CFD Analysis of the Salt Flow in a Tube Heat Exchanger

Wies A. Hageraats, Hans J. Bos

Abstract

The results of the Computational Fluid Dynamics (CFD) analysis are presented for a oil-salt heat exchanger that is located on a thermal solar plant. The analyses focus on the salt flow in the shell side of the exchanger and are intended to capture the salt velocity distribution, pressure drop and possible wake regions that can cause excessive corrosion.
In prior to the full 3D flow simulations a detailed study is made to model the pressure drop of the tube bundle by means of a porous medium. For the purpose of the CFD simulations OpenFOAM® is used.

1. Introduction

In this abstract the results of the CFD analysis are presented for a single oil-salt heat exchanger that is located on a thermal solar plant. The analysis focus on the salt flow in the shell side of the exchanger and are primarily intended to capture two important flow characteristics:

- *The wake region near the stationary tubesheet.* This region is formed at the location where the tubesheet is welded to the longitudinal baffle plate. The size, flow pattern and the velocity magnitude of the wake strongly determine the corrosion intensity.
- *The cross sectional distribution of the molten salt along the axial direction of the exchanger.* A fast homogenous distribution of the salt along the cross section of the exchanger will benefit thermal performance of the unit.

A sketch of the heat exchanger is shown in figure 1, where various components are highlighted. In the shell side of the exchanger approximately $6,000$ tubes are located, which are supported by 63 baffle grids. The exchanger has a length of approximately $25m$ and a diameter of approximately $2.5m$.
The tube bundle that is located inside the main shell strongly influences the salt flow in the exchanger and it is therefore significant to capture these effects adequately.
However, to include all 6,000 tubes in a CFD model and to implement a suitable mesh is

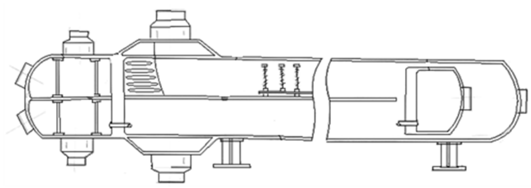

Figure 1: Sketch of the heat exchanger (tubes are not shown).

extremely elaborate and has significant consequences on both preparation and computation time.

In order to reduce preparation and computation time and bring the study into a feasible time frame, it is chosen to apply the concept of a "porous medium". This concept removes the need to model the relatively complex geometry of the tube bundle and replaces it by an "equivalent" volume. This equivalent volume is geometrically simple, since the volume is shaped as a solid semi-cylinder, with a radius equal to the outer tube limit (OTL) of the tube bundle.

The medium located inside is replaced by an equivalent medium called the porous medium. The mathematical definition of this porous medium will involve the definition of pressure loss coefficients that will describe the pressure loss of the salt within the equivalent volume and will approximate the real pressure loss within the tube bundle.

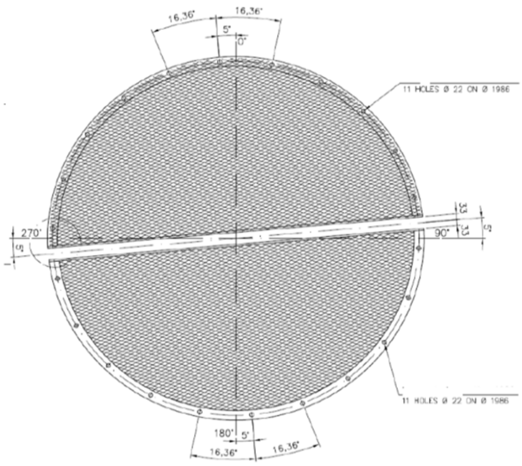

Figure 2: Sketch of cross-section with baffle grid and longitudinal baffle.

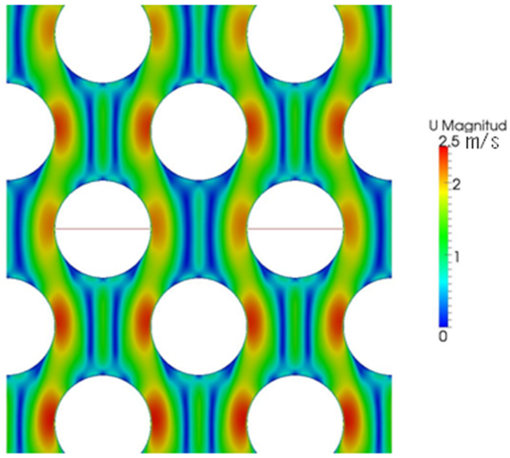

Figure 3: CFD results of salt cross flow in the tube sheet.

2. The Definition of a Porous Medium

In advance to the 3D flow simulations a detailed study is performed to obtain the pressure drop over the tube bundle in all relevant flow directions. These values are then used to define the porous medium.
For this purpose the following steps are performed:

Step 1. CFD analyses of a 2D model, containing an elementary tube arrangement under cross-flow, are made to compare the obtained pressure loss with what has been found in literature.

Step 2. A numbers of CFD models are made to simulate the cross flow along the diamond tube pattern used in the heat exchanger to obtain typical pressure loss coefficients as a function of lateral flow inclination.

Step 3. The axial pressure loss coefficient for the heat exchanger tube bundle is determined on basis of formulations found in various literature:

$$\frac{1}{\Lambda^{0.5}} = 2.0 \log_{10}(0.64 Re_{D_h} \Lambda^{0.5}) - 0.8 \quad (1)$$

Step 4. With the calculated pressure loss coefficients, a porous medium model is implemented in the CFD code OpenFoam® that will approximate the pressure loss in the real tube bundle.

3. Setup of the 3D CFD Model

The earlier defined pressure loss coefficients are implemented as a porous medium into the full 3D CFD model. The following steps are performed:

Step 5. A 3D model is created in Pro-Engineer containing the annular distributor and the main shell of the heat exchanger.

Step 6. Proper boundary conditions are applied to the model. Furthermore the flow solver with the porous medium characteristics is prepared.

Step 7. An efficient and accurate CFD mesh is generated.

Step 8. Relevant simulation cases are defined and the porous model is set-up.

4. CFD Simulation Results

From the calculations it is found that in all simulation cases the wake region near the stationary tubesheet has the characteristics of a stagnating flow. A stagnation flow occurs whenever (1) a flow impinges on a solid object or when (2) two separate streams of flow meet under a sharp angle are forced in the same direction. In both cases a point / region will exist where flow velocity values are close to zero, however no recirculation of the flow occurs for this flow types.
From the simulations it is found that the first type of stagnating flow occurs when the salt inlet is considered and the second type occurs when the salt outlet is considered.
The wake has a horse-shoe like shape in the cross-sectional plane. It extends conically with axial direction of the exchanger (i.e. reducing in size going downstream). The smallest velocities are measured in the two "toes" of the horse-shoe.

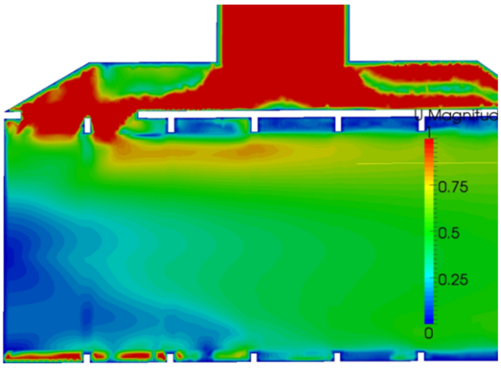

Figure 4: CFD contour plot of velocity near tube sheet/longitudinal baffle.

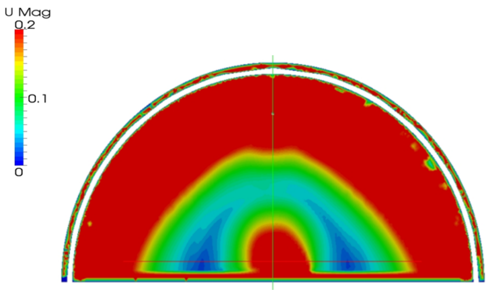

Figure 5: CFD contour plot of velocity near tube sheet/longitudinal baffle.

Finally it should be noted that from the simulations it is visible that after approximately 8 baffle spacing the axial flow does not vary significantly along the axial direction of the exchanger, indicating that the salt flow has reached a quasi-three-dimensional state (flow properties only change in cross sectional plane) and that the thermal performance is optimal.

5. Conclusions

The salt flow in the heat exchanger is analysed by means of CFD. The two main goals of these simulations is:

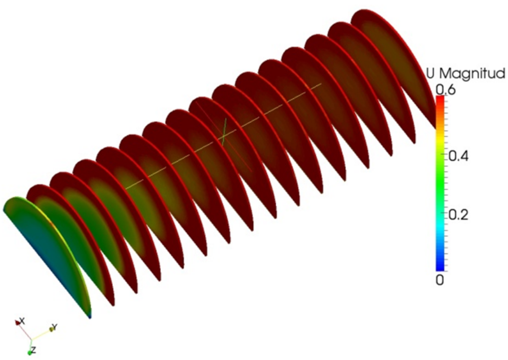

Figure 6: CFD contour plot of velocity near tube sheet/longitudinal baffle.

- Determination of the wake region near the stationary tubesheet (for the purpose of corrosion analysis).
- Determination of the flow distribution in axial direction of the heat exchanger.

From the analysis it is seen that the wake region near the tubesheet is a stagnation type of flow and not a recirculation area, which is an important conclusion for corrosion analysis. Furthermore it is seen that the salt flow distributes relatively fast after entering the heat exchanger and that after approximately 8 baffle spacing the flow does not change in axial direction any more, which is benefical for the thermal performance of the unit.

6. 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.), Vdi-warme atlas.
White, F. M. (1991), *Viscous Fluid Flow*, second edn, McGraw-Hill Inc.
Wilkes (n.d.), *Fluid Mechanics for Chemical Engineers*.