

UTILIZING TUBE WALL TEMPERATURE MAPPING IN HIGH TEMPERATURE  
GAS-TO-GAS HEAT EXCHANGERS TO MITIGATE SUB-DEWPOINT CORROSION

James Shook

and

Keith Robinson

of AirBTU, Inc

with help from

Greg Becherer

CG Thermal LLC

## TABLE OF CONTENTS

	<b>Page</b>
LIST OF FIGURES AND TABLES	3
SCOPE AND PURPOSE	4
INTRODUCTION	4
METHOD AND PROCESS	5
SAMPLE MAPPING ANALYSIS	6
COMPUTER	8
GAS PROPERTIES	8
BOUNDARY CONDITIONS	8
TUBE WALL MODELLING	9
TURBULENCE MODELING	12
CFD CONVERGENCE	12
RESULTS	14
CONCLUSION AND RECOMMENDATIONS	14
REFERENCES	15

## LIST OF FIGURES

<b>Figure</b>		<b>Page</b>
1	Cold End heat exchanger picture	4
2	CFD model showing flow through a heat exchanger	5
3	Schematic of example heat exchanger used in this study.	6
4	Close up view of temperature profile in a single heat exchanger tube.	7
5	Close up view of the thermal boundary layer along the tube wall.	7
6	Example mesh for analysis that includes shell side, tube wall, and tube side.	10
7	Example mesh for CFD analysis using a thin surface for the tube wall.	10
8	Example mesh for a CFD analysis with a boundary condition for the tube side.	11
9	Plot showing the residuals during convergence.	13
10	Plot of overall heat transfer convergence.	13
11	Outer tube wall temperatures for cold end of example heat exchanger.	14

## LIST OF TABLES

<b>Table</b>		<b>Page</b>
1	Summary of example heat exchanger sizing data.	6
2	Summary of gas properties used in the CFD analysis.	8
3	Parameters used for the shell side gas inlet boundary condition.	9
4	Parameters used for shell side outlet boundary condition.	9
5	Parameters used for the tube side convective boundary condition.	12

## SCOPE AND PURPOSE

In acid manufacturing it is common to use heat exchangers in gas to gas applications like those found in catalytic operations. For exchangers operating at the cold end air entry to the plant, or inter-stage in gas streams nearer the acid dewpoint, there are higher risks of deterioration due to sub-dewpoint acid corrosion or cold-end corrosion.

Sub-Dewpoint Corrosion is the primary cause of early (pre-mature) failure of heat exchangers in the acid manufacturing industry. Designing heat exchangers to mitigate this problem will clearly extend the service life and decrease the maintenance costs for heat exchangers in an acid plant.



Figure 1. Heat exchanger in the cold position

An effective method for controlling sub-dewpoint exchanger corrosion is by fully understanding the tube temperature profile and designing the exchanger to eliminate cold spots. When the heat transfer surface or tube wall temperatures are accurately mapped the heat exchanger can be optimally designed to maintain wall temperatures above the acid dewpoint.

## INTRODUCTION

Heat exchangers are required for thermal heat transfer between gas streams in catalytic operations such as acid manufacturing. In multi-stage catalytic reactions, the process gas is heated by the exothermic catalytic reaction. Heat must then be removed from this gas stream prior to the next catalytic stage to maintain the correct catalytic inlet temperature for the reaction. The heat that is removed from the gas stream is used to heat the cooler process gas entering the plant or a first stage catalyst bed.

Operation of heat exchangers at the cold end air entry to the plant or in inter-stage in gas streams near acid dewpoint increases the risk of sub-dewpoint acid corrosion. Sub-dewpoint or cold-end corrosion is common in large heat exchanger installations in acid manufacturing plants. The most positive method of controlling sub-dewpoint heat exchanger corrosion is by controlling the temperature of the heat transfer surfaces to be above the prevailing acid dewpoint. The acid dewpoint of a gas stream in an acid manufacturing plant is a relative constant since the process is steady. Therefore, if the

heat transfer surface or tube wall temperatures can be accurately mapped, then the heat exchanger can be optimally designed to maintain the wall temperatures above the acid dewpoint.

Computational Fluid Dynamics (CFD) software analyzes fluid systems by finding solutions to the equations that govern fluid flow. These equations coupled with empirically developed turbulence models provide accurate solutions for many different types of fluid flows. As the power of computers has increased, the ability of CFD to analyze more complex flow has increased dramatically over the years. This ability allows a designer to determine the likelihood of condensation problems and to correct them early in the design process. This project outlines one method that can be used to conduct such a study using an example heat exchanger.

## METHOD AND PROCESS

The analysis approach uses a combination of Computational Fluid Dynamics (CFD) modelling, solid works modeling, mesh size and boundary conditions to generate an accurate temperature map to effectively determine areas with the potential to experience condensation. This approach should be used early in the design cycle so that desired modifications in the heat exchanger design can be made to reduce the potential for condensation.

The CFD analysis focuses on the portion of the heat exchanger where the coldest gas impinges upon the tubes or enters the tubes, depending whether the process gas is located on the inside or outside of the tubes. This is the critical region because the flow in this area is the most complicated region within the exchange. Confining this modeling process to the local areas of corrosion concern allows for a computational result in a practical, cost effective method. Modeling and computing time can be measured in the range of several days as opposed to weeks of work and computing to temperature map an entire heat exchanger.

A solid model of the heat exchanger is generated based on tube surfaces, geometry and key cross sections and local tube modeling for coldest area of the heat exchanger.

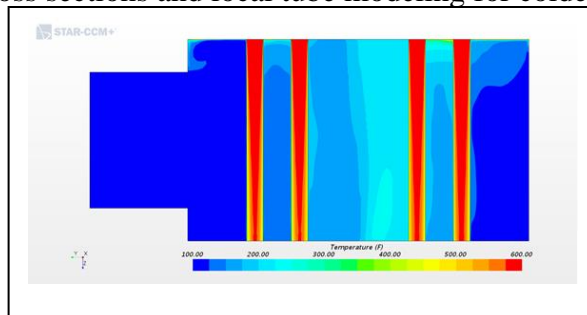


Figure 2. CFD model showing flow through a heat exchanger

The solid model of the “cold end” sections and tube surfaces are then transferred into a CFD model.

The internal and external process gas streams are applied to the model. The CFD model is executed and resolved on Star CCM+ CFD software and meshed to provide a detailed map of heat transfer surface temperatures.

**SAMPLE MAPPING ANALYSIS**

A vast majority of heat exchangers are sized using traditional methods such as NTU methods or spreadsheet methods. These methods calculate overall heat transfer rates using bulk properties of the fluid using rules of thumb, classical heat transfer equations, or data from tests or previous heat exchanger designs. Table 1 summarizes the data used in the initial sizing of the example heat exchanger.

Parameter	Unit	Shell Side	Tube Side
Gas Flow Rate	Lb/hr [kg/s]	880 [400]	880 [400]
Pressure	PSIA [Pa]	15.8 [108,937]	15.8 [108,937]
Inlet Temperature	°F [K]	100 [311]	1475 [1075]
Outlet Temperature	°F [K]	1083 [857]	495 [751]
Duty	BTU/hr [kW]	212,305 [62.2]	-212,305 [-62.2]
Number of Passes	-	10	-
Number of Tubes	-	-	104
Diameter of Tubes	In [mm]	-	0.75 [19]
Wall thickness	In [mm]	-	0.083 [ 2]

Table 1. Sample Heat Exchanger Process Design Data



Figure 3. Heat Exchanger Schematic Drawing

While these methods are effective in calculating overall rates they cannot take non-uniformities in fluid properties or temperatures into account. They can be modified

to analyze a heat exchanger along the length of a tube but this type of information is not needed to size a heat exchanger. The use of CFD allows a look at the properties within the heat exchanger without assuming uniform properties.

As a hot fluid travels through a heat exchanger tube surrounded by a cooler fluid, a layer of cooler fluid begins to form along the boundary with the tube wall. This layer will approach the temperature of the wall and can create an area of condensation within a tube. Since the temperature in the boundary layer is close to the temperature of the wall, the tube wall temperature can be used to predict areas where condensation is likely to occur. Figure 4 and figure 5 below are close up profiles of figure 1.

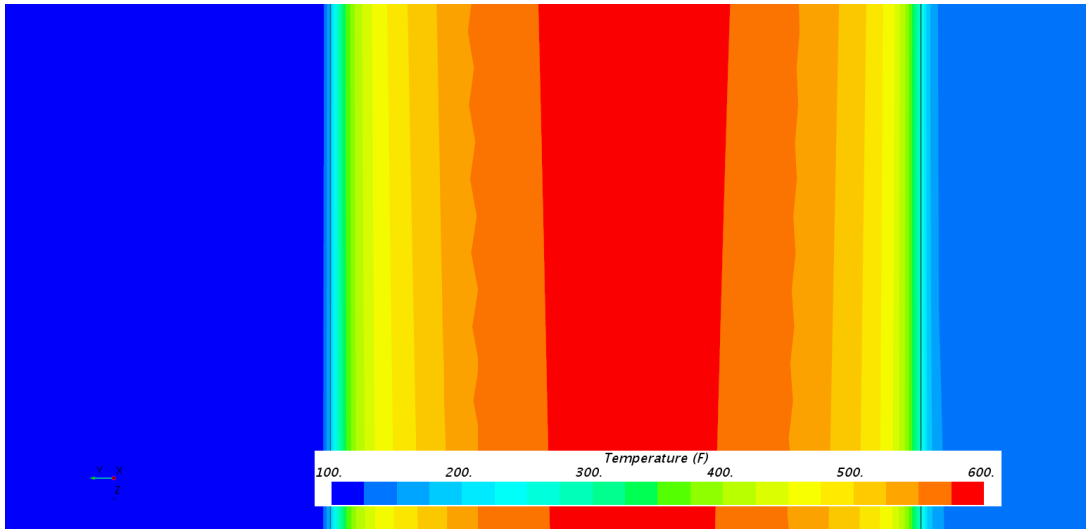


Figure 4. Close up view of temperature profile in a single heat exchanger tube.

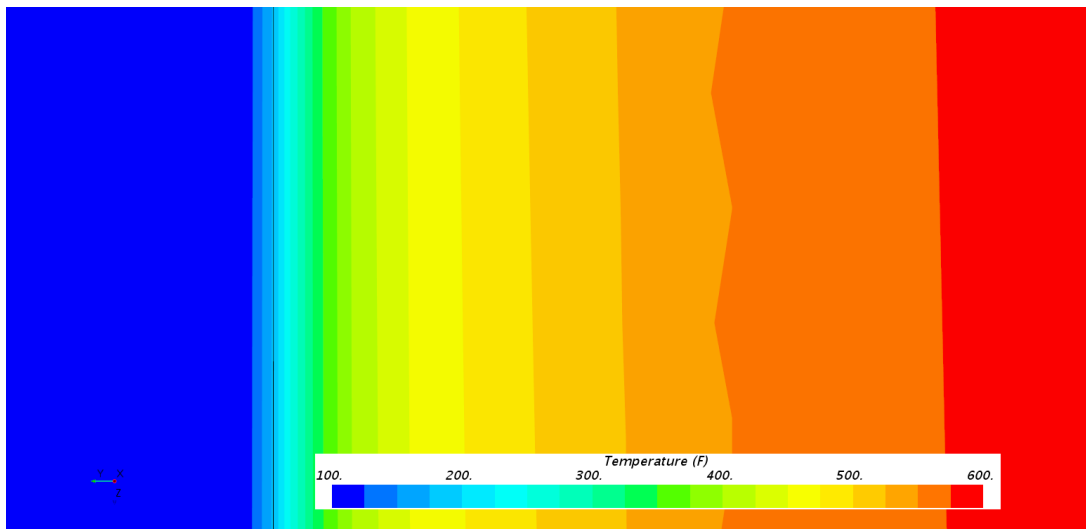


Figure 5. Close up view of the thermal boundary layer along the tube wall.

## COMPUTER

All CFD modeling in this paper is carried out using a Dell Precision T5610 Workstation equipped with two Intel XEON E5-2680 8 core processors and 64 GB of memory. The operating system is Windows 10 and the CFD software is Star-CCM+ Version 12.04.

## GAS PROPERTIES

The gas is modeled as an ideal gas using the properties included in the built in gas database for air along with handbook data (Bolz 1976). The properties used in this model are summarized in Table 2.

Table 2. Summary of gas properties used in the CFD analysis.

Property	Value/Method
Molecular Weight	28.9664 (fixed)
Density	Calculated from Ideal gas law
Specific heat	Polynomial curve fit to data
Thermal conductivity	Power Law fit to data
Viscosity	Polynomial fit to data

## BOUNDARY CONDITIONS

Boundary conditions are one of the most important parts of a CFD analysis. The iterative solver approach will attempt to find a solution that matches the boundary conditions imposed upon the system. If these boundary conditions are incorrect then the solution will also be incorrect. As a result it is very important that the imposed boundary conditions be well thought out. Each of the boundary conditions used in this analysis are described in detail below.

### Shell Inlet

Since this modeling takes place during the design phase there is no information about the actual configuration of the inlet duct. As a result, the velocity magnitude is uniformly applied to the inlet surface and the direction is assumed to be normal to the surface. The shell inlet is modeled as a velocity inlet boundary condition.

The primary components of the velocity inlet boundary condition are the velocity and the temperature of the gas entering the system. These two factors define the mass flow in the system and as a result have the direct effect upon the performance of the system. The turbulence in the inlet is defined using an additional two parameters that define the turbulence intensity and the scale of the turbulence. The turbulence intensity is related to the Reynolds Number at the inlet can be determined using the equation (ANSYS 2009 ):

$$I = 0.16(Re_{DH})^{-0.125}$$



The scale of the turbulence is related to the diameter of the inlet and can generally be assumed to be approximately 7% of the diameter of the inlet (ANSYS 2009). Table 3 gives the parameters used for the shell side gas inlet.

Table 3. Parameters used for the shell side gas inlet boundary condition.

Parameter	Value
Temperature	100 °F [311 K]
Turbulence Intensity	0.0415
Turbulence Scale	0.425 in [11 mm]
Velocity (Normal to inlet surface)	15.97 ft/s [4.87 m/s]

### Shell Outlet

The outlet of the system is modeled using a pressure outlet boundary condition and there are four parameters used to define this boundary. The primary parameter is the outlet pressure which controls the flow rate through the system and in this case is defined to be ambient pressure. The other three factors are temperature, turbulence intensity, and turbulence scale. These three factors are only used in the case of reverse flow through the boundary. In any simulation it is important that the boundaries be located in a location that minimizes any reverse flow so these factors are of less importance than the pressure. Table 4 gives the parameters used for the outlet boundary condition.

Table 4. Parameters used for shell side outlet boundary condition.

Parameter	Value
Pressure (ambient)	0 psig [0 Pa]
Temperature	200 °F [456 K]
Turbulence Intensity	0.415
Turbulence Scale	0.425 in [11 mm]

## TUBE WALL MODELLING

The most accurate way to model a heat exchanger is to use the conjugate heat transfer abilities of the CFD software. In this approach both the tubes and the shell are included in the flow domain and the heat transfer between the two gas streams is calculated by CFD software. It is necessary to mesh the shell as well as each individual tube to use this approach and this can result in a very large mesh and very long computing time.

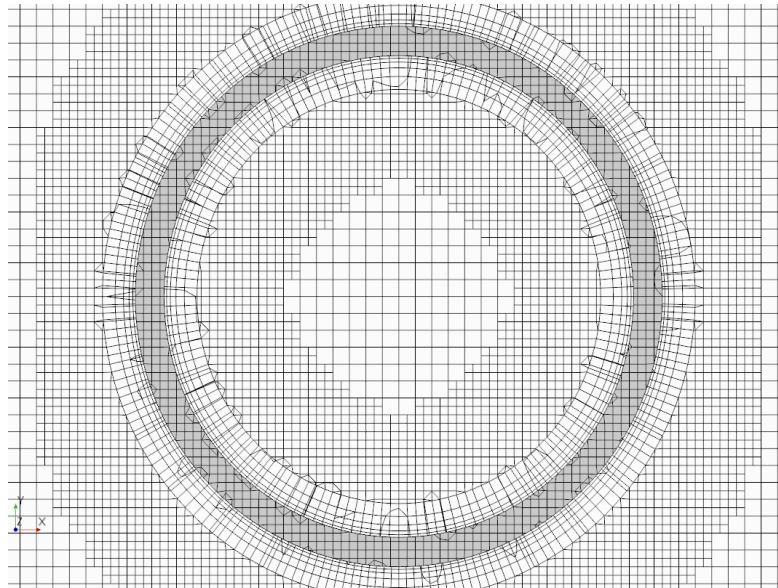


Figure 6. Example mesh for analysis that includes shell side, tube wall, and tube side.

The mesh size can be reduced by modeling the tubes as thin surfaces rather than solid tube and applying a thermal resistance to the tube. Even with this simplification it is still necessary to model the interior of each individual tube in order to solve the flow within each tube. This can result in mesh sizes on the order of 50,000,000 to 100,000,000 cells for a heat exchanger with several hundred tubes.

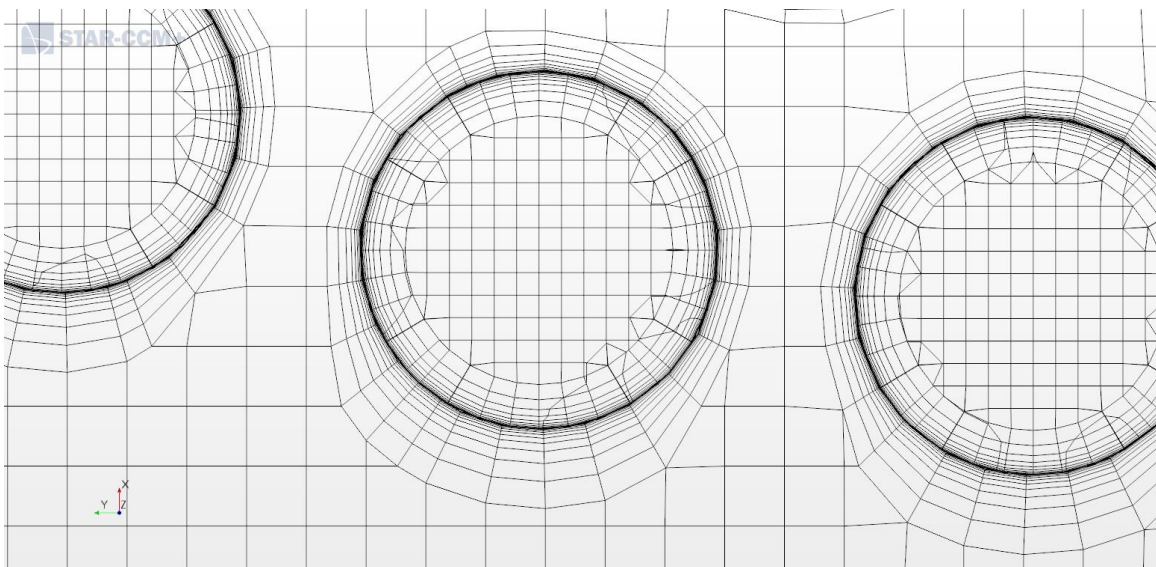


Figure 7. Example mesh for CFD analysis using a thin surface for the tube wall.

Since the heat transfer within a tube is well known and can be calculated using fundamental heat transfer equations and empirical relationships it is possible to simplify the calculation of the heat transfer between the two gas streams by removing the inside

convection and the conduction through the solid from the CFD model by applying a convection boundary condition. This removes the need to create a mesh for the flow area within each individual tube.

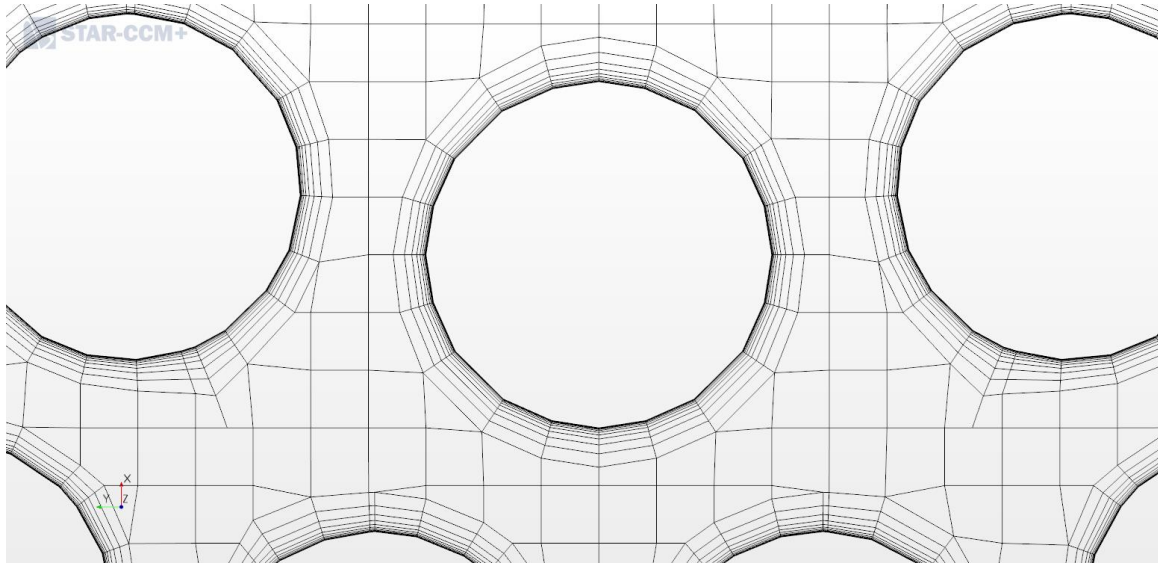


Figure 8. Example mesh for a CFD analysis with a boundary condition for the tube side.

The convection boundary condition requires an ambient temperature and a convective heat transfer coefficient. The ambient temperature is the average of the tube side temperature entering and exiting this section of the heat exchanger. The convective heat transfer coefficient for the tubes can be calculated using the fluid properties evaluated at this average temperature. The average density is used to calculate an average velocity in the tubes and then one of two relationships are used to determine the convective heat transfer coefficient depending upon whether the flow within the tubes is turbulent or laminar. These relationships are (Kreith 1980):

Reynolds Number:

$$Re = \frac{\rho DV}{\mu}$$

Where  $\rho$  is the density of the fluid,  $D$  is the diameter of the tube,  $V$  is the velocity, and  $\mu$  is the viscosity of the gas.

For  $Re \leq 10,000$  (Laminar/Transition flow):

$$h_c = \frac{k}{D} 1.86(RePr)^{0.33} \left(\frac{D}{L}\right)^{0.33} \left(\frac{\mu_b}{\mu_s}\right)^{0.14}$$

Where  $h_c$  is the convective heat transfer coefficient,  $k$  is the thermal conductivity,  $D$  is the diameter of the tube,  $Pr$  is the fluid Prandtl number,  $L$  is the length of the tube,  $\mu_b$

is the fluid viscosity evaluated at the average conditions, and  $\mu_s$  is the viscosity of the fluid evaluated at the surface temperature.

For  $Re > 10,000$  (Turbulent flow):

$$h_c = \frac{k}{D} 0.023 Re^{0.8} Pr^{0.33}$$

Table 5 summarizes the values used for the convective boundary condition on the tubes.

Table 5. Parameters used for the tube side convective boundary condition.

Parameter	Value
Temperature entering section	593 °F [849 K]
Temperature existing section	495 °F [751 K]
Average Temperature	544 °F [800 K]
Average Velocity	29.88 ft/s [9.11 m/s]
Reynolds number	3,170
Convective heat transfer coefficient	5.12 BTU/hr-ft <sup>2</sup> -°F [29.1 W/m <sup>2</sup> -K]

## TURBULENCE MODELING

This analysis is a steady state Reynolds Averaged Navier Stokes (RANS) approach with the Realizable k-epsilon two layer turbulence model. This turbulence model is a refinement of the classic k-epsilon model that has been refined to give better results for boundary layer flow (Siemens 2018). This model may not be appropriate for more detailed analyses that are studying dynamic effects, vibration, vortex shedding or other complicated flow phenomenon. The selection of the appropriate turbulence model will vary depending upon what is being analyzed. Consult the documentation that accompanies the software to get more information on the appropriate model to use.

## CFD CONVERGENENCE

The CFD solver uses an iterative approach to solve the Navier-Stokes and continuity equations (along with other equations) that govern fluid flow. Essentially an initial solution for velocity, temperature, pressure, etc. is applied to every cell in the domain. The CFD solver then solves the pertinent equations in each cell and creates a new solution. The changes between the two solutions, typically termed residuals, are tracked as a way to measure the progress of the solution process. This process is repeated on the new solution to find another new solution. This process repeats until the solution no longer changes when the solution is considered converged.

Convergence of the model can be judged in several ways. The most typical method is to continue the solution process until the residuals reach some predetermined value, typically around  $10^{-4}$ . Another method is to track certain values of interest such as pressure drop, mass flow, or temperature until that value no longer changes. In this study several factors were used to judge convergence. The primary factor was the residuals reaching a value between  $10^{-4}$  and  $10^{-3}$ . Secondary factors used were the net mass flow rate in and out of the system, the heat flux through the tube wall, and the pressure drop through the system.

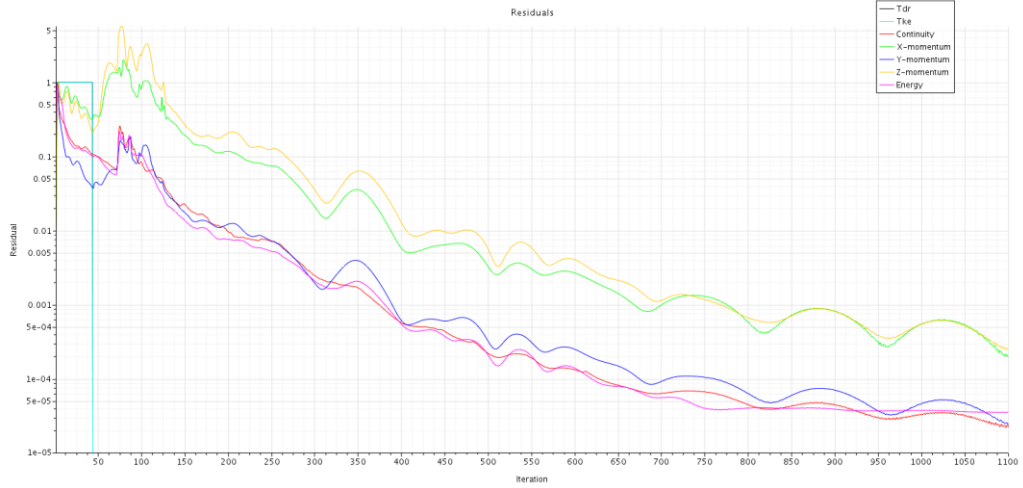


Figure 9. Plot showing the residuals during convergence.

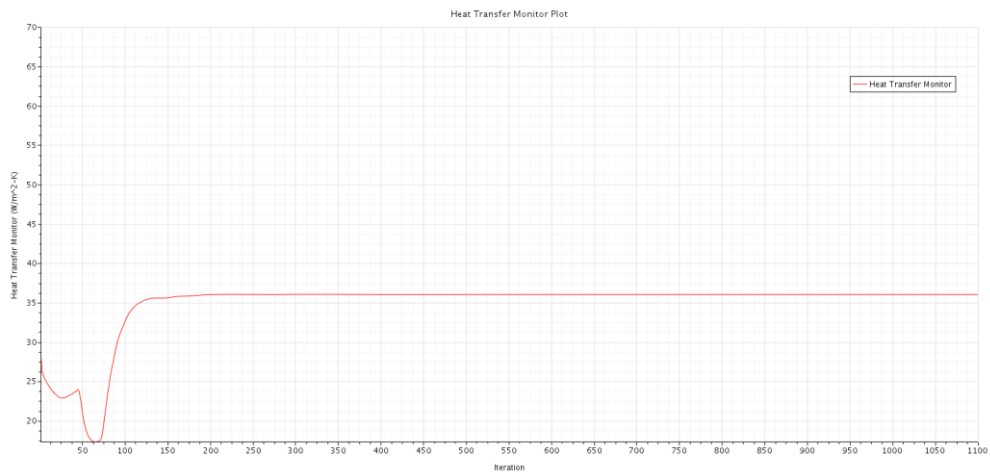


Figure 10. Plot of overall heat transfer convergence.

## RESULTS

One of the most useful ways to present the results from a CFD analysis is through visual output. The figure below shows the temperature on the shell side (the cold side) of the cold end of the example heat exchanger. This figure quickly shows that the outer tube wall temperature varies across the tubes directly in front of the cold air inlet. With a thin metallic tube the temperature on the inside wall is within one or two degrees of the outer tube wall temperature. If this inner wall temperature is near or below the condensation point of the fluid within the tube it can be expected that condensation will occur during operation. In the case of gases containing SO<sub>x</sub> compounds, these areas will experience buildup of solids and acids.

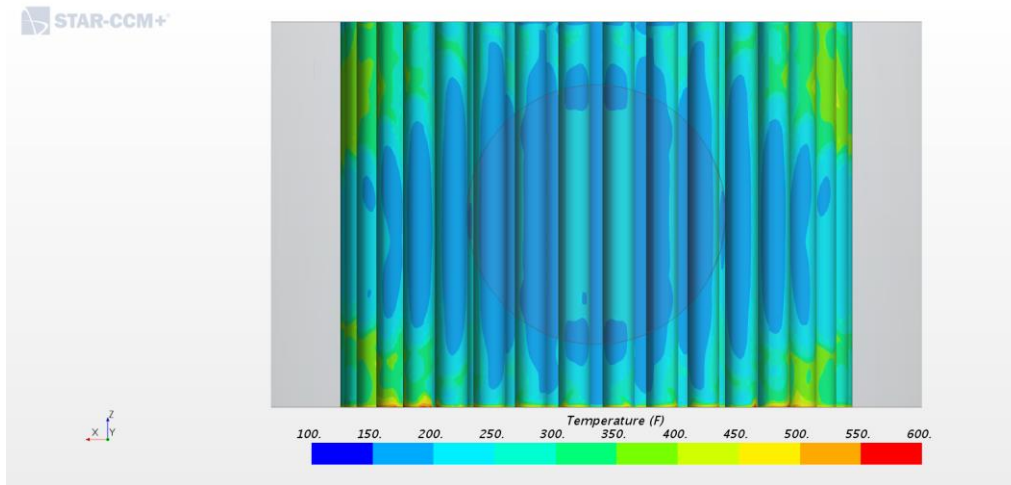


Figure 11. Outer tube wall temperatures for cold end of example heat exchanger.

## Conclusions and Recommendations

1. Confining this modeling process to the local areas of corrosion concern allows for a computational result in a practical, cost effective method. Modeling and computing time can be measured in the range of several days as opposed to weeks of work and computing to temperature map an entire heat exchanger.
2. Model development, mesh size, boundary conditions and CFD execution are studied and demonstrated to execute a tube wall temperature map efficiently. Critical areas of a large radial flow heat exchanger in the cold end of an acid manufacturing process are studied in this presentation.
3. Specific CFD components of setting up the model, strategies for determining mesh size and cell count, and boundary conditions are presented in detail.
4. Development of a tube wall surface temperature map is a useful tool in designing a heat exchanger with increased life. This model, once developed for a given heat exchange system, can then be efficiently modified for different design scenarios.

## REFERENCES

Bolz R., Tuve G. 1976. CRC Handbook of tables for Applied Engineering Science 2<sup>nd</sup> Edition. CRC Press Boca Raton, FL. pp 9-12.

ANSYS FLUENT 12.0 Using Flow Boundary Conditions. 2009. ANSYS FLUENT 12.0 User's Guide. Online at:  
<https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node238.htm>

ANSYS FLUENT 12.0 Realizable k-e Model. 2009. ANSYS FLUENT 12.0 User's Guide. Online at:  
<https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node60.htm>

Kreith F, Black W. 1980 Basic Heat Transfer. Harper & Row. New York. pp 237-248.