CFD Analysis of the Salt Flow in a Tube Heat Exchanger

Application of the Porous Medium Concept

Ir. W. Hageraats (MSc), wies.hageraats@dynaflow.com **Dr. Ir. Hans Bos (MSc, PhD)**	Dynaflow Research Group Houtsingel 95, 2719 EB, Zoetermeer The Netherlands

Abstract

The results of the Computational Fluid Dynamics (CFD) analysis are presented for an 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.

The CFD analysis focus on the salt flow in the shell side of the exchanger is primarily intended to capture two important flow characteristics, the wake region near the stationary tubesheet and the cross sectional distribution of the moten salt along the axial direction of the heat exchanger, see Figure 2. The shell side of the heat exchanger contains approximately 6,000 tubes, which are supported by 63 vertical baffle plates. The exchanger has a length of approximately 25 m and a diameter of approximately 2.5 m. The tube bundle strongly influences the salt flow in the exchanger and it is therefore significant to study its influence in detail. However, the inclusion of all 6,000 tubes in a CFD model and to implement a suitable computational mesh are extremely elaborate and has significant consequences on both preparation and computation time, which are relavant for industrial applications.

In order to reduce the computational effort, it is chosen to model the tube bundle by a porous medium. This concept eliminates 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 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. A flow solver in the framework of OpenFOAM® is developed and improved in order to solve the instationary incompressible Navier-Stokes equations for turbulent flows using a porous medium region. In advance to the 3D flow simulations a detailed CFD study is performed to obtain the pressure drop over the tube bundle in all relevant flow directions (Figure 3). These results are then used to define the appropriate porous medium properties to model the tube bundle adequatly. We will present detailed flow field results (Figure 1) in the porous region as well as the thermal performance of the heat exchanger.

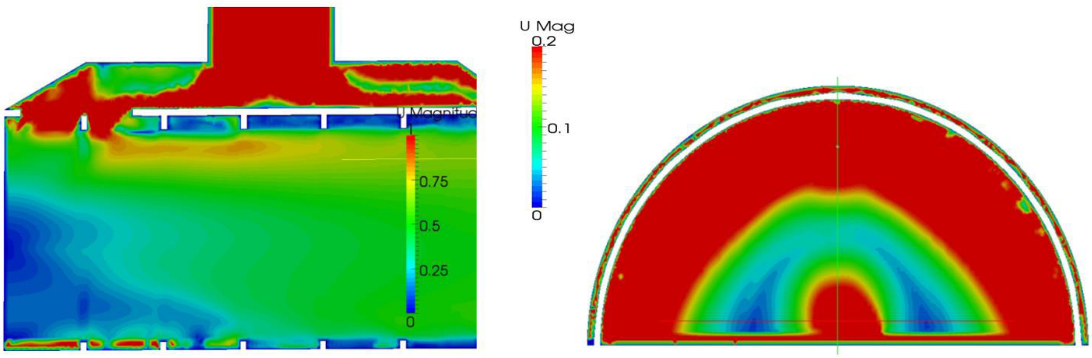

Figure 1: Velocity contours near in the Heat Exchanger. The tubesheet is modelled by a porous medium.

Key words: Heat and mass transfer, Porous medium, Heat Exchanger, OpenFOAM

References

[1] Holman, J. (2001), Heat Transfer, McGraw-Hill Inc.

[2] Vdi- Warme Atlas, V.-G. V. und Chemieingenieurwesen.

[3] White, F. M. (1991), Viscous Fluid Flow, Second Edition, McGraw-Hill Inc.

[4] Wilkes, J. O. (1999), Fluid Mechanics for Chemical Engineers, Prentice-Hall Int. Ltd.

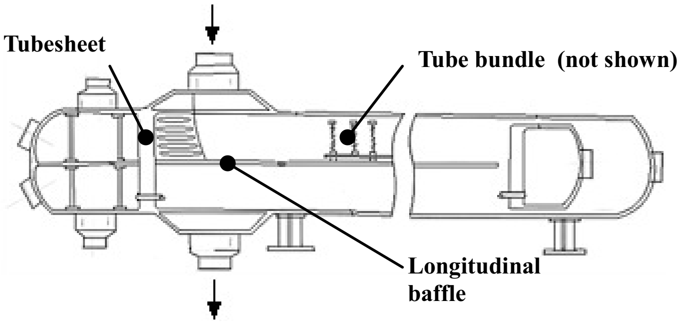

Figure 2: Configuration of the Heat Exchanger.

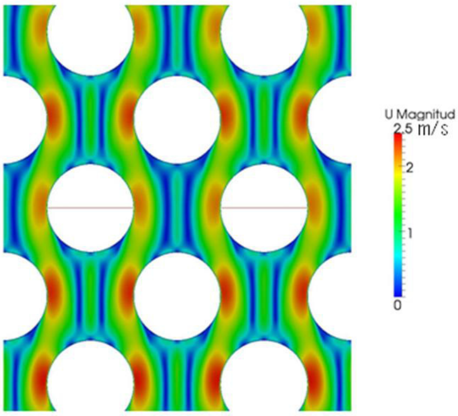

Figure 3: Cross sectional velocity field of the salt flow in the tubesheet.