EVALUATING PERFORMANCE OF AIR COOLED HEAT EXCHANGERS IN LNG PLANTS

Vibhor Mehrotra, Jon Berkoe
Research and Development, Bechtel National Inc.

David Messersmith
Petroleum and Chemical, Bechtel Corporation

3000 Post Oak Boulevard
Houston, TX – 77056

Prepared for Presentation at the AIChE Spring National Meeting 2003, New Orleans, LNG Equipment Design

April 2003

Unpublished

Abstract

In recent years, rapid expansion of LNG train capacity from 1.5 mtpa to more than 5 mtpa have resulted in reduced supply of fresh air to air cooled heat exchangers. To compound the problem, plot space is not increasing at the same rate as plant capacity and in some cases double bank of air-cooled heat exchangers have to be utilized to fit large plants in a relatively small plot space. Modeling of double bank of air-cooled heat exchangers with considerations towards wind speed and direction is discussed in this paper. Addition of horizontal or vertical skirt and its impact on air recirculation temperature is also analyzed. By using CFD modeling, it has been observed that the most desired wind direction for optimum performance of air coolers depends upon a number of factors such as wind speed, height of the cooler fans above grade and number of bays along the length and width. Addition of 10ft horizontal skirt reduces recirculation temperature in both parallel as well as crosswind direction. Finally other equipments surrounding the air-coolers also have a major impact on the cooler performance. This paper discusses the advancement in air-cooled heat exchanger model as applied to large LNG facilities to accurately evaluate the impact of ambient conditions and hot air recirculation on plant production.

1. Introduction

The international Liquefied Natural Gas (LNG) trade went through a rapid expansion in the second half of the 1990's. More than 30 million tons per annum of new LNG capacity from grass-roots plants came on stream in this period. Liquefaction plants have always been designed along the "train" concept. Depending on gas reserves and markets, some new plants start up with a single train, and most eventually have several parallel process trains. Early on, small-size LNG trains matched smaller market demands, and fit available equipment capabilities throughout the LNG value chain (which also includes LNG carriers, and storage facilities). LNG train capacities have been increasing steadily (Figure 1). Single train size has increased from the 0.5 – 1 mtpa range in the early days of LNG, to 1 – 1.5 mtpa during the 1970's and 1980's. Typical train size has then quickly grown from 2 mtpa in 1990 to 3 – 4.5 mtpa today, and trains of more than 5 MTPA capacities are being considered by several projects [1].

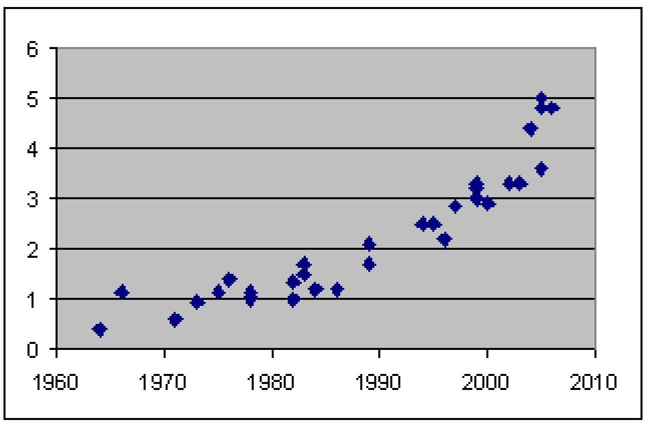

Figure 1. LNG Train size trends in MTPA

The trend of increasing train size adds importance to optimal plot arrangement and equipment design margins. Plot space benefits from the increasing train size; although it is proportional to train capacity, it is not a 1:1 ratio. Experience has shown that plot size increases to the 0.81 powers with train size, while the heat output relationship to plant capacity fits a more linear trend (Figure 2). More efficient plants are needed to reduce wasted heat that is released to the environment.

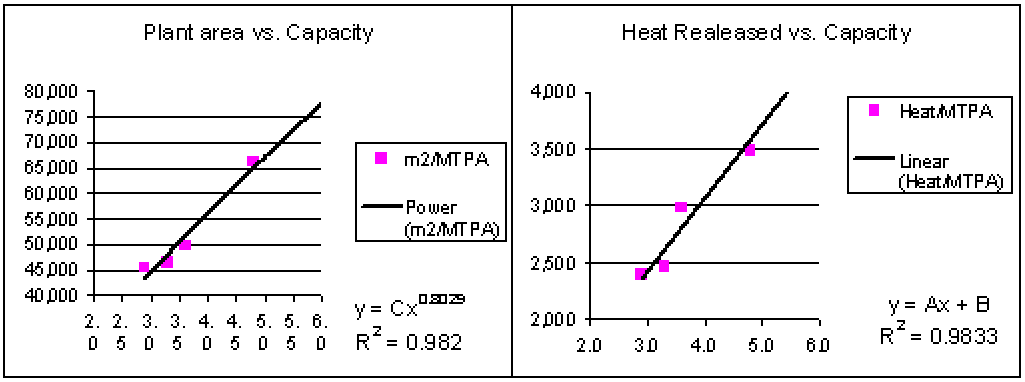

Figure 2. Plant area and heat release versus capacity [2]

What is significant about this information? With plant size, the disposition of the waste heat becomes more important. Not only will the economics of the facility design be driven by normal equipment costs, operating costs but also by the optimization of design margin with the overall facility arrangement and capacity requirements. Even with state of the art equipment and thermally efficient designs such as combined cycle power and process integration, the heat released from a large multi-train facility is significant and can affect the facility's ability to produce at relative design capacities for all potential ambient conditions.

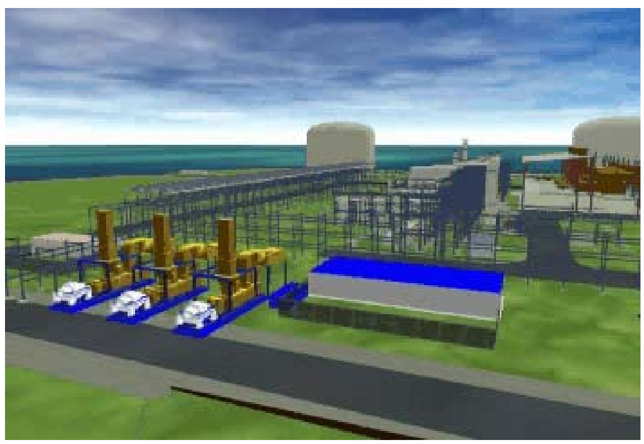

Figure 3. Visual rendering of a LNG plant

In order to ensure plant performance and lower costs, it is necessary to predict how air coolered heat exchangers respond not only to changes in ambient conditions but also how they affect those ambient conditions near the facility.

Computational Fluid Dynamics (CFD) provides a tool to meet that need. It is through the use of this tool that models can be built for extended facilities (future expansion) and to evaluate the effect that these will have on the area in general as well as on each other. This modeling has been done on a very large scale to look at the overall siting requirements such as orientation and spacing, and then also on a very detailed scale to look at the air velocity profiles around individual equipment. This is useful when evaluating the impact of siting on process performance and developing design margins for equipment.

2. Air-Cooled Heat Exchangers Overview

Air-cooled heat exchangers utilize large axial flow fans to blow air over finned tubes, thus removing heat and thereby condensing the process gas. Most manufacturers do design and testing of a unit based upon a single bay set-up. The performance guarantee of each unit is based on availability of sufficient air at design temperatures at the inlet face of the condenser unit. The total number of units required for given duty, space considerations and equipment layout plan governs actual construction of bays in the field. However, the performance of each bay may be different compared to the single bay design, due to variations in airflow distribution and hot air recirculation. The goal of this study is to evaluate the extent of mal-distribution when hundreds of bays are put next to each other in the field. A comparison of the effect of manufacturer suggested total height versus original design height of the condenser bays was also made. Since the flow under air cooler was of importance, the model consisted of only the air coolers. Rest of the equipment in the plant was omitted for simplicity for most of this analysis. However, the last section discusses the overall implications of air cooler performance due to air recirculation on LNG plant production.

The modeling evaluated the alternative of orienting the rows of condensers so they were either parallel or across to the prevailing wind direction. The relative efficiency of these alternative cases has important implications on the plant design philosophy wherever such a facility might be built.

3. Model Description

Computational fluid dynamics (CFD) is the method of solving the governing mass, momentum, scalar and energy equations on a computational grid representing the domain in question. CFD allows for the construction of realistic full-scale computer models, which simulate the airflow patterns inside the structure and all modes of heat transfer. The CFD model accounts for the actual physical geometry of the equipment and applies fundamental physical principles to compute the temperatures, velocities, and pressures. Beginning with the design sketch for the facility, and based on input provided for boundary conditions, this evaluation was conducted using CFD modeling based on a commercial software program with the Fluent 5.5 solver.

3.1 Computational Geometry and Boundary Conditions

The inlet to the domain was specified as velocity inlet, with a parabolic wind profile describing the boundary layer. Inlet to the fans was specified as a fan boundary with the fan curve defining a velocity – pressure drop relation. The equations for each part of the study are detailed in the respective sections. The exhaust from the fans was an interior face, with flow rate calculated by the model itself. The tube bundle boundary was specified as a radiator boundary with manufacturer specified pressure drop equation and a high heat transfer coefficient between the tube surface and air.

A full-scale computer model of double bank air cooler was created. Mass flow boundary was used for the exhaust from the fans and an exhaust fan boundary was used on the inlet to the fans. The pressure drop across the inlet was specified as a correlation between pressure and velocity derived from the fan curve. The study was conducted with changes in the model geometry and pressure definition on inlet boundary face, and is discussed below.

3.1.1 Fan Curves and Pressure Drop due to Tube Bundle

Fans provide the driving force for the air to travel through the tube bundle and flow passage through the system. Therefore the pressure balance has to account for all of the contributions. The fan curve from the vendor provides the pressure equation for the fan and the pressure loss due to the tube bundle is calculated from the manufacturer provided equation. The static pressure is obtained by subtracting the fan internal pressure loss value. These static pressure values were further used to compute the polynomial expression for ΔP in terms of velocity as described below. The pressure drop through the tube bundle is given by the following equation [3]:

$$\Delta P = C2\ (\rho\ c)\ (V/100)^{X} \qquad (1)$$

Where C and X are constants

$\rho\ c = 1$ for standard conditions

and V = velocity on the face of tube bundle in ft/min

In the initial part of the study, the inlet faces were rectangular. For a given air velocity through the fan, the static pressure was obtained from the fan curve. From this value, the pressure drop due to the tube bundle was subtracted. The resultant pressure drop value was regressed as a polynomial against the velocity to obtain a correlation. It was observed during the course of CFD runs that when the velocity assumed negative values on the inlet face, owing to the flow direction and the geometry, a reverse flow was obtained. To avoid this problem, the tube bundle pressure drop equation was regressed for negative velocity values. Then the two polynomials were used as piecewise functions (tube bundle polynomial for negative velocity regime and fan curve polynomial for positive velocity regime).

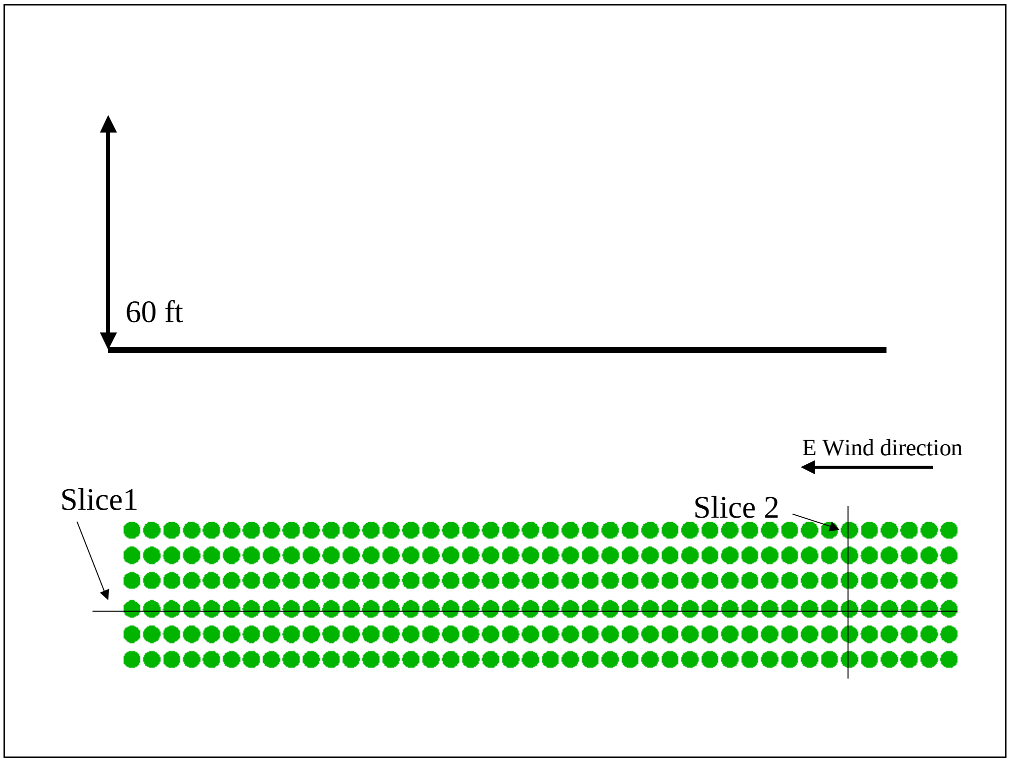

Figure 4: Rectangular inlet and circular exhaust

3.1.2 Circular Inlets

The above-mentioned modification did not result in to any significant change in the results. Especially, the crosswind cases still showed very large reversed flow regions with a negative total airflow into the cooler, which does not appear realistic. The geometry was modified such that each fan has its own individual inlet face, shown in Figure 5. Pressure was specified for this face by adjusting the polynomial equation to a parabola, such that reverse flow into the domain was prevented from the pressure outlet. Additionally, the face representing tube bundle was modeled as a radiator boundary with no heat loss. This prevented flow back in to the domain from the pressure inlet.

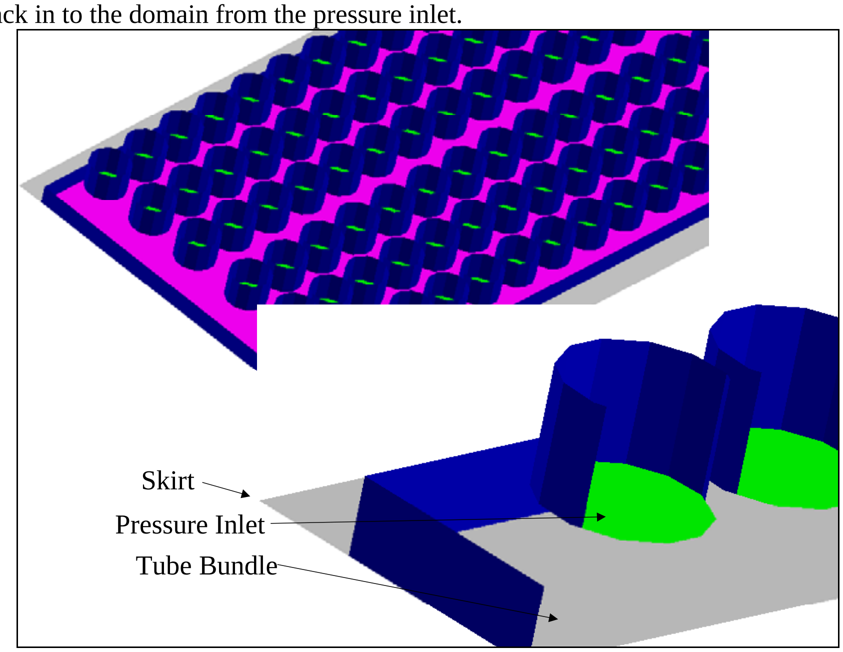

Figure 5: Circular inlets

3.1.3 Fan model and exact geometry

Replicating the exact fan geometry and creating a mesh inside the fan volume, further improvement in the model was achieved. Instead of specifying a flow rate out of the exhaust, the suction of the fan (specified using the fan curve) determined the mass flow from the exhaust. This established a more realistic flow of air around and under the air coolers. The exhaust flow dynamics was linked to the intake, resulting in a well-defined recirculation pattern (Figure 6). As in the previous case, the tube bundle was modeled as a radiator with a corrected loss coefficient (calculated from tube bundle pressure drop equation) and a high heat transfer coefficient to attain the desired exhaust temperature.

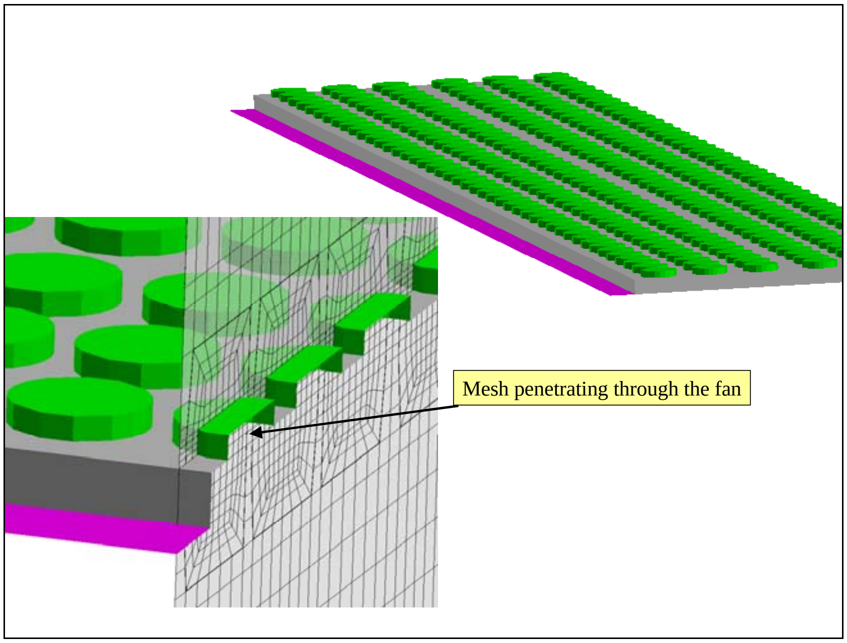

Figure 6: Fan model with mesh through the fan body

4. Results

4.1 Comparison of vertical and horizontal skirts

As seen in Figure 7, the vertical skirt cause downward flow outside the skirt and gives a wider exhaust plume. Horizontal skirt offsets this flow away from the air cooler by a distance as wide as the skirt, giving a narrow plume. This causes more lift and less possibility of recirculating back in to the inlet. Both cases predict the same amount of fan throughput but the horizontal skirt results in a lower inlet recirculation temperature into the coolers.

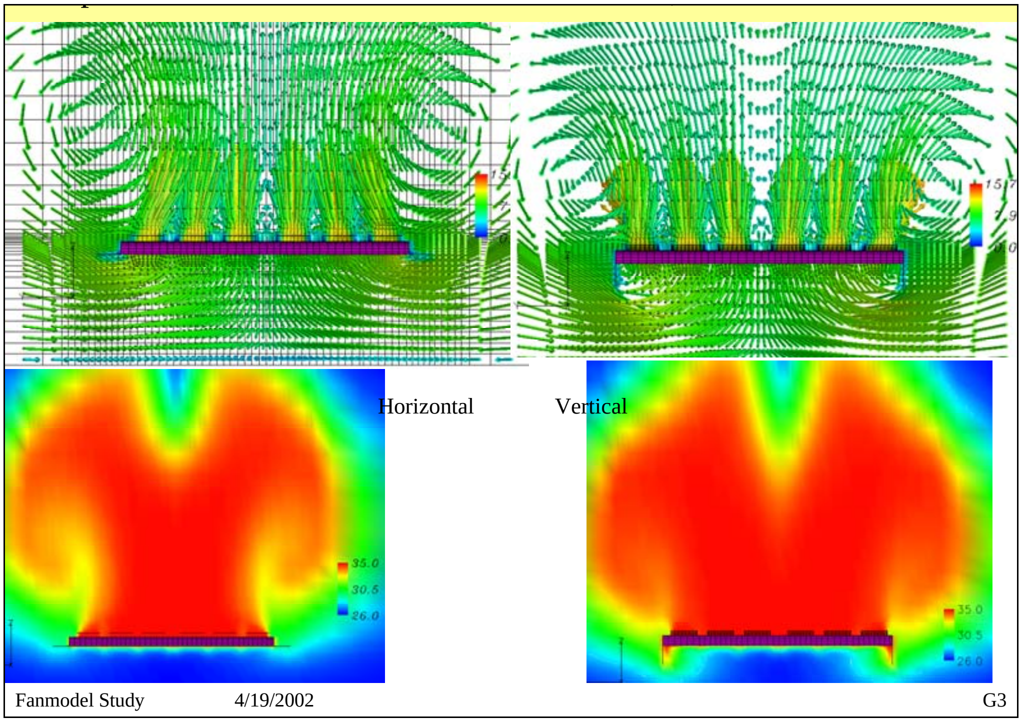

Figure 7: Comparison of 10 ft. horizontal & vertical skirts, parallel wind

In the crosswind wind case, the vertical skirt offers larger resistance to airflow. This causes acceleration of flow below the first 1-2 rows of fans (Figure 8). As seen in the vector plot (Figure 8), the downward pull of the airflow upstream of vertical skirt renders the first row of fans nearly dysfunctional. Moreover, since the crosswind approaches the air coolers from one side, therefore, the entire flow has to enter the air cooler from that side resulting in a higher average and peak velocity below the skirts.

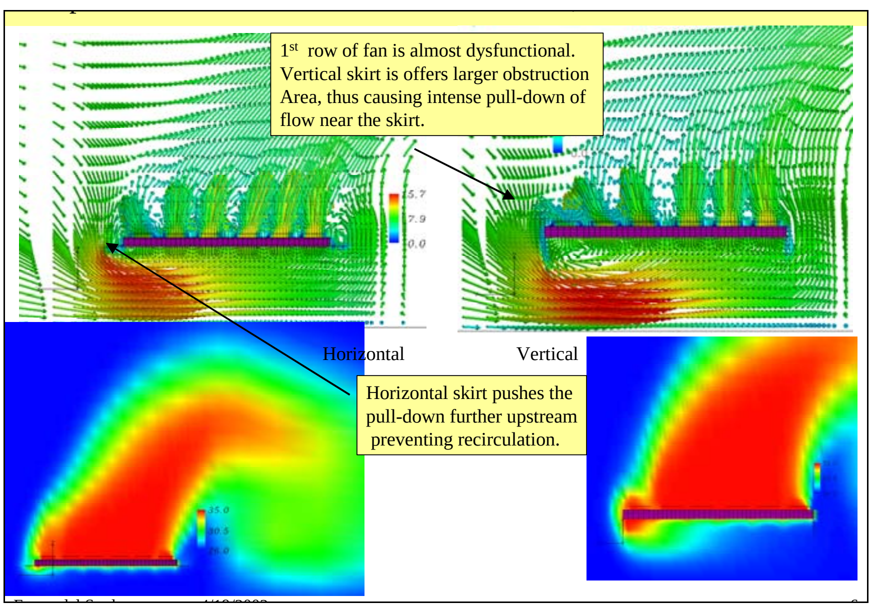

Figure 8: Comparison of 10ft horizontal and vertical skirts, crosswind case.

4.2 Comparison of 5, 10 & 15ft horizontal skirt cases

Figure 9 shows a comparison of 5, 10 & 15ft horizontal skirt in the crosswind case. As can be seen in the figure, the widest skirt (15 ft) gives least amount of recirculation due to the downward flow being pushed away from the air cooler, thus preventing it from recirculating back into the inlet. The first rows of fans appear to be being severely affected in exhausting flow for 5ft skirt case. The fan performance gradually improves as the skirt width is increased to 15ft. Increase in flow and decrease in the inlet temperature is also noticed as the skirt width increases.

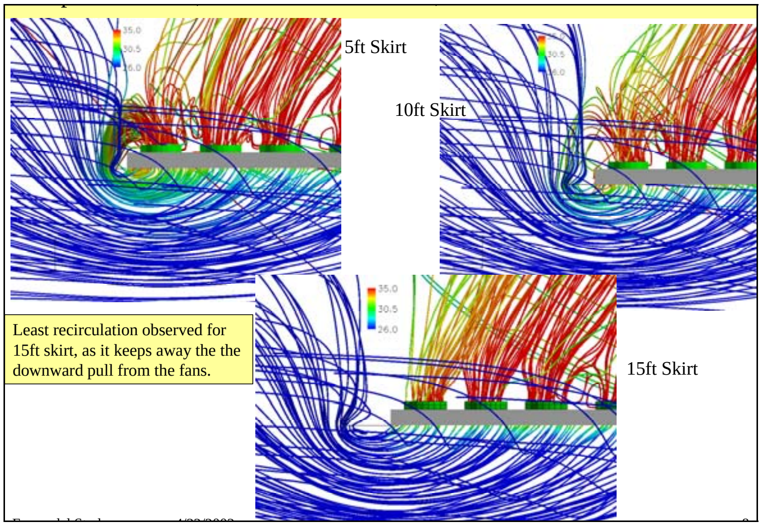

Figure 9: Comparison of 5, 10 & 15ft skirts, crosswind case

4.3 Comparison of no skirt and vertical skirt cases

It is interesting to note that vertical skirts are disadvantageous to the air coolers in terms of flow as well as recirculation for both cross and parallel wind cases (Figure 10). The major drawback of the vertical skirt is the obstruction to free flow that causes a downward pull near the skirt. This pull is not only responsible for recirculation but also disrupts the exhaust flow pattern of the fan row closest to the skirt, thus reducing its throughput capacity. The no-skirt case has higher flow rates and lower inlet temperatures for both parallel and crosswind cases, with the difference being more pronounced for the parallel wind case.

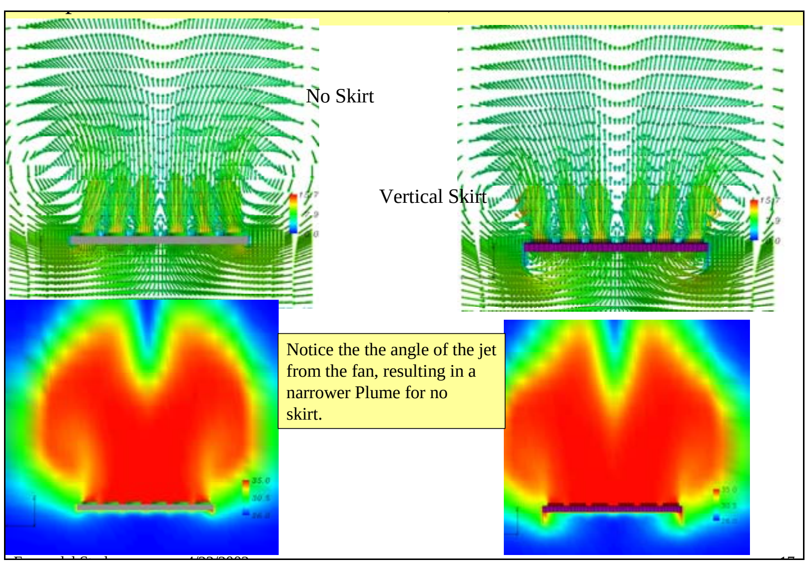

Figure 10: Comparison of no-skirt and vertical skirt, parallel wind case.

5. Conclusion

Standalone double bank of air coolers are studied using CFD modeling. It is observed that the fluid dynamics of the plume emanating from the cooler changes significantly with the type of skirts (vertical/horzontal) and wind direction (parallel/crosswind). It is demonstrated that the horizontal skirt improves air cooler performance under all wind speed and direction. As the skirt width is increased above 10ft, it results in diminishing return on performance improvement with cost increase. From the results of the analysis it can also be concluded that a vertical skirt causes more harm than performance improvement to air coolers by accelerating the flow and entraining hot air from the nearest fan. It also increases flow resistance due to area reduction that may result in reduced airflow to the air coolers. A horizontal skirt is a more aerodynamic design, therefore; it helps airflow into the coolers while reducing air recirculation.

Fluid dynamics of the plume and its dispersion may change significantly with air cooler location in a LNG plant. Surrounding equipments such as compressors and/or additional bays of air coolers in a different location may impact the air recirculation pattern [4]. This study is limited to a specific situation where fan outlet may recirculate back into the air cooler inlet. However, an overall increase in ambient temperature due to heat generation by multiple LNG trains may also contribute towards higher recirculation temperature. However, this study points out that the plant should be oriented in such a way, that air coolers are upstream of all other equipments. The author realizes that in most plants wind is blowing in multiple directions throughout the year. In such situations this analysis should be conducted under all wind speed and direction, and the most optimum location based upon average condition should be selected. For maximum benefit, air recirculation study using CFD modeling should be performed in the early stages of site selection study.

No modeling study is complete without proper validation. Bechtel Corporation has instrumented banks of air coolers to validate the CFD recirculation model. Initial validation of the study confirms that the model predictions are close to the measured values. A paper on validation of air recirculation modeling will be presented soon.

6. References

1. Avidan, A., Messersmith, D. and Martinez, B., “LNG liquefaction technologies move towards greater efficiencies, lower emissions”, Oil & Gas Journal, August 19, 2002
2. Ohishi, Masaaki, Moritaka Nakamura and Yoshitsugi Kikkawa, “Availability of refrigeration Process of Baseload LNG Plant”, AIChE Spring Meeting, New Orleans, La, March 10-14, 2002
3. Baumeister, T., “Marks’ Mechanical Engineers’ Handbook”, McGraw Hill, 6th Edition, 1958
4. Berkoe, J., “Fluid dynamics visualization solves LNG plant recirculation problem”, Oil & Gas Journal, March 29, 1999