 Shell And Tube Condensers

Shell and tube heat exchangers are one of the oldest forms of condensers in power plants. Steam from the turbine enters one side of the tube and exits at the other side of the tube as a saturated liquid. Meanwhile a stream of water is passed across the tubes through what is known as the shell side. The shell side does not change phase, remains liquid, however it exits with a higher temperature than it entered with. The liquid for the shell side is provided by a large body of water neighboring the plant such as a lake, ocean, or river. The heated liquid is then fed back to the same body of water. A schematic of a shell and tube heat exchanger is shown in Figure 1.6.

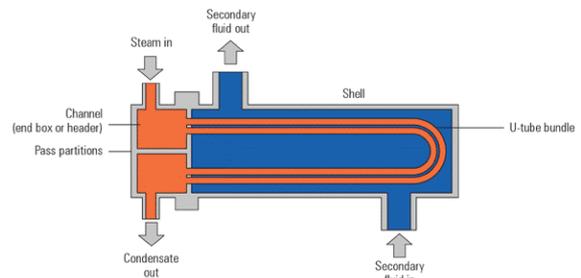


Fig.3. Schematic of Shell and Tube Heat Exchanger

Advantages of the shell and tube heat exchanger include its compact size, due to high heat transfer rate between the shell and the tube sides. With the size and the age of these condensers, cost and reliability are also advantages. However, the age of this type of condenser limits the research topics associated with it. Disadvantages of the shell and tube condenser are centered on locating a power plant near a body of water and legislation limiting the use of water for industrial use. Some legislation stipulates that the shell side water cannot be fed back to the body of water it was taken from unless it meets strict temperature criteria. This ensures that wildlife around the power plant will not be affected. Therefore this type of condenser cannot be used without meeting the necessary specifications (PowerMag). Another problem is that this type of condenser requires periodic cleaning, as the water on both shell and tube side leaves deposits known as fouling which over time limits the heat transfer and increases the pressure drop, both of which decrease the

overall plant efficiency. In order to clean either the shell or the tube side the whole plant must be shut down.

1.3.2 Air Cooled Condensers

A third option for condensing steam for the power industry and the scope of this research is the air cooled condenser. The air cooled condenser or dry cooling works simply by passing steam through pipes as atmospheric air is forced across them. A schematic of an air cooled condenser is shown in Figure 4. Steam exiting the turbine enters the top of this condenser at what is known as the steam inlet manifold. From here the steam flows downward through pipes known as condensing rows. A fan pushes air upward across these rows creating forced convection. When the steam reaches the bottom of these rows it is in what is known as the row header and the steam should be entirely condensed back into a liquid. At this point it is ready to enter a pump and start the Rankine cycle over again. There is a path back up towards the top of the condenser known as the venting row, allowing any high pressure non-condensed steam to escape to the atmosphere.

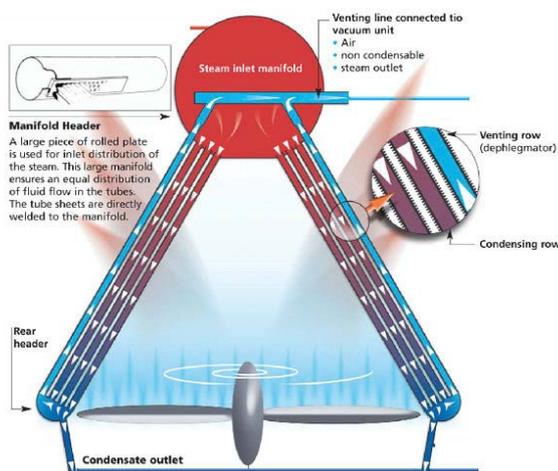


Fig.4. Schematic of an Air Cooled Condenser

Dry cooling is becoming a popular method of condensing steam because the unit is completely closed and there is no water required to cool it. Only the water enclosed within the Rankine cycle is needed for the entire plant. This allows power plants to be built away from bodies of water compared to those with the shell and tube heat condensers and the cooling tower. The Gateway Generating Station in Antioch, California uses 97% less water with its new air cooled condenser than when it used cooling towers (PowerMag). In order to comply with the legislation in the Southwest United States limiting water for industrial use, a great amount of research is being done to improve the efficiencies of this type of condenser.

As with the shell and tube condenser fouling can be a problem with dry cooling. However, fouling rarely occurs on the inside of the pipes due to anti-corrosion chemicals that are added to the water. The outside of the condenser

that is exposed to air may need to be cleaned periodically. Unlike the shell and tube condenser, the plant does not need to be shut down for such maintenance. Companies that design and build Air Cooled Condensers generally include automatic cleaning robots with the condenser. Fouling on the air side is the scope of this research and will be discussed in detail in the next section.

The main disadvantage to the air cooled condenser is the size of it. Because convection heat transfer to air is low, the heat transfer capacity must be made up with an increase and surface area. Therefore the cost also increases, due to the size increase of the condenser. Figure 1.10 shows an air cooled condenser for a 500MWe plant. In addition air side fouling lowers the convective heat transferability to condense the liquid.



Fig.5. Air Cooled Condenser for a 500MWe Plant

1.4 Fouling

Fouling is the deposition of unwanted particles or species on a solid surface restricting the fluid flow or the performance of a heat transfer surface. Fouling types include but are not limited to precipitation, particulate, corrosion, chemical reaction, solidification, biofouling, and composite fouling. Fouling increases pressure drops in pipes which consequently limits the fluid flow through pipes. The fouling acts as a thin layer of insulation on the heat transfer surface, which causes the decrease in heat transfer. Figure 6 shows a cutaway of pipe with calcium carbonate deposited on the surface. It can be seen that the diameter will be reduced which will increase the pressure drop according to equation 1-5. Also if this pipe were a heat transfer surface, the calcium carbonate would act as a barrier, limiting the heat transfer in or out of the pipe.



Fig.6. Pipe Cutaway with Calcium Carbonate is Deposited

For this thesis the reduction in performance of heat transfer surfaces will be studied for particulate fouling. An example of particulate fouling is shown in shown in Figure 1.12. This photograph shows dust particles on cooling fins for a air conditioning condenser. This reduces the efficiency of air conditioners and is cleaned routinely blowing compressed air through the fins.



Fig.7. Particulate Fouling on Heat Transfer Fins

Fouling on the air side of air cooled condensers is a known problem to the industry. However little research has been published on the effects of air side fouling on the heat transfer and the efficiencies of air cooled condensers. Power plants using air cooled condensers have noticed improvements in the efficiencies of their condensers after cleaning. Rosebud Operating Systems in Billings Montana uses a cleaner on their Air Cooled Condenser, containing more the 1.6 million square feet of surface area. After cleaning, Rosebud notices a 10% boost in the efficiency of the condenser, resulting in the recovery of 3,000MWh per year, which is approximately \$180,000per year (Conoco). An example of Conoco's automated cleaning system is shown in Figure 1.13. Ideally plants will develop regular cleaning schedules to maximize efficiencies. These schedules will depend on geographic location and season, as these two variables will determine and greatly affect the fouling rate.



Fig.8. Conoco Cleaning System

1.5 Scope Of Research

The primary goal of this research was to experimentally obtain and theoretically predict the convection heat transfer coefficients for clean and fouled air cooled condensers. In order to conduct experiments, a condenser was first constructed made from a 60' coil of 7/8" copper tubing. Further design specifications are explained in Chapter two. The experimental results were first taken for the clean condenser at varied forced air speeds and then for the fouled condenser at the same forced air speeds. For fouling, bee pollen was chosen to be the only contaminant considered. Once the experimental results were obtained, the impact that pollen has on the heat transfer coefficient was determined. A computational fluid dynamics (CFD) model was constructed using the commercially available software ANSYS Fluent to compare with experimental results. The CFD model was then run at a higher air velocity than the experimental setup could produce. This model allows for a correlation of how pollen deposition and air velocity affect the heat transfer coefficient. The convection heat transfer coefficients of the experiment were compared to those of the CFD model. It was assumed that the experimental results were more accurate than the computer model.

II. COMPUTATIONAL MODELS

There is no closed form solution to the governing equations that model the system studied in this research. Therefore numerical techniques must be applied to these equations in order to obtain an approximate solution. Computational fluid dynamics (CFD) software such as ANSYS Fluent has been developed to numerically solve fluid flow and heat transfer for complex problems. The next sections discuss the methodology utilized by the software to find the convection heat transfer coefficient of the modeled air cooled condenser.

2.1 Computational Fluid Dynamics

ANSYS Fluent uses a numerical finite volume method of solution. The finite volume method takes the partial differential equations and solves them algebraically on the grid in a discrete manor. Each element or finite volume has its own set algebraic equations applied to it. As these elements get smaller the algebraic equations become closer to a continuous solution rather than a discrete solution, satisfying the original partial differential equations. The finite volume method is a conservative method, where flux leaving a volume is equal to the flux entering the adjacent volume. This is largely important as Fluent is numerically solving equations based on conservation.

Fluent takes complex geometries and applies a finite volume grid or mesh on to the geometry. Using laws of conservation Fluent then numerically solves the fluid flow and heat transfer governing equations on this grid. The

user sets the error and Fluent continuously iterates using C programming code until solution contains the convergence criteria set by the user.

2.2 Method of Modeling ACC

To simulate the fluid flow and heat transfer of the Air Cooled Condenser the following steps were used for each model.

- The geometry of the fluid domain for the Air Cooled Condenser was drawn in Solid Work computer aided design software.
- The geometry was divided into finite volumes using ANSYS's meshing package.
- Appropriate fundamental laws were applied such as Energy and Turbulence.
- Physical properties of the fluid and interacting surfaces were imposed along with their boundary conditions.
- Convergence criteria were set.
- Analysis was run until convergence was met.

2.3 Clean Condenser Models

The first model, the quarter model, was drawn in Solid Works and imported into Fluent, Figure 9. The green coil is the heat transfer surface, simulating that of the experimental setup. The coil has a diameter of 7/8inch, a radius of curvature of 9inch, and a pitch of 5inch. The fluid zone is 22inches wide by 22inches deep by 20inches tall. These dimensions are approximately the same as one fourth of the experimental model.

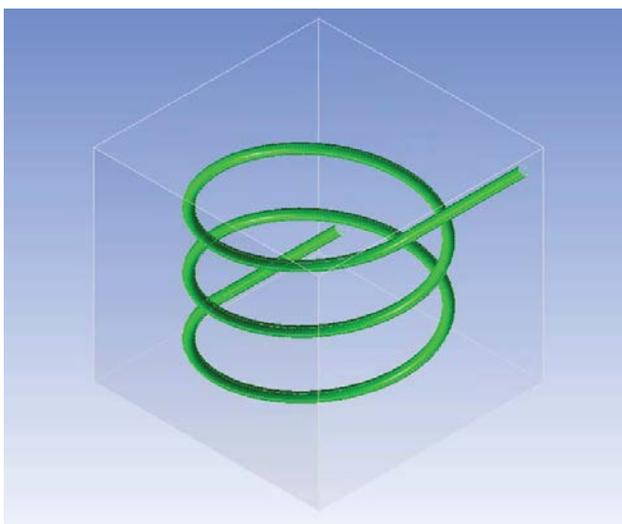


Fig.9. Geometry of Quarter Model

Similar to the quarter model, the full scale model can be seen in Figure 5.2. It has the same dimensions other, than the overall height which is 72inches due to the 12.5 coils.

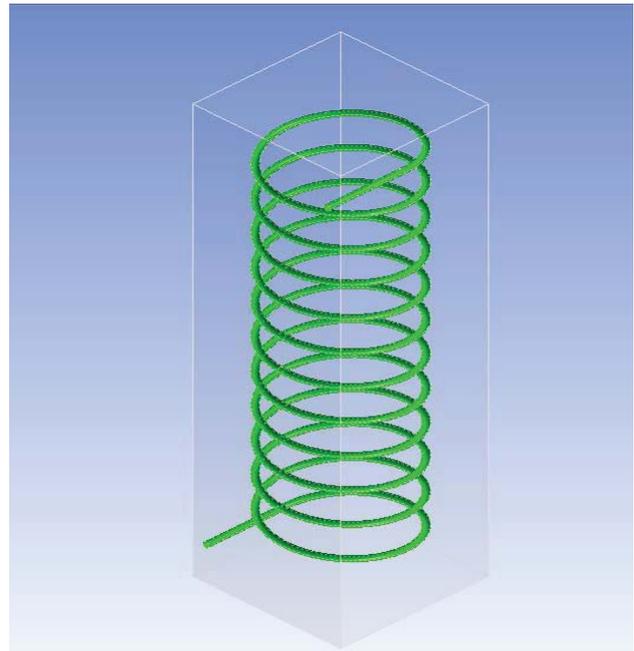


Fig.10. Geometry of Full Model

2.4 Clean Condenser Meshing

The next step was to mesh the models using the automatic meshing software in ANSYS. As previously stated, this breaks the solid geometry into a finite number of control volumes upon which the governing equations are applied. Figure 10 is a close up of the mesh near the heat transfer surface. This area needs to be extremely fine as the temperature gradient is quite large. A fine mesh was imposed along the coil by applying Face Sizing criteria available in ANSYS to the surface of the coil. The Face Sizing criterion forces any length the element on the surface selected to be a less than a given length input by the user. This allows the mesh along the coil to be fine while keeping the rest of the mesh where temperature gradients are lower at larger size. This helped reduce the total number of elements.

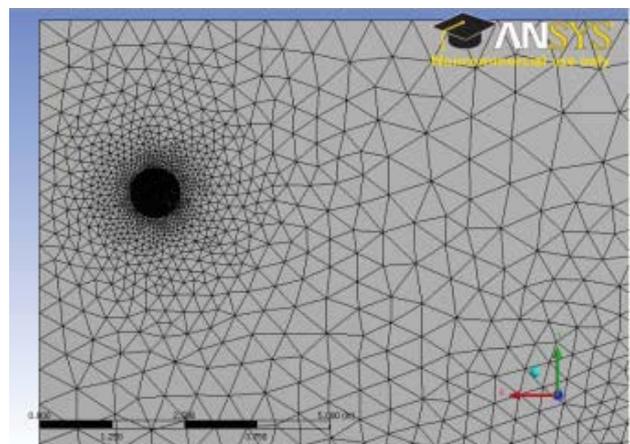


Fig.11. Medium Mesh

The first mesh, with a face sizing criteria of .05inches, was considered a medium mesh as a finer mesh was created to show that the results have converged. That is that the solution was basically the same for a finer mesh size.

Figure 10 shows the fine mesh that was created in the same fashion, but with smaller Face Sizing criteria, for the quarter model. The same process was repeated for the full scale model shown in Figure 11 but only one mesh was created. The Face Sizing criteria, total elements and total nodes can be found in Table 1 for the three meshes. The quarter models were assumed to be more accurate as they have more elements and nodes where the temperature gradient will be the greatest.

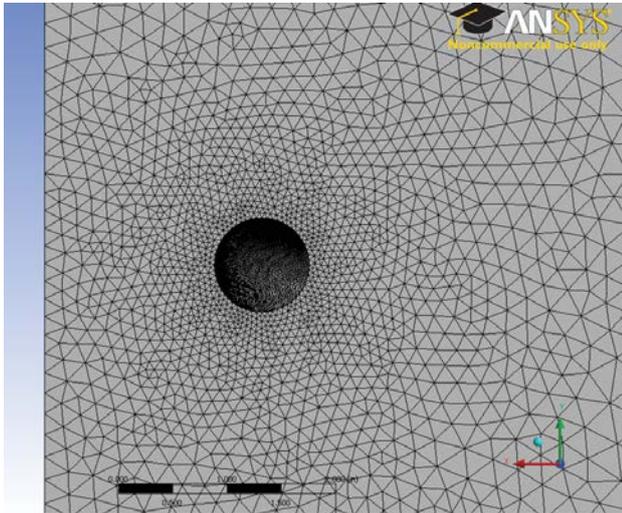


Fig.12. Fine Mesh

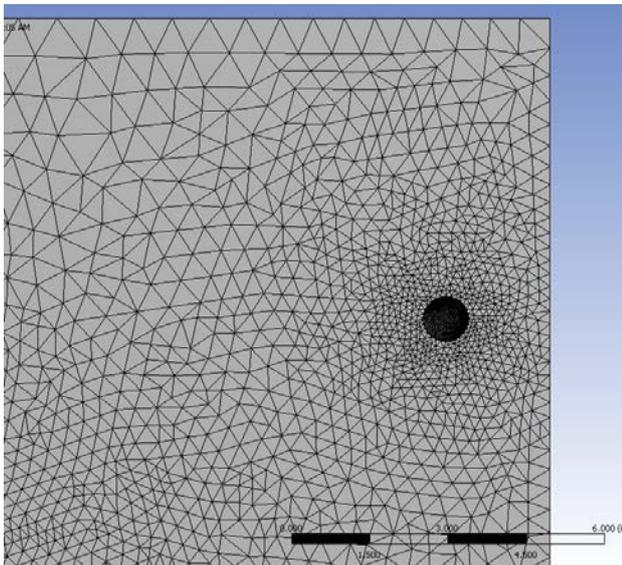


Fig.13. Full Scale Mesh

Table 1 Mesh Statistics

Mesh	Face Sizing Criteria	Elements	Nodes
Quarter Model Medium Mesh	.05 inches	11251407	1978882
Quarter Model Fine Mesh	.03 inches	24757118	4247731
Full Model Mesh	0.1 inch	25618943	4400309

An element quality mesh metric was applied to the three meshes. This checks the quality of the mesh and helps deem if the mesh is fine enough to yield accurate results based on tetrahedron element shapes. The element quality mesh metric takes each element and rates it from zero to one based on the following. Elements that have a number close to one are considered to be of high quality while zero is low quality. Figure 5.6 shows the element quality graphs for the following meshes: Quarter model medium mesh, quarter model fine mesh, full model mesh. It can be seen the most of the elements are acceptable and that the fine mesh is better than the medium mesh. The full scale mesh has good elements overall but there are not as many in critical areas which could result in a non-converged solution.

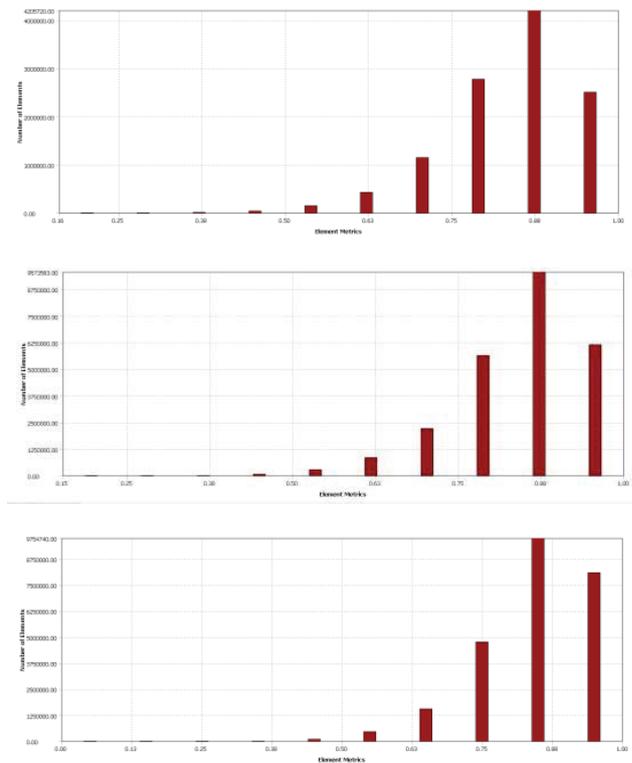


Fig.14. Element Quality

Table 2 Heat Transfer Results

	Air Velocity (fps)	Heat Transfer (btu/lbm)	Coefficient of convection (btu/lbm·ft ² ·°F)
Quarter Model Medium Mesh	3.5	4164.12	9.87
	5	4552.55	10.79
	7	4992.79	11.83
Quarter Model Fine Mesh	3.5	4325.27	10.25
	5	5012.98	11.88
	7	4973.97	11.79
Full Model	3.5	17647.93	9.76
	5	19564.62	10.82
	7	20812.27	11.51

Figure 9 shows the temperature contours for the quarter model fine mesh at 3.5 fps. These contours were similar with all models. This confirms that the exit temperature of the air is the same as the inlet temperature.

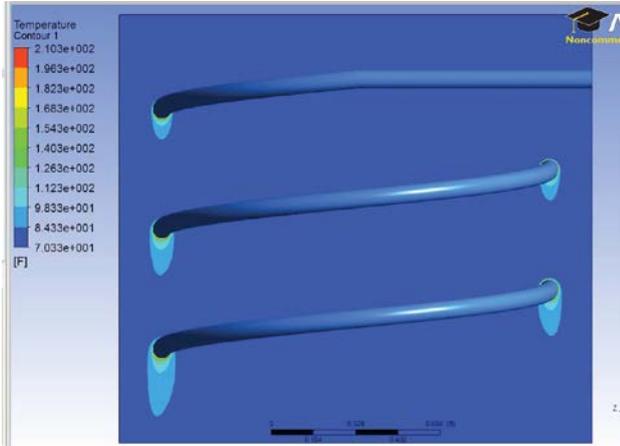


Fig.15. Temperature Contours

Figure 3.2 shows the velocity vectors on the condenser as well as a close up showing how the air flows around the pipe. This Figure is from the quarter model fine mesh as 3.5 fps. Again, all other models show similar results.

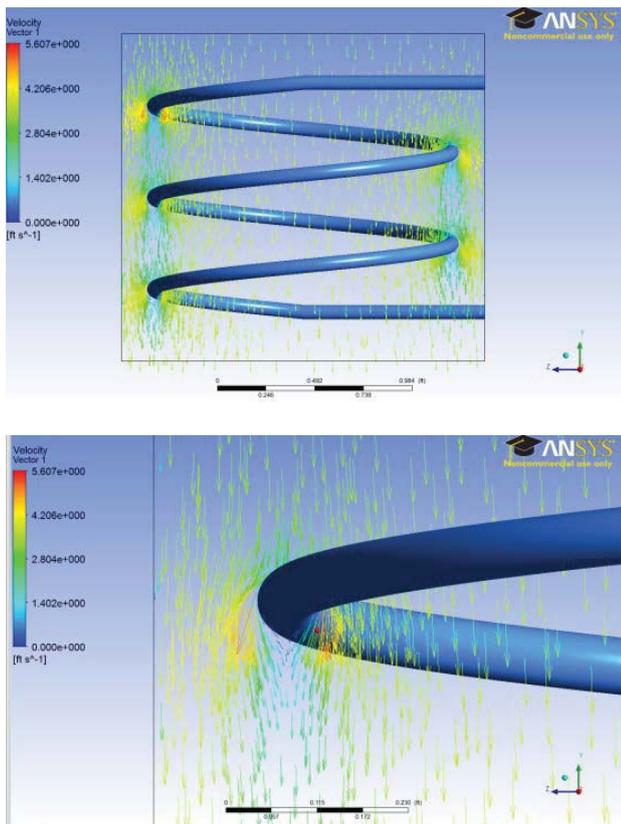


Fig.16. Velocity Vectors

III. CONDENSER

3.1 Fouled Condenser

The next step was to create a model that shows how pollen would be deposited on the surface of the pipe. Using the

same methodology as the clean condenser, a fouled condenser was setup and run with particle deposition.

3.2 Fouled Condenser Geometry And Mesh

The geometry for the particle deposition model had to be simplified, as the computing time can quickly become prohibitive. Figure 17 below shows the geometry that was used. It consists of a 10inch long straight pipe that is 7/8inch in diameter, with the fluid zone enclosed in a 10inch cube. It was assumed that the curved geometry has no significant impact on the heat transfer coefficient and therefore was acceptable.

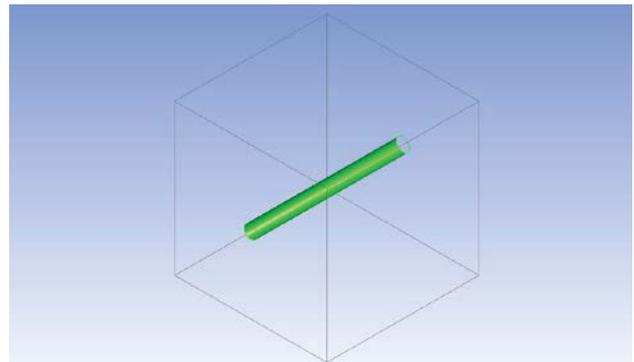


Fig.17. Fouled Condenser Geometry

The mesh was setup in the same way as previously explained using a face sizing grid on the heat transfer surface of .03inches, resulted in 431346 nodes and 411800 elements. A close up of the mesh by the heat transfer surface is shown in Figure 18. A mesh metric analysis showed that the mesh had similar element qualities to that of the clean models and therefore was acceptable.

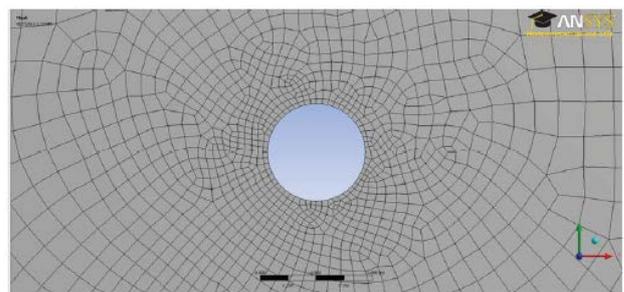


Fig.18. Mesh of Fouled Condenser

3.3 Fundamental Laws And Boundary Conditions For Fouling

Like the clean condenser, the energy and turbulence equations were applied. In order to simulate deposition, a third equation needed to be applied. The third equation known as species transport, allows for particles to be injected and tracked throughout the simulation. This works by tracking the particles velocity. When velocity becomes zero, it is assumed that the particle has been deposited on that surface and will no longer move.

Because of this added equation, more boundary conditions were needed. The top of the condenser previously had a velocity and a temperature. Added to these boundary conditions was an injection condition where it would inject particles having the following: size-45 μ m, mass-6.83x10-8lbm, thermal conductivity-1.2W/m-K. Particles were injected at the rate of 0.25lbm/min for 60 minutes. The thermal conductivity was that of pine wood. This was thought to be acceptable for this initial proof of concept model. The bottom surface, originally prescribed a velocity outlet and temperature, was given an escape boundary condition, such that all particles that arrived at this surface would simply be removed from the analysis. The four side walls, that previously with no slip walls and no heat flux through the walls, were imposed with a reflect condition. This meant that no particle would stick to the wall; it would simply be reflected at the same angle with which it was introduced with. The final altered boundary condition was that on the heat transfer surface. Previously it just had a surface temperature; added to this surface was a trap condition collecting all particles having a zero velocity.

3.4 Fouled Condenser Results

After the simulation was, complete the heat flux from the pipe and the particle mass deposited on the pipe were written to journal files and used to solve for the convection heat transfer coefficient. It was determined that .0206lbm was deposited on the surface of the pipe. The deposition was primarily on the top surface of the pipe. The average mass per area was found to be 7.487x10⁻³ lbm/in². The heat transfer coefficients for the three air velocities are shown in Table 3

3.5 Fouled Condenser Convective Coefficient Results

Table 3

Air Velocity (fps)	Convection Coefficient (btu/lbm·ft ² ·°F)
3.5	9.47
5	10.42
7	10.94

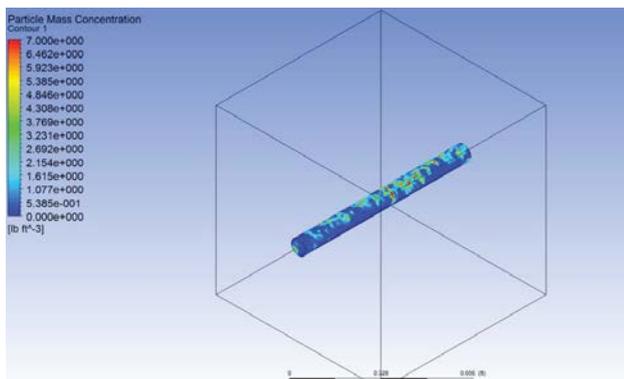


Fig.19. Particle Deposition Contour

The temperature contours and the velocity vectors compare very close to that of the clean condenser. The particle deposition contour is shown in Figure 19

3.6 Fluent Results

Results from each model can be seen in Figures 21,22 and 23 shows the convection heat transfer coefficient with respect to the air velocity where 21 shows the same but with respect to Reynolds Number for possible non dimensional analysis. All the data have similar trends such that a power series trend lines have been fit to the graphs. It can be seen that the three clean condenser models yielded approximately the same result and therefore will be considered to be correct. Of the three clean models the quarter model fine mesh was the most correct; it is compared to the fouled model for reduction in heat transfer and is shown in Figure 22

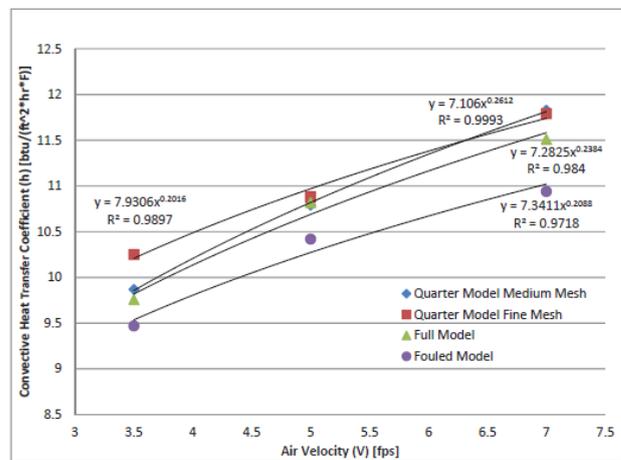


Fig.21. Convection Heat Transfer Coefficient of Models versus Velocity

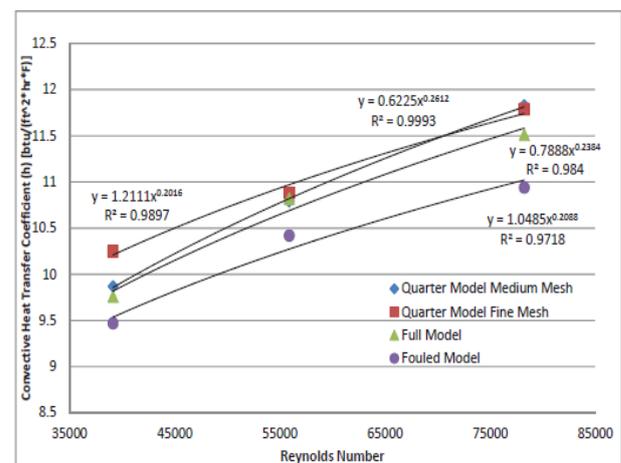


Fig.22. Heat Transfer Coefficient of Models versus Reynolds Number

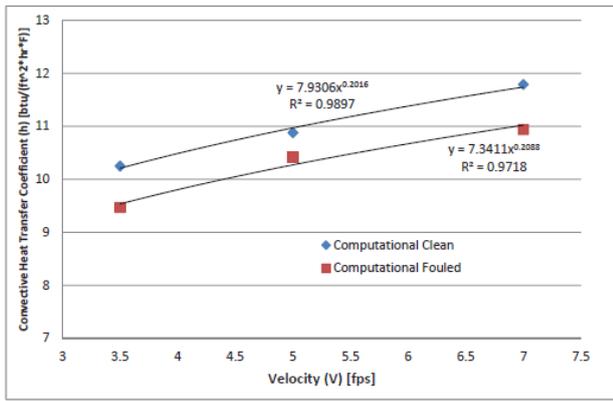


Fig.23. Heat Transfer Coefficient

Using the power equation form the Figure 5.14 for the values of the clean condenser to that of the fouled condenser it can be stated that pollen does degrade the heat transfer coefficient as seen in Table 4.

Table 4 Reduction in Convection Heat Transfer Coefficient

Percent of Heat Transfer Coefficient Reduction Computational	
3.5fps	6.6%
5fps	6.4%
7fps	6.1%

IV. CONCLUSION

4.1 Comparison of Results And Discussion

The primary goal of determining the heat transfer coefficient experimentally and computationally was completed for both the clean condenser and the fouled condenser.

In this chapter comparisons and contrasts are made between the two methods. Since the computational results did not completely match the experimental results, future work is discussed to further improve this research.

4.2 Comparison Of Results

All results, both computational and experimental, show similar trends in that the convective heat transfer coefficient increases with increasing air velocity. The results for the clean condenser are, in particular good agreement with the experimental results indicating more sensitivity of the coefficient to the increased air velocity than the computational results. The fouled condenser shows a similar trend between the computational and experimental results, and in fact at first look seem to correlate better.

However, the computational results predict a higher level of particle deposition than the experiment had. Despite this the heat transfer of the computational results are higher

than those from the experiment. This is unexpected and will be discussed more in the next section. The graphical comparison of heat transfer coefficients results can be found in Figures 24 and 25 with respect to the air velocity and Reynolds Number, respectively.

The results also have power series trend lines fit to them for possible correlations.

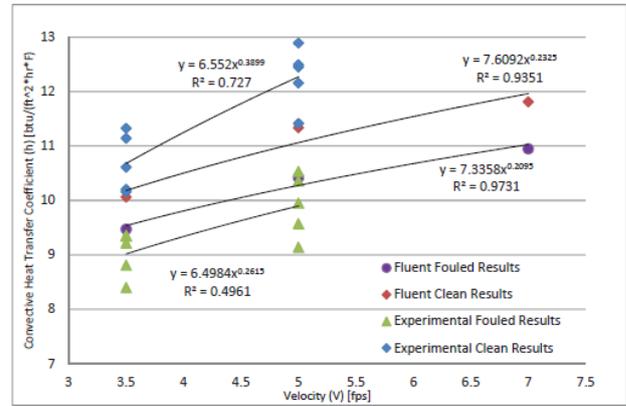


Fig.24. Convection Heat Transfer Coefficients versus Velocity

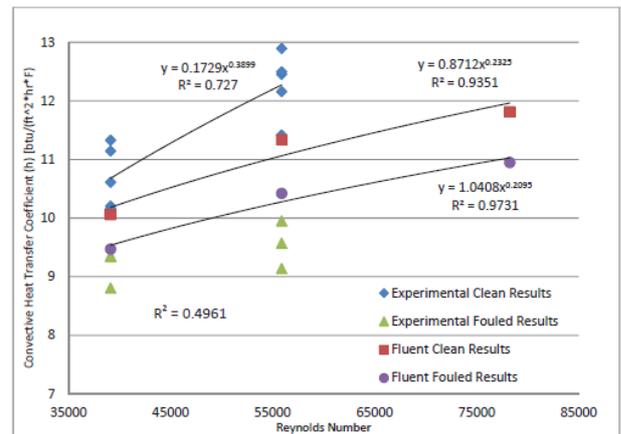


Fig.25. Convection Heat Transfer Coefficients versus Reynolds Number

Table 5 shows the percent difference between computational and experimental results for the clean condenser. It can be seen they are well within an acceptable range for engineering predictive simulation. The fouled condenser heat transfer coefficient was not compared in this manner due to the significant difference in pollen concentration between the experimental at 4.415×10^{-5} lbm/in² concentration and the computational at 7.487×10^{-3} lbm/in² concentration. Instead the percent reduction in heat transfer from clean to fouled for both the experiment and the computational was calculated and is tallied in Table 5 Notice that the percent reduction in the heat transfer coefficient was significantly more in the experimental results even though the particle deposition was less. This is opposite of what one would expect. However, both the experimental and computational results indicate that fouling has a great effect on the coefficient reduction at higher air velocities. That is fouling seems to

impede the boost that air speed gives to the heat transfer coefficient.

Clean Condenser				
Velocity (fps)	Experimental Heat Transfer Coef. (btu/lbm·ft ² ·°F)	Computational Heat Transfer Coef. (btu/lbm·ft ² ·°F)	Percent Difference	
3.5	10.68	10.06	5.9%	
5	12.28	11.33	7.8%	
7	NA	11.81	NA	
Fouled Condenser				
Velocity (fps)	Experimental Heat Transfer Coef. (btu/lbm·ft ² ·°F)	Computational Heat Transfer Coef. (btu/lbm·ft ² ·°F)	Percent Reduction Experimental	Percent Reduction Computational
3.5	9.91	9.54	15.6%	6.6%
5	10.68	10.27	19.3%	6.4%
7	NA	11.02	NA	6.1%

Table 5 Percent Differences in Heat Transfer Coefficients

REFERENCES

[1] Cengel, YA and boles, M.A Thermodynamics: An Engineering Approach 6th ed. Boston: McGraw Hill,2006.

[2] Energy Information Administration (EIA). “Electricity Explained: Electricity in the United States”. 16 july.2011.

[3] Conoco Systems.“Air Cooled Condenser Cleaning Services” 24 March. 2012
http://www.concosystems.com/services/industrial_cleaning/air_cooled_condenser_cleaning

[4] Neville,A.PowerMag. Vol. 156 No. 4 April 2012

[5] ElimelechM.,Etal. Particle Deposition& Aggregation Measurement, Modeling and Simulation Oxford: Butterworth Hienmann Ltd.,1995.

[6] Fluent Inc. “Fluent User’s Guide.” Fluent Version 13,Fluent Inc., 2010.

[7] Fox,Robert W.,AlanT. McDonald,and PhilipJ. Pritchard. Introduction to Fluid Mechanics.6th ed.John Wiley & Sons,Inc.,2004

[8] Stultz,S.C.Steam: Its Generation and Use. 41st ed. Barberton,Ohio: The Babcock & Wilcox Company,2005.