Numerical Flow Visualization Of A Rectangular Shell-and-Tube Thermosyphon Heat Exchanger with Baffled Cross Flow

Mebougna Drabo
Department of Mechanical and Civil Engineereing and Construction Management
Alabama A&M University
mebougna.drabo@aamu.edu

He Tao, Wei Zhong, Jon P. Longtin
Deparmtent of Mechanical Engineering, Stony Brook University
heluis880203@hotmail.com , wei.zhong@stonybrook.edu, jon.longtin@stonybrook.edu

Thomas Butcher, Narinder Tutu, Rebecca Trojanowski
Brookhaven National Labortory-Sustainable Energy Technologies Department
butcher@bnl.gov, tutu@bnl.gov, rtrojanowski@bnl.gov

Abstract

The vast majority of baffled shell-and-tube heat exchangers that have been studied in the past are cylindrical in design. Correlations developed for these designs may not always be applicable for a rectangular geometry since the fluid flow on the shell side can be considerably different than that in a circular geometry. In this paper we present the results of a shell-side flow simulation for a rectangular cross-section shell-and tube heat exchanger that is being designed for cooling and condensing the water vapor in the flue gas from a natural-gas-fired power plant. Numerical solutions to the flow system were used to evaluate alternative design parameters, including the number of baffles, baffle dimensions, orientation of baffles etc. to improve the heat exchanger performance. The results show that a substantial fraction of the tubes immediately underneath the baffles are flow starved. As a result, changing the basic design parameters of the heat exchanger do not necessarily improve the heat exchanger performance: either the heat transfer may not improve significantly, or the pressure drop will become unacceptably high. Thus, to obtain optimal heat transfer performance a number of ideas for redistribution of flow need to be investigated. It should also be noted that even if nearly uniform flow distribution is achieved for all regions, the heat transfer may not be the highest because the average flow velocity (and hence heat transfer coefficient) may not be the highest.

1. INTRODUCTION

1.1 Background and Objective

Baffled shell-and-tube heat exchangers are popular heat exchangers used in a wide range of thermal engineering applications. These devices have been studied extensively in the past, including experimental, numerical and analytical approaches. The vast majority of these devices, however, are cylindrical in design. Far less common is a rectangular geometry.

ISBN 978-1-60643-379-9

Fluid flow on the shell side of shell-and-tube heat exchangers can be very complicated depending upon the particular design geometry. As a result, heat transfer and friction drop correlations available in the literature may not be always applicable if none are to be found for the specific geometry and for the valid range of governing dimensionless numbers in the correlation. Furthermore, the choice of optimal design parameters (number of baffles, dimensions and orientation of baffles, etc.) may not be always obvious. As a result, numerical flow visualization via Computational Fluid Dynamics may provide valuable insight and benefit for design purposes.

In this paper we present the results of flow simulation for a rectangular cross-section shell-and tube heat exchanger that is being designed for cooling and condensing the water vapor in the flue gas from a natural-gas-fired power plant. The tubes in the device are actually two-phase thermosyphons, which provide excellent heat transfer characteristics.

In shell and tube heat exchangers with baffles, the fluid on the shell side must undergo at least a few 180° changes in flow directions. As a result, it is to be expected that many regions of many tubes may be flow starved. This departure for non-uniformity of flow (velocity) distribution is clearly a function of the particular design geometry. The heat transfer and pressure drop performance of the designed heat exchanger are clearly dependent on flow distribution. Thus, to obtain optimal heat transfer performance a large number of ideas (for redistribution of flow) need to be investigated. In order to understand the performance for our DEWCOOL (see below) heat exchanger basic design and to generate ideas for design improvements, we have performed numerical flow visualizations via SOLIDWORKS commercial software.

There are many standard methods for the shell-side design for shell-and-tube heat exchangers. For example, among these are the Bell-Delaware and Kern methods [1]. Clearly, these would be applicable for the many standard shapes of shell-and-tube heat exchangers on which these methods are based upon. For appreciably different design geometries other numerical computational methods must be used to optimize the heat exchanger design. An extensive review of CFD analyses of heat exchangers for design evaluation is provided by Bhutta et al [2]. It documents various models and commercial CFD codes used by various investigators along with comments on comparison of CFD predictions with experimental data where available. A vast majority of such analyses are for circular cross-section shells and very specialized novel designs.

In addition to the CFD models that simulate real objects like tubes in the shell, there are also numerical methods that are based upon a resistance model for the tube bundles. He et al. [3] used such a numerical computational model that is based upon the concept of distributed resistance and a porous medium model (instead of simulating actual tubes) to simulate the flow on the shell side for various baffle configurations. This analysis was also for a circular cross-section shell. Ozden and Tari (4) investigated the shell side design performance of circular cross-section shell-and tube heat exchanger as a function of baffle spacing, baffle cut and shell diameter using a commercial CFD code. They compared their predictions to the Bell-Delaware and Kern methods and found that the CFD predictions for total heat exchange rate in very good agreement with the Bell-Delaware method. They also found large regions of low velocity recirculation zones between baffles.

The objective of this investigation was to evaluate the details of the flow pattern (velocity distribution) on the shell side and the performance (pressure drop, sensible heat removal rate) of the DEWCOOL unit for a few design configurations. Velocity distribution patterns could provide clues to better design configurations, or at least show the reasons why some design modifications do not yield better results. Since SolidWorks flow module does not include two-phase flow and condensation, it is not expected to provide the correct heat removal rate due to condensation heat transfer. Nevertheless, it is expected to show which design (in terms of number of baffles, their locations, or other flow diverting obstructions) is better.

1.2 The Heat Exchanger

A 3-D drawing of the heat exchanger is shown in Figure 1. It contains 45 1.00-inch (O.D.) tubes arranged in a rectangular pattern. The 1 meter long tubes are made of high thermal conductivity (15 W/m.K) polyphenylene sulfide (PPS) polymer with 70% exfoliated graphite. The tubes are manufactured by Technoform Kunststoffprofile GmbH in Germany.

The heat exchanger has several baffles in it that force the flow alternately across the tube bundles, similar to many tube-in-shell heat exchangers. The baffle plates are 1/32" thick sheet metal, and the number and position of the baffles can be adjusted as desired. The baffles have either 1" x 1" square holes or 1" round holes for polymer tubes to pass through. In practice, the baffles are made from two interlaced metal sheets that form 1" x 1" square holes around the tubes. The corners of the square opening provide a small path for flue gas to flow through. These openings can be covered up with thin washers to seal the corners and produce, effectively, 1" round holes that completely seal around the tubes. The washer material can be thin plastic, felt, or aluminum foil. Both configurations were explored in this study. A larger number of baffles increases heat transfer, but the pressure drop increases as well. For this study, it was desirable to keep the pressure drop through the heat exchanger to no more than 1 inch of water (250 Pa).

The tubes actually are configured as thermosyphons, which are two-phase heat exchange devices that provide extremely high heat transfer. The tubes contain a saturated liquid-vapor mixture of water under vacuum. Water is pumped to the top of the tubes to form a thin liquid film on the inside of the tubes. Heat passing into the tube evaporates the liquid film, whose vapor is carried to a separate external condenser where heat is removed to condense the vapor back to liquid and the process repeats. Because of the extremely high heat transfer rates inside the thermosyphon, for the purposes of Computational Fluid Dynamics (CFD) simulation, we shall assume that the inside surface temperature of the tubes is a constant 30 ºC. Thus, only the shell side fluid flow (and heat transfer) and thermal conduction within the tube is simulated.

2. CFD FLOW SIMULATION

SOLIDWORKS Premium 2016 with embedded SOLIDWORKS Flow Simulation module was used for all simulations. The mesh sizes for these computations ranged from 3.5 million to 3.8 million. The total CPU time for the 3 Baffles-Modified run was 16 hours and 21 minutes. For all other cases the CPU time was about 4 hours and 16 minutes. The computations were made on a PC with Intel (R) Core(TM) i7-3770K CPU running at 3.50GHz.

The resulting SolidWorks simulations provide the pressure drop, flow patterns and convective heat transfer through the system. The SOLIDWORKS flow simulation is incapable of calculating phase change heat transfer, however, the results are still of utility both for the pressure drop and to perform relative comparisons of the designs. Once a particular configuration has been down-selected, a more comprehensive (and much more time consuming) analysis will be performed that includes the phase change heat transfer. This can be done, e.g., with FLUENT.

2.1 Inlet and Outlet Boundary Conditions

For these simulations all shell walls are assumed to be real with friction (zero slip velocity) but are adiabatic. Thus, only the heat transfer from the tubes is calculated. This not only saves computational time, but also provides a lower bound for the heat removal rate from the flue gas. The baffle plates are also assumed to be adiabatic but real with friction.

To simulate the flue gas, humid air at a temperature of 45 ºC, a relative humidity of 100%, and a flow rate = 0.218 kg/s is used, since these are the conditions that will be used in experimental validation, and are very close to the composition of the actual flue gas. The temperature and relative humidity conditions are required for the flue gas to condense on the thermosyphon tube walls. This temperature is considerably cooler than the exhaust temperature of a traditional power plant, so some upstream cooling is required before delivering the flue gas to the heat exchanger for condensation.

At the exit face of the computational domain, the pressure is assumed to be atmospheric minus 1 inch (250 Pa) of water, i.e., about 101 kPa.

2.2 Setup

Several versions of the 3-D drawing shown in Figure 1 were setup in SolidWorks for flow simulation: two versions each for 4 Baffle design and 3 Baffle design with 1” x 1” square holes, and one version of 3 Baffle design with 1” round holes. The version labeled “Base Design” (shown in Figure 1) is with 4 baffles and 1” square holes. Drawings of other versions are shown in Figures 2 (4-Baffles-modified), 3 (3-Baffles), and 4 (3-Baffles-modified), respectively. Only the version labeled as 3-Baffles-Round Hole has 1” round holes in the baffle plates.

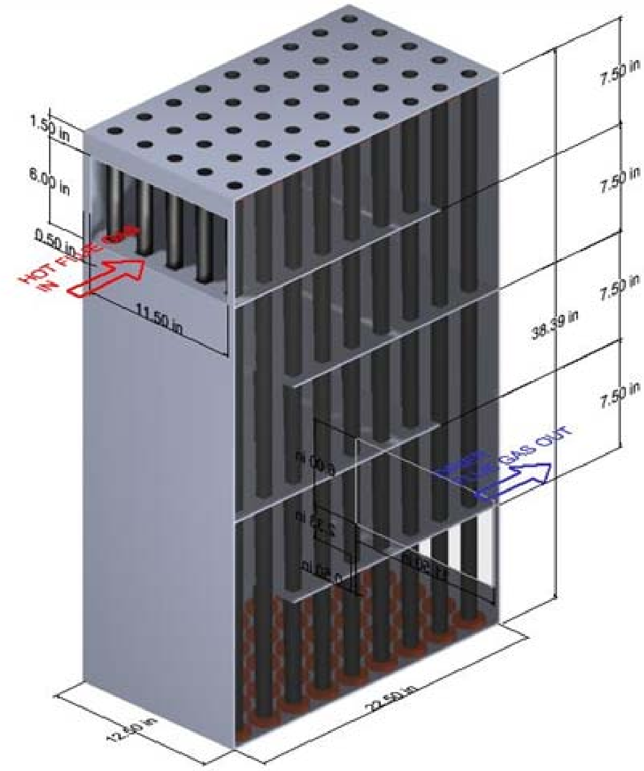

Figure 1. A 3-D rendering of the DEWCOOL shell-and-tube Heat Exchanger for the 4-Baffle Base Design configuration. The front wall of the shell is transparent for visualization of tubes. Exit duct not shown here.

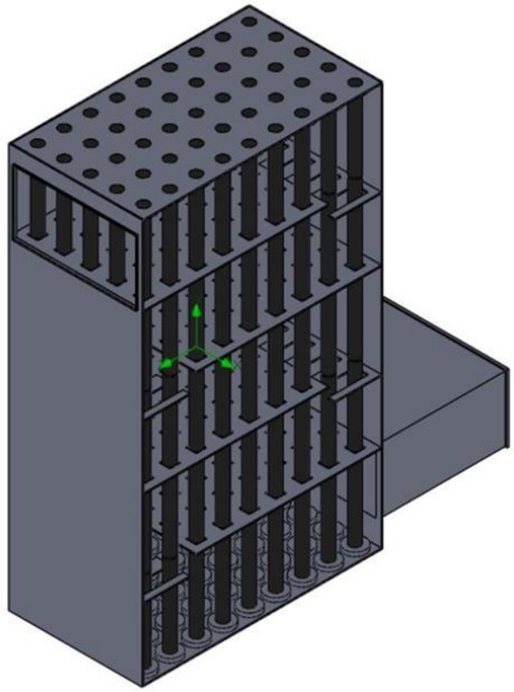

Figure 2. A 3-D rendering of the DEWCOOL shell-and-tube Heat Exchanger for the 4-Baffle - Modified Design configuration

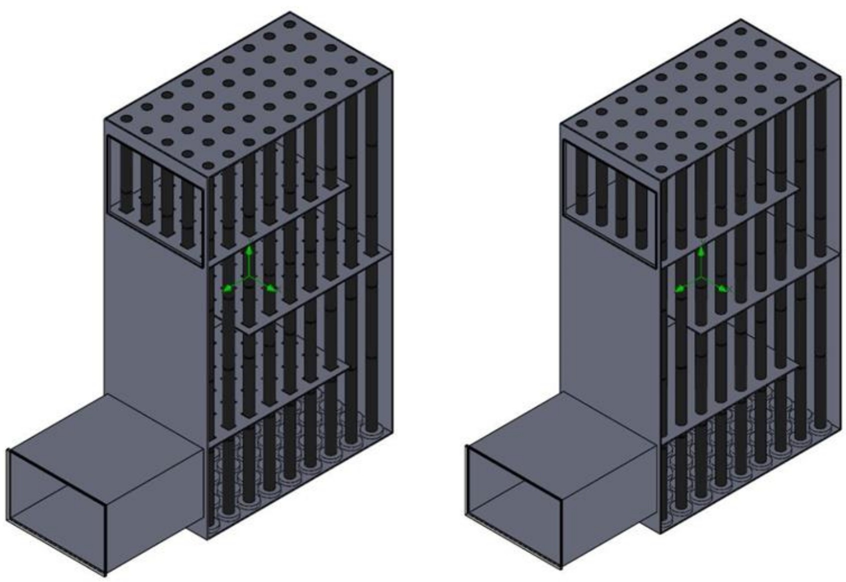

Figure 3. 3-D renderings of the DEWCOOL shell-and-tube Heat Exchanger for the 3-Baffle Design configurations. LEFT: Baffles with Square Holes, RIGHT: Baffles with Round Holes.

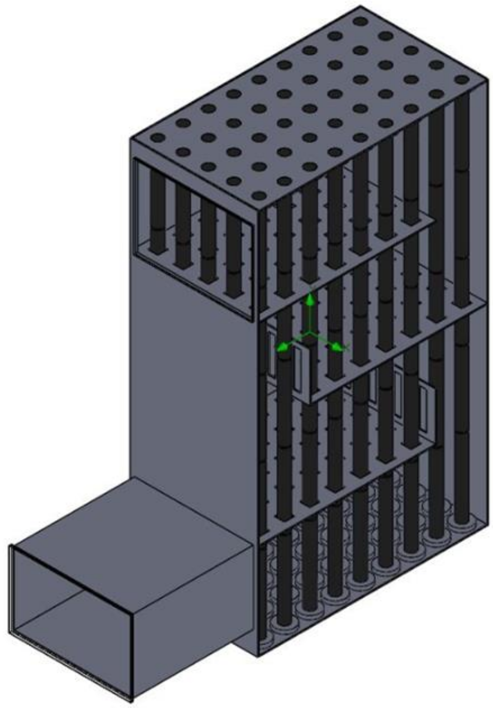

Figure 4. A 3-D rendering of the DEWCOOL shell-and-tube Heat Exchanger for the 3-Baffles - Modified (with square holes) Design configuration.

3. RESULTS AND DISCUSSION

A summary of the results for the pressure drop and overall heat removal rate for all the design options is shown in Table 1.

Table 1: Simulation Summary of Overall Pressure Drop and Heat Removal Rate for Various Design Configurations of the DEWCOOL Heat Exchanger Unit.

Baffle Design	**Pressure Drop (Pa)**	**Heat Removal Rate (W)**	**Comments**
4-Baffles-Base Design	254	4030	Base Design.
4-Baffles - Modified	475	4230	Undesirable. Pressure drop too large with minimal heat transfer increase.
3-Baffles	160	3560	Base case
3-Baffles – Modified	162	3230	Undesirable. Lower heat transfer.
3-Baffles with round Holes (no leakage)	195	3790	Higher heat transfer rate with modest increase in pressure drop.

3.1 4-Baffles Designs

Flow trajectories for the 4-baffle base design are shown in Figures 5 and 6. The color code indicates the total magnitude of the velocity vector. It should also be noted that because of 1" square holes in the baffle plates some of the fluid can be seen crossing the baffle plates directly. As can be clearly seen, the velocity distribution below the baffles is highly non-uniform after the flow undergoes a 180º turn. These figures show that a large fraction of the surface area for most tubes are only minimally participating in heat transfer due to very low flow velocities surrounding the tubes in these regions. This can also be seen from the contour plots of velocity at several horizontal planes below and above the baffle plates shown in Figures 7 through 9.

Based upon these results, an attempt was made to improve the heat transfer rate by trying to deflect some of the airflow upwards as it is making a U-turn. As shown in Figure 2, this was done by adding short baffles in the region just after the flow turns downward for the next section. The intent was that additional baffles would re-direct the flow and produce a jet effect that would force some of the flow to the upper regions of the tube. Flow trajectories for this design are shown in Figures 10 and 11. A comparison with Figures 5 and 6 for the base design shows, however, that there is little improvement in the uniformity of flow. The reason appears to be that the fluid prefers to take the path of least resistance between opposite open ends where the baffles terminate. As can be seen from Table 1, there is only 5% improvement in the heat removal rate, whereas there is a substantially larger (87%) increase in pressure drop. Further exploration of this approach was abandoned. The pressure drop and convective heat transfer for both 4-baffle designs is shown in Table 1.

A web link for the CFD flow trajectories animation for these designs is provided in Reference [5]

The predicted pressure drop for the base case 4-baffle design was 254 Pa. While this technically satisfies the pressure drop requirement of 250 Pa, if the actual device as built has a slightly higher pressure drop in practice, this would present a problem. A three-baffle design was thus explored as well, with the intent of reducing the pressure drop. Also, any additional baffle modifications to increase the heat transfer would likely not increase the pressure drop above the design limit.

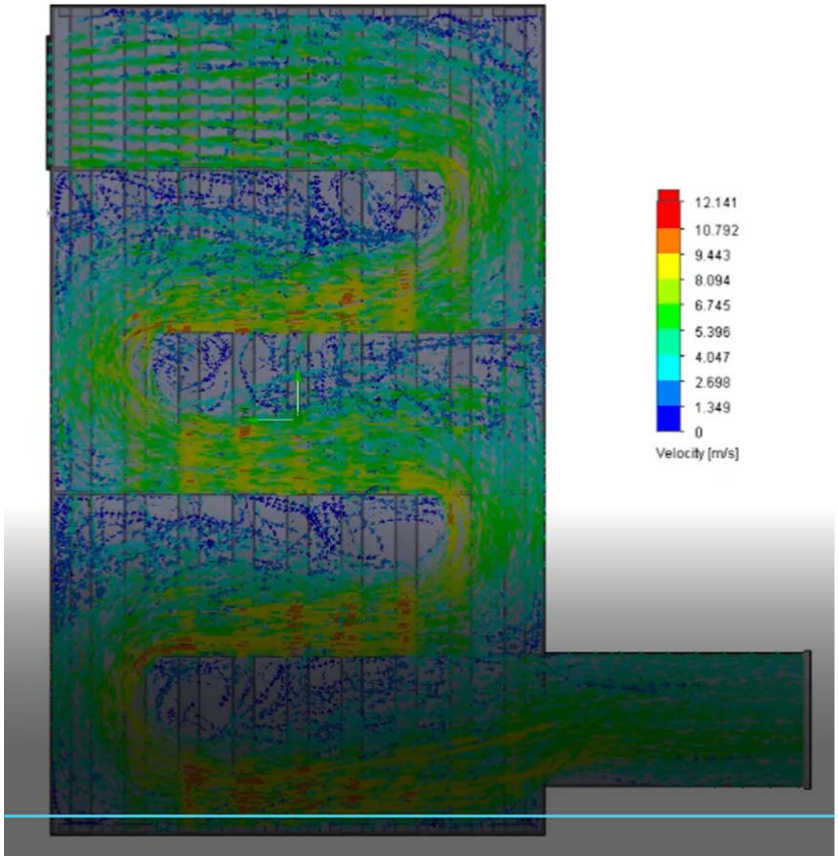

Figure 5. Flow trajectories for the 4-Baffle Base Design.

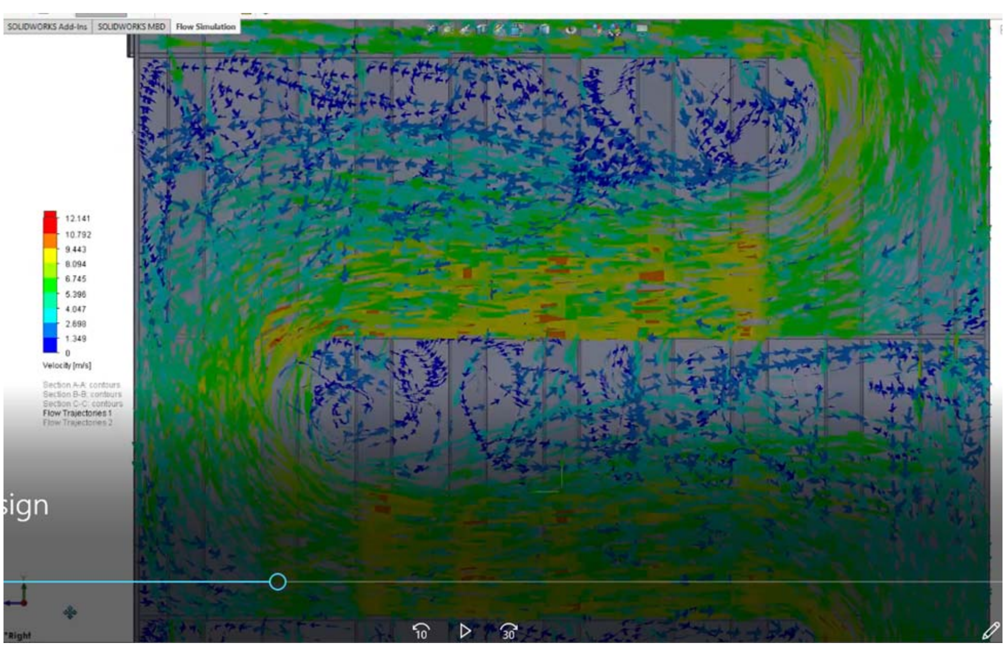

Figure 6. A Close-Up of the Flow trajectories for the 4-Baffle Base Design

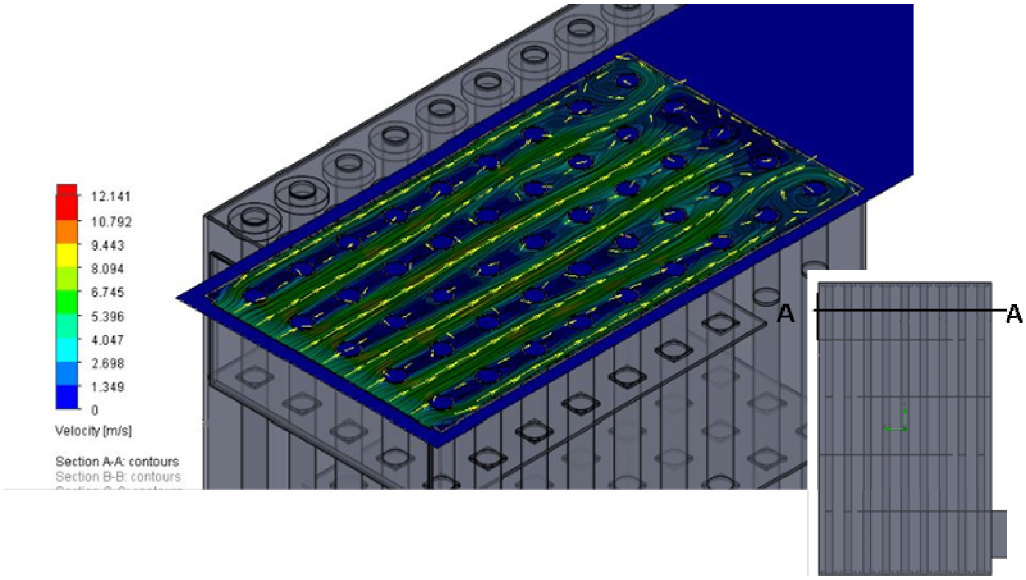

Figure 7. A contour plot of velocity vectors at section A-A above the top baffle adjacent to Inlet.

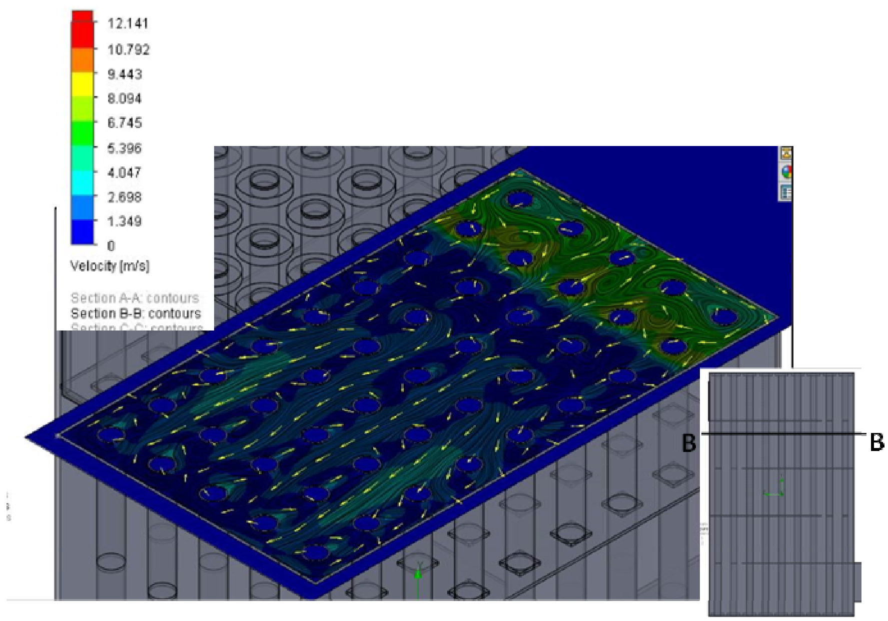

Figure 8. A contour plot of velocity vectors at section B-B below the top baffle.

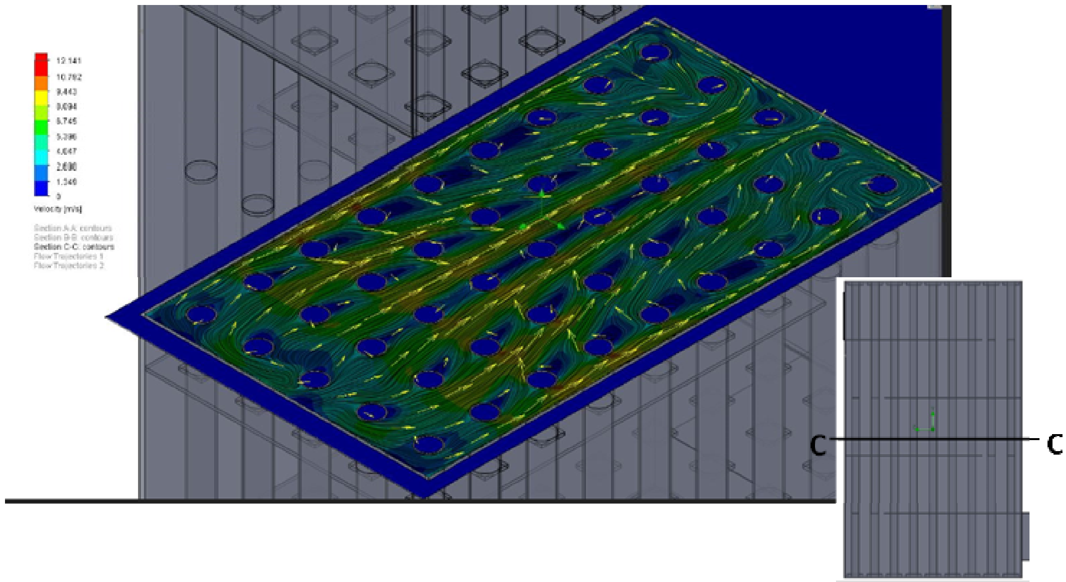

Figure 9. A contour plot of velocity vectors at section C-C above the second baffle from Bottom

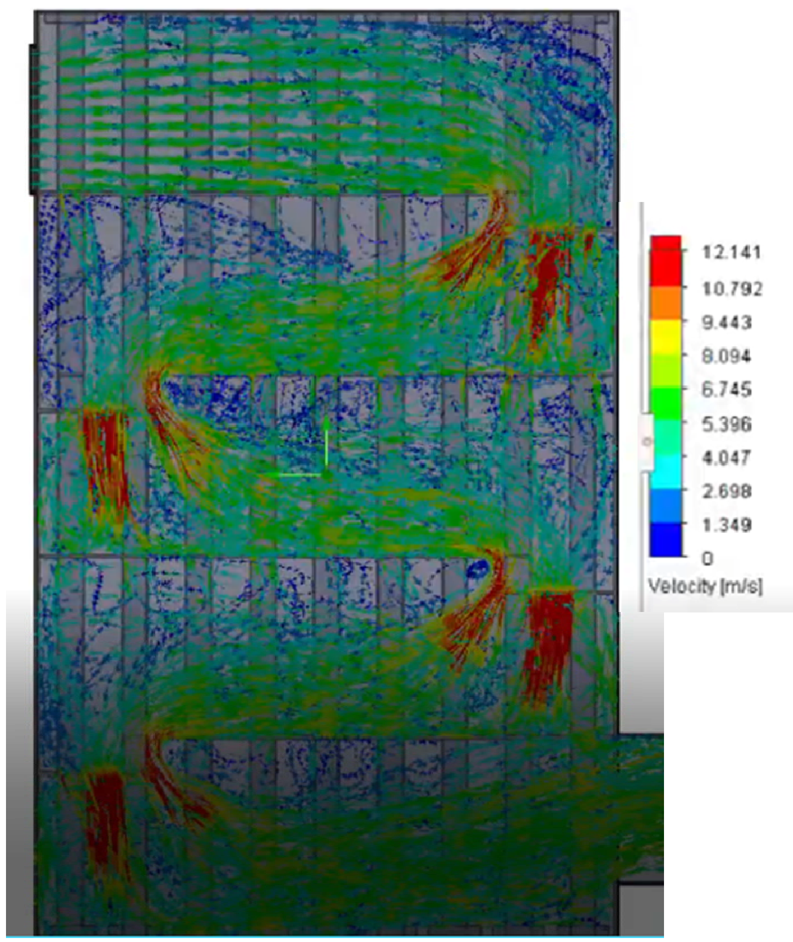

Figure 10. Flow trajectories for the 4-Baffles - Modified Design.

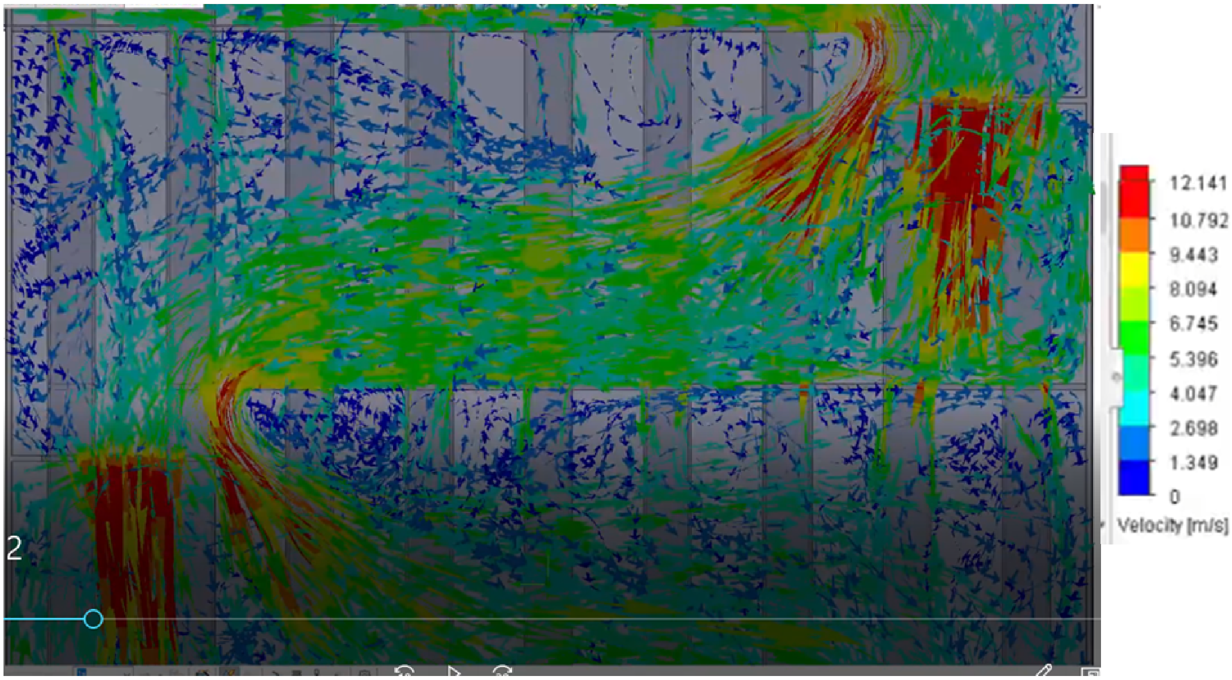

Figure 11. A Close-up of Flow trajectories for the 4-Baffles - Modified Design.

3.2 3-Baffles Designs.

With only three baffles, the pressure drop will be reduced. Furthermore, if the pressure drop is reduced enough, it may be possible to modify the design to yield a better heat removal rate with only a modest increase in pressure drop.

The results are shown in Table 1, the predicted pressure is only 160 Pa for the 3-Baffles design with 1” square holes. However, the heat removal rate drops to 3,560 W as compared to 4030 W for the 4-Baffles Base Design. Flow trajectories for the 3-Baffles design are shown in Figure 12. Again, as for the 4-Baffles case, the flow distribution is highly non-uniform.

In an attempt to improve the flow distribution, we modified the bottom two baffles by including a small slotted vertical baffle (see Figure 4) at the end of each baffle. As can be seen from Figure 13, the velocity distribution is more uniform (as compared to Figure 12), but the velocities seem lower. However, as seen from Table 1, while the predicted pressure drop is about the same, the heat removal rate is about 9% lower than the 3-Baffles base (unmodified) design. This clearly shows the difficulty in improving the design in order to get better heat transfer rate.

With square holes in the baffle plates, fluid leaks from top to down across the baffle plates. This is likely to result in slightly higher pressure just below the baffles as compared to the case if baffles had 1” round holes. Therefore, we suspected that this may be another contributing reason for the lower flow velocities below the baffles. In order to see improvement, if any, by eliminating this cause, we next simulated the 3-Baffles design with 1” round holes. As seen in Table-1, the heat transfer rate improved by 6.6%. However, this comes at a cost of 22%

increase in pressure drop. A comparison of flow trajectories for the 3_Baffles design with square and round holes is shown in Figure 14. As can be seen from this figure the flow distribution is a little better and flow velocities are a little higher for the design with round baffle holes. A link for the CFD animation is given in Reference [5].

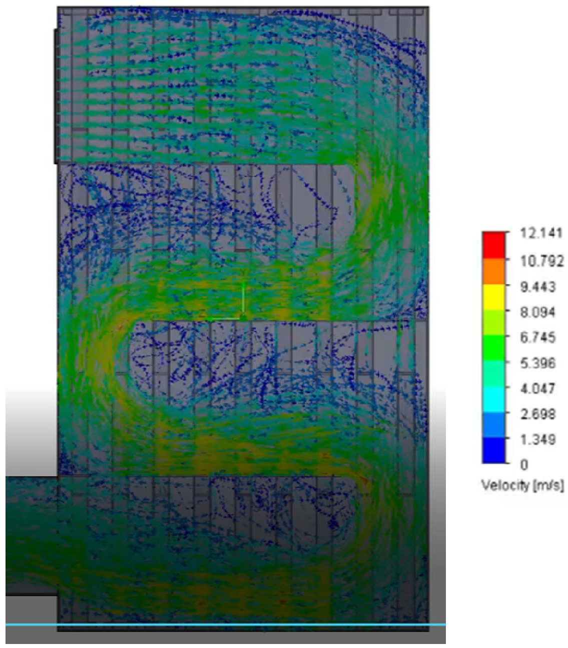

Figure 12. Flow trajectories for the 3-Baffles Design with 1” square holes in the baffle plates.

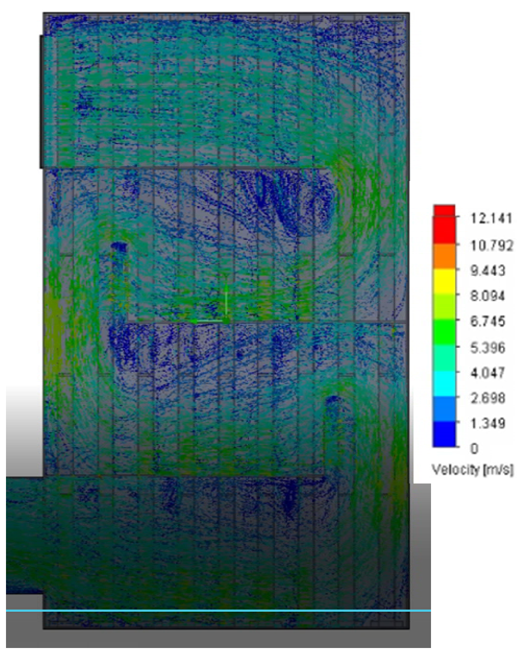

Figure 13. Flow trajectories for the 3-Baffles - Modified Design with 1” square holes in the baffle plates.

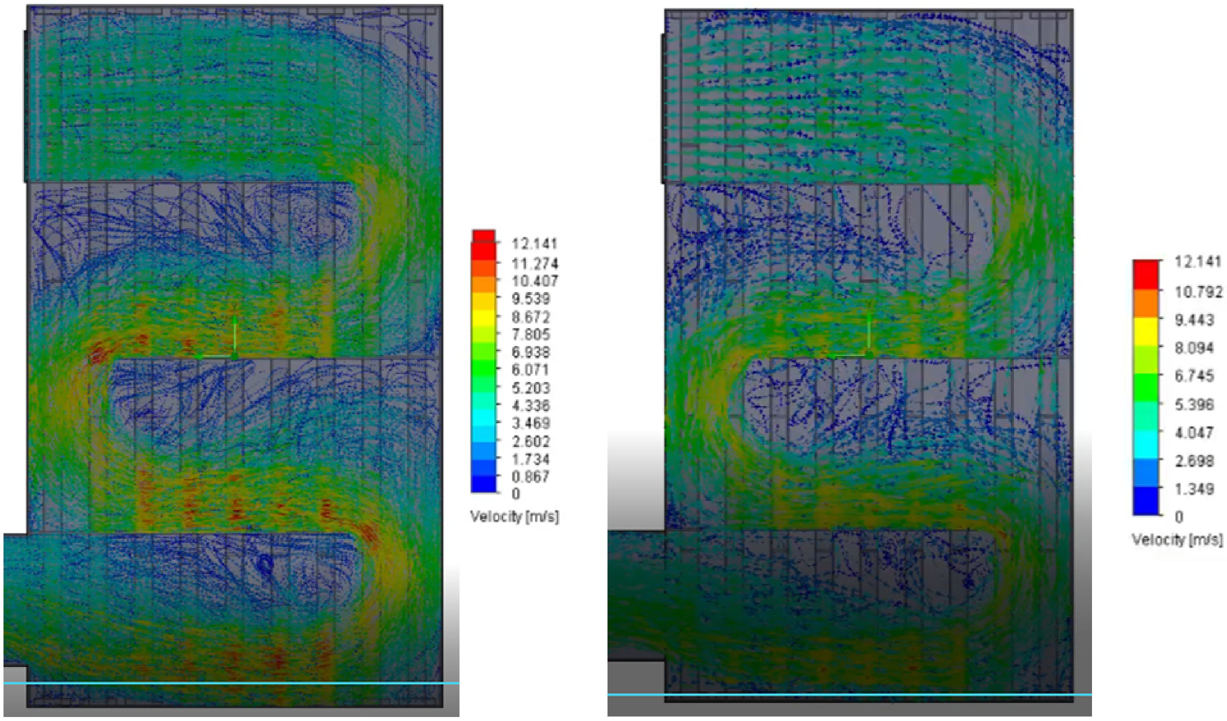

Figure 14. A comparison of the Flow trajectories for the 3-Baffles Designs with 1” Round Holes (left figure) and 1” x1” square holes (right figure) in the baffle plates.

3.3 Comparison to Literature Correlations

Interestingly, the CFD analysis showed a markedly lower predicted pressure drop compared to two popular configurations from the literature. Correlations for the literature are nearly always for round shell-and-tube heat exchangers. To adopt these correlations to the rectangular geometry herein, the equivalent hydraulic diameter, and fraction of exposed baffles are used.

As shown in Table 2, for both the 4-baffle and 3-baffle cases, the estimate pressure drop is only 60 to 80% that predicted from the correlations.

Table – 2. Comparison of CFD Predictions and Literature Correlations for the Overall Pressure Drop for Two Design Configurations of the DEWCOOL Heat Exchanger Unit.

	CFD	**CORRELATIONS**	
	ΔP (Pa) **CFD**	ΔP (Pa) [6]	ΔP (Pa) [1]
4-Baffles-original	254	315	415
3-Baffles-original	160	183	236

The discrepancy in the results is reasonable, given the approximations made in adopting the round correlations to a rectangular geometry.

4. CONCLUDING REMARKS

A CFD simulation of a rectangular baffled, cross-flow heat exchanger has been performed to simulate sensible heat transfer and pressure drop through the system. The purpose of the heat exchanger is to extract water from the combustion flue gas of a boiler or similar combustion device. The simulations were made using SolidWorks with embedded SOLIDWORKS Flow Simulation module. Both a 3-baffle and 4-baffle design were explored, with the tradeoffs between heat transfer and pressure explored. CFD simulations show that the use of baffles leads to highly non-uniform velocity distribution below the baffles. As a result, significant portions of many tubes experience very low heat transfer rates because of local flow starvation in these regions. The simulations provide insight for design modification purposes. Two baffle modifications were attempted to improve the heat transfer, but the results were inconclusive. Further opportunities for study in this regard include experimental validation of test cases, which are in progress, and the development of correlations for pressure drop and heat transfer for rectangular geometries based on both modeling and empirical measurements.

5. REFERENCES

[1] KJ Bell, "Delaware method for shell side design," in: S. Kakac, A.E. Bergles, F. Mayinger, editors. Heat exchanger: thermal-hydraulic fundamentals and design. New York: Hemisphere; 1981, pp. 581 -618.

[2] Muhammad Mahmood Aslam Bhutta*, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, and Sarfaraz Khan, "CFD applications in various heat exchangers design: A review," Applied Thermal Engineering 32, pp. 1-12, (2012).

[3] Y. L. He, W. Q. Tao, B. Deng, X. Li, and Y. Wu, "Numerical Simulation and Experimental Study of Flow and Heat Transfer Characteristics of Shell Side Fluid in Shell-and-Tube Heat Exchangers," Proceedings of Fifth International Conference on Enhanced, Compact and Ultra-Compact Heat Exchangers: Science, Engineering and Technology, Eds. R. K. Shah et al, Engineering Conferences International, Hoboken, NJ, USA, September 2005.

[4] Ender Özden and Ilker Tari, "Shell side CFD Analysis of a Small Shell-and-tube Heat Exchanger," Energy Conversion and Management, Vol. 51, No. 5, pp. 1004-1014 (2010).

[5] M. Drabo, "Video Animation of Flow Through DEWCOOL Heat Exchanger," Private Communication,WebURL: https://1drv.ms/f/s!AhL6U6TalgxmkyACg6ft0RVkkgRq 2018.

[6] E.S. Gaddis and V. Gnielinski, "Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles," Chemical Engineering and Processing, Vol 36, No. 2, pp.149-159 (1997).

Biographies

Mebougna Drabo is an associate professor of mechanical engineering at Alabama A&M University. His research interests include advanced flash atomization technology, burner/Atomizer Integration, Material Characterization, multi fuel optimization, biofuels, Computational Fluid Dynamics Analysis, heat transfer and advanced combustion IC engine concepts. Dr. Drabo may be reached at mebougna.drabo@aamu.edu

Tao He is a Ph.D candidate in Mechanical Engineering at Stony Brook University. He received his B.S degree in Harbin Engineering University, China in 2010. His current research focuses on thermosyphons in cooling applications. His interests include advanced heat transfer, renewable and clean energy, and computational fluid dynamics and heat transfer. Mr. He may be reached at heluis880203@hotmail.com

Wei Zhong is a Ph.D student in Mechanical Engineering at Stony Brook University. He received his B.Eng. degree from Thermal Energy and Power Engineering, at the Beijing Institute of Technology in 2013. His current research focuses on thermosyphons and phase change heat transfer. Mr. Zhong may be reached at wei.zhong@stonybrook.edu

Jon P. Longtin joined the Mechanical Engineering Faculty at Stony Brook University in 1996. He came to Stony Brook after receiving his Ph.D. degree in 1995 from U.C. Berkeley, followed by a one-year postdoc at the Tokyo Institute of Technology in Japan. His research interests include energy conservation, innovative energy transfer and storage, and energy monitoring and diagnostics, as well as laser materials processing, particularly with ultrafast lasers and the development of sensors for harsh environments. His research has been funded by NSF, DOE, DOD, NASA, NYSERDA, and a variety of industrial sources. He is the author of over 130 technical publications and holds 10 issued and pending patents. He has received the Presidential Early Career Award for Scientists and Engineers, two Excellence in Teaching Awards, and an R&D 100 award. He is a licensed Professional Engineer in New York State and serves as a technical advisor to a variety of companies and non-profit organizations. Dr. Longtin may be reached at jon.longtin@stonybrook.edu

THOMAS BUTCHER is the deputy chair of the Sustainable Energy Technologies Department and leader of the Energy Conversion Group. Work within this group includes liquid fuel combustion and oil-fired heating system efficiency, biofuels applications, polymer composite heat exchangers, solar thermal systems, micro-combined heat and power concepts, absorption heat pumps, wood boiler emissions and efficiency, and advanced materials for geothermal energy applications. Dr. Butcher may be reached at butcher@bnl.gov

NARINDER K. TUTU is currently a guest scientist at the Brookhaven National Laboratory (BNL), who retired from the BNL after about 27 years of research. His areas of interest include: a) instrumentation in the fields of fluid dynamics, multiphase flows, and heat transfer; b) two-phase flow and transient heat transfer in porous media; c) safety studies of severe nuclear reactor accident scenarios; and, d) energy conservation in buildings. Dr. Tutu can be reached at tutu@bnl.gov

REBECCA TROJANOWSKI received her BS in Chemical Engineering and M.S. in Mechanical Engineering from Worcester Polytechnic Institute and is now currently working towards her EngScD at Columbia University in the Earth and Environmental Engineering Department. She is an associate staff engineer in the Energy Conversion Group at Brookhaven National Laboratory. Her interests focus primarily on technical solutions to advance building energy systems, fossil fuel reduction, and emissions. Some of her research and development areas include advanced HVAC concepts, biofuels, solid fuels, air pollution, and combustion and system concepts. Ms. Trojanowski may be reached at rtrojanowski@bnl.gov