CFD and Thermal Calculations of the Pre-heating of a Heat Exchanger

Wies A. Hageraats, Hans J. Bos

Abstract

A CFD study is performed to predict the velocity distribution of the hot air flow that is used to pre-heat the tube bundle of a heat exchanger. The obtained data is used to predict the difference in temperature growth between individual tubes located in the exchanger over time. Too large thermal gradients between individual tubes may cause the tubes to buckle. For the purpose of the Computational Fluid Dynamics (CFD) simulations, OpenFOAM® has been used and to obtain detailed thermal results a separate Finite Element Analysis (FEA) program has been written in Visual Basic 6.

1. Introduction

For the design of a heat exchanger, a number of options have been considered to (pre)heat the exchanger appropriately. (Pre)heating is required at the start-up of the exchanger and for Post Weld Heat Treatment (PWHT) purposes.
From a preliminary design study it was clear that the temperature gradients in the tube bundle should not be too large in order to avoid tube buckling.
In a later stage the preheating process by means of electric heat tracing was studied. The conclusion indicated that heating by means of electric heat tracing only is not recommended since radiation will generate considerable large temperature gradients across the tube bundle.
This project focuses on the heating process of the tube bundle by means of a hot air stream that is forced through the tubes. This heating process is part of the Post Weld Heat Treatment. The hot air stream is used to minimize thermal gradients across the tubes.
Heating is realized by placing the exchanger inside a furnace, and heat it up gradually from ambient to 610° in approximately 15 hours with the application of burners that are placed inside the furnace. Then the temperature is maintained for 2 hours after which the exchanger is cooled-off gradually in approximately 15 hours back to ambient.
CFD calculations were performed to predict the maximum possible flow velocity differences within individual tubes. The results are used to estimate the resulting thermal "lag" between individual tubes that is a consequence of velocity variations.

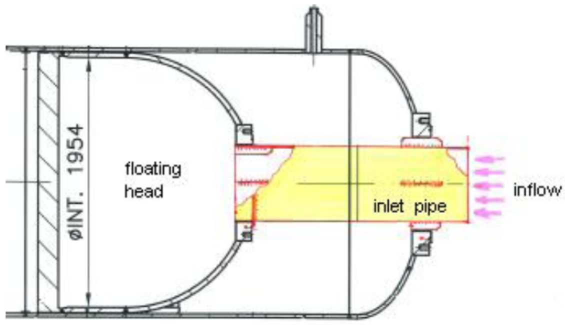

Figure 1: Floating head compartment with air inlet device.

To predict the resulting thermal lag, a finite element (FE) program has been written to simulate the heating of the tubes in time. The calculated temperature difference is then compared to the allowed buckling limit.

2. Hot Air Intake

The hot air is released into the floating head compartment through the floating head manhole by means of an inlet pipe that is connected to a hot air fan. The air is than forced through the 6092 tubes that are connected to the floating head tubesheet. The air leaves on the other side of the exchanger through a process nozzle. A sketch of this configuration is shown in figure 1, illustrating the floating head compartment only. Based on the possible flow conditions a worst case scenario is determined that corresponds to a mass

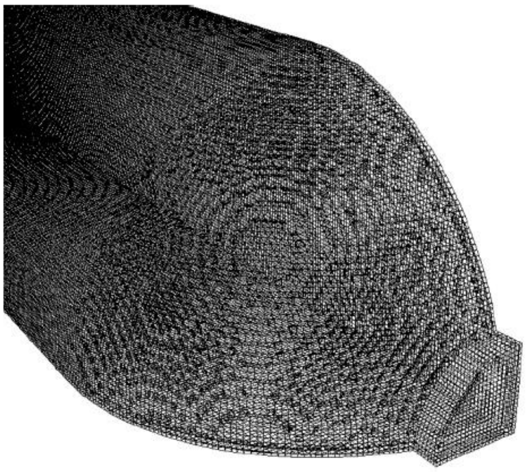

Figure 2: 3D plot of the CFD grid of the floating head.

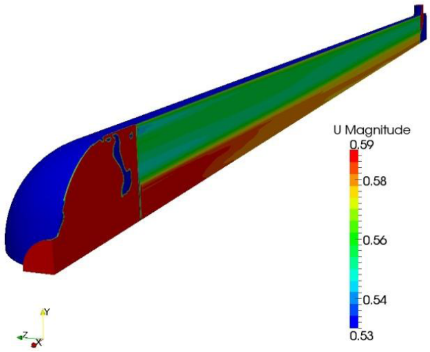

Figure 3: Velocity distribution inside the tubes modeled as a porous medium.

flow yielding the highest possible flow variations among individual tubes.

3. CFD Calculation Method

The tube bundle that is located between the two tube sheets is modeled as a "porous medium". Modeling all 6,092 tubes directly and implementing a suitable CFD mesh would take too much preparation and computational time. The concept of a porous medium takes away the need to model the relatively complex geometry of the tube bundle and replaces it by an "equivalent" volume. The pressure loss in the porous medium is based on a common expression for the pressure drop through pipes, which is the following:

$$\boldsymbol{\nabla} p = \lambda \cdot \frac{L}{d}\frac{\rho}{2} \cdot u_{avg}^2 \qquad (1)$$

λ is the Darcy Friction Factor, L is the length of a tube and D is the inner diameter of a tube. u_{avg} is the average air velocity inside a tube. For laminar flow (Poiseuille flow), the following equation may be used to determine the friction factor:

$$\lambda = \frac{64}{Re} \qquad (2)$$

Two different meshes have been applied one with 4,000,000 cells and a second one with 7,000,000 polyhedral cells.

4. CFD Calculation Results

The CFD results showed that the difference between the highest measured and the lowest measured velocity in the tube bundle is approximately 11% when compared to the average tube velocity.

5. Thermal Analysis

As was indicated earlier, a possible thermal lag may be present among individual tubes during the heating of the exchanger. Due to air flow velocity variations among tubes the convection of heat to the tube metal will be different for different tubes. These differences may build-up in time and eventually yield severe temperature differences which in turn may buckle the tubes.
To evaluate the heating of the tubes it was decided to write a small finite element (FE) program code that describes the heating process of the tubes in time. To do so, the following governing heat (differential) equations are used:

Heat Equation of Air:

$$\begin{aligned} c_p \cdot \rho \cdot A \cdot \frac{\partial T}{\partial t} \cdot dx &= c_p \cdot \rho \cdot A \cdot \frac{\partial T}{\partial t} \cdot u \cdot dx \\ &- q_w \cdot 2\pi r_0 \cdot dx \end{aligned}$$

Heat Equation of Metal:

$$c_{p_{tube}} \cdot \rho_{tube} \cdot A_{tube} \cdot \frac{\partial T_{tube}}{\partial t} \cdot dx = q_w \cdot 2\pi r_0 \cdot dx$$

In order to solve the differential equations a numerical discretization is applied both for space and time. An explicit numerical scheme is applied with backward differencing for the spatial derivatives. The applicable spatial coordinate is x (length along tube) and the applicable temporal variable is t. The following finite differences are used:

$$\begin{aligned} c_p \rho A \frac{T_j^{t+1} - T_j^t}{\delta t} &= c_p \rho A \frac{-\left(T_j^t - T_{j-1}^t\right)}{\delta x} u \\ &- Nuk\left(T_j^t - (T_{tube})_j^t\right)\pi \end{aligned} \qquad (3)$$

$$\begin{aligned} c_{p_{tube}} \rho_{tube} A_{tube} \frac{{T_{tube}}_j^{t+1} - {T_{tube}}_j^t}{\delta t} &= \\ Nuk\left(T_j^t - (T_{tube})_j^t\right)\pi \end{aligned} \qquad (4)$$

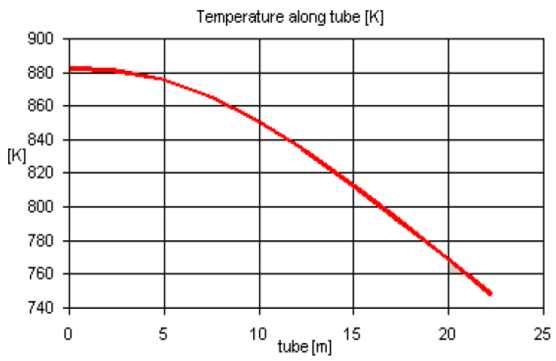

Figure 4: Temperature distribution as function of tube length after 17 hours.

Time [hr]	11% lower	u_{avg}	11% higher
15	761	774	784
17	820	832	841

Table 1: Calculated temperature variations.

The following boundary conditions are applied:

$$T^t_{j=0} = T_{inlet} \tag{5}$$

The following initial conditions are used:

$$T^{t=0}_j = T_{ambient}, \qquad T^{t=0}_{tube_j} = T_{ambient} \tag{6}$$

The results show that the expected average tube metal temperature after 17 hours of heating is approximately 559°C.
Figure 4 shows the temperature distribution along the length of the tubes after 17 hours. Next the thermal lag is determined, such that the maximum occurring velocity difference is used, as determined from the CFD calculations. The maximum velocity difference was calculated as 11%.
Three runs are made: one with the average velocity inside the tubes, one with a velocity that is 11% larger and one with a velocity that is 11% smaller. The average metal temperatures after 15 and 17 hours are reported in table 1; since the largest differences are expected at the end of the heating process (the temperature difference accumulates in time).
The maximum difference between the average velocity case and either one of the two extremes is 13°C, which is smaller than the allowable buckling limit temperature.

6. Conclusions

This article described the CFD analysis of the velocity distribution of hot air inside the tubes of a tube bundle heat exchanger. The hot air is used to minimize the thermal gradients among tubes during Post Weld Heat Treatment.

From the CFD study it was found that the largest velocity variations are 11% in comparison to the average velocity through the tubes.
From a thermal analysis it was seen that this velocity variation causes a temperature difference of approximately 13°C after 17 hours of heating between individual tubes.

7. Bibliography

Ferziger, J. H. & Peric, M. (2002), *Computational Methods for Fluid Dynamics*, third edn, Springer-Verlag Berlin.
Holman, J. (n.d.), *Heat Transfer*.
und Chemieingenieurwesen, V.-G. V. (n.d.), Vdiwarme atlas.
White, F. M. (1991), *Viscous Fluid Flow*, second edn, McGraw-Hill Inc.
Wilkes (n.d.), *Fluid Mechanics for Chemical Engineers*.