Natural Convection and Radiation Cooling of a Flow-Meter Heat Sink

Kathryn Potten[1], Timothy Brauner[1] and Hu Leilei[2]

CHAM, London,UK[1], ZWSOFT, Guangzhou, China[2]

Introduction

Beijing Instrument Industry Group Co., Ltd. specializes in the design and manufacture of pipe flow meters, which are designed to measure the volumetric or mass flow rate of fluid within a pipeline. For high temperature fluids, a metal heatsink with cooling fins is often installed between the flow meter and the pipeline. This is done to ensure that the fluid temperature entering the flow meter is not so high as to damage to the meter. The length of the heat sink is increased with the temperature of the pipe-line fluid to allow for more cooling. To address different temperatures of the pipeline fluid, it is necessary to conduct convection heat dissipation analysis of the heatsink in advance and verify its heat dissipation effect under different temperatures. PHOENICS can assist in the design process through CFD simulations of the heatsink under these different conditions. For the purposes of this demonstration heatsink simulation, a number of simplifying modelling assumptions have been made, as explained below.

Figure 1 - Flow meter

Figure 2 - Geometry used for CFD model in PHOENICS

The PHOENICS CFD model

Modelling Assumptions and Strategy

The geometry to be modelled is a pipe flow meter. This consists of a finned metal tube (the heatsink) connecting the flow meter - where measured flow is displayed - to the pipe - where the apparatus for measuring the flow is situated. The section of pipe attached to the flow meter is then inserted into a larger pipe through which the fluid whose flow is to be determined moves (**Error! Reference source not found.**). For this demonstration, the temperature of this fluid is taken to be 200^{0}C. The heatsink is a hollow tube of steel containing water.

To create the model for this demonstration, the geometry is simplified, keeping only the heatsink and omitting the flow meter (above the heatsink) and pipe (below the heatsink), as it is the temperature at the top of the heatsink that is of interest. Combined with approximations of the boundary conditions, a complete heat dissipation analysis model is formed.

The energy equation is solved for temperature to predict the temperature entering the flow meter at the top of the heatsink. A hand-calculated estimation of the radiative heat flux indicates that radiation is expected to be of significance in this case when compared to the convective heat flux, and thus radiation is modelled using the PHOENICS IMMERSOL radiation model. The emissivity of the steel is taken to be 0.3 based on experimental and numerical results at the expected temperatures [1].

The Rayleigh number for this problem has been estimated to be $2x10^8$, which indicates that the flow is laminar, although perhaps undergoing transition to turbulence [2]. Thus the simulation is run as laminar. The case is entirely free convection, and the buoyancy model used is density-difference.
Two additional simulations are performed without radiation, one with fins and one without fins. The first allows an understanding of the effects of radiation and its importance in the model. The latter serves to provide a validation of free convection for a vertical cylinder at the specified Rayleigh number, and does not include the radiation model to remain consistent with the literature.

Physical Properties:
The domain is filled with air, at an ambient temperature of 20^oC, modelled using the ideal gas law. As air shows reasonable changes in properties across the range of expected temperatures (from $20\text{-}200^oC$) [2], kinematic viscosity and thermal conductivity were made variable, both using Sutherland's law.
The water inside the heatsink is modelled as a solid with the properties of water, as movement within the water is negligible and thus can be assumed stagnant. Properties of water are taken as constant as these do not change a significant amount at the expected temperatures of this model.

Geometry:
The heatsink is made up of two sections, with a total length of 25cm. The top section is a smooth steel cylinder, and the bottom a steel cylinder with 26 regularly spaced annular fins (Figure 2). The water is modelled as a cylinder aligned centrally within the heatsink.

Boundary Conditions:
(1) The bottom of the flow meter is given a fixed temperature of 200°C to represent the fluid temperature inside the pipe. This is satisfied by creating a plate at the bottom of the heatsink with surface temperature set to 200^oC.
(2) The bottom of the fluid domain uses a heat-insulated wall boundary (to represent the top of the pipe), and the remaining outer surfaces use open boundary conditions. Due to the problem being entirely natural convection, there are no velocity inlet or outlet boundaries.

Solution Domain and Mesh

This simulation is run in 2D cylindrical polar coordinates to exploit the axisymmetric nature of the problem. The solution domain is taken to be a little over twice the height of the heatsink, with 4 times the diameter of the heatsink in the Y direction, corresponding to a cylindrical polar grid of 0.1 radians in the X direction, radius (Y) 0.1m, and height (Z) 0.58m. The grid is 1x61x484 cells, with sufficient cells in the Z direction to allow 5 cells per fin and in between each fin on the heatsink. This should be enough cells to properly resolve the heat transfer around the fins, and capture some of the flow between them. No grid sensitivity study has been conducted for this demonstration exercise.

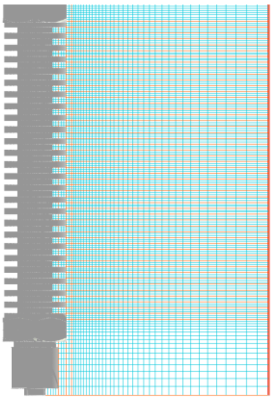

Figure 3 - Zoom on grid around

Results and Discussion

The maximum temperature inside the heat sink is 198°C, which is found at the bottom of the heatsink where it meets the pipe (and 200°C boundary condition is specified), with temperature gradually decreasing towards the top end of the heatsink, where the flow meter would be situated (Figure 4). At the top of the heatsink, where the heatsink meets the flow meter, the temperature is 53°C.

Table 1 presents results from the three runs: case 1, finned with radiation; case 2, finned without radiation; case 3, smooth without radiation. The Nusselt number (Nu) was estimated for each case, and case 3 compared to correlations for a smooth vertical cylinder in the literature. These were found to be consistent. Total heat transfer coefficient (h), as well as heat fluxes (convective Qc, radiative Qr and total Qt) and temperature at the top of the heatsink are shown. For the estimation of h and Nu, the total area over the fins was used, and average surface temperature over the heatsink. Figure 5 shows the velocity contours.

Table 1 - Nusselt number, heat transfer coefficients, fluxes and temperature at top of heatsink results

	Nu	h (W/m^{2}K)	Qc (W)	Qr (W)	Qt (W)	Temperature at heatsink top (°C)
PHOENICS Case 1	62.79	6.59	0.1206	0.1205	0.2411	53.19
PHOENICS Case 2	31.22	3.28	0.1693	0	0.1693	78.22
PHOENICS Case 3	63.70	6.69	0.2577	0	0.2577	109.39
Data [2]/[3]/[4]/[5]	67.97-74.79	7.14-7.85	-	-	-	-

The temperature contours show that the heatsink is behaving as expected, with the heatsink cooling towards the top as heat is convected away from the fins. Comparing the run with fins to that without shows the importance of the fins in reducing the temperature at the top of the heat sink sufficiently, with a difference of around 30^{o}C seen at the top of the heatsink. Comparing with and without radiation, it is clear that radiation plays a significant role in the cooling, with a difference in temperature at the top of the heatsink of 25^{o}C, and heat flux due to radiation being the same as that due to convection. Most of the heat loss due to radiation occurs near the base of the heatsink where the temperature is highest. This has allowed a reduction in temperature from 200^{o}C to 53^{o}C, enough that the flow meter will not be damaged. This demonstrates that the heatsink is working properly and is consistent with true conditions.

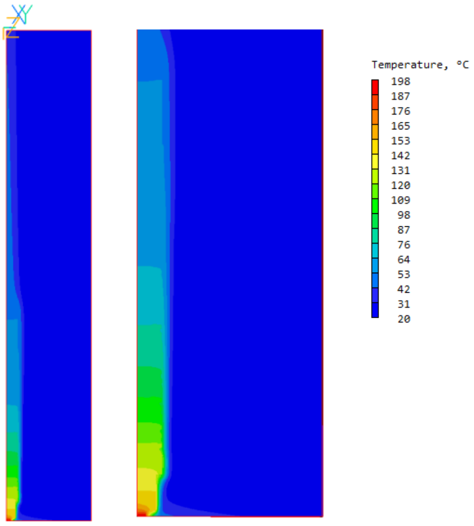

Figure 5 - Temperature contours

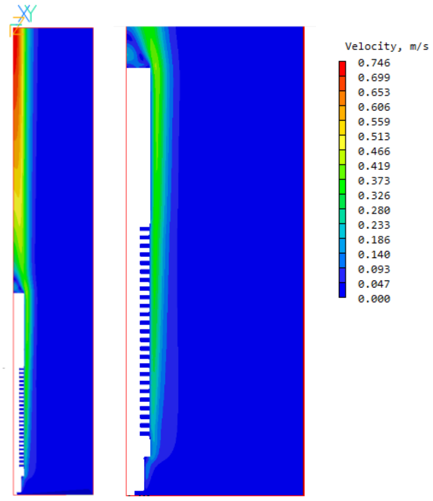

Figure 4 - Velocity contours

Conclusions

PHOENICS has been used to create a simplified CFD model of the heatsink for a pipe flow meter, in order to predict the temperature at the top of the heatsink, where it meets the flow meter. The heat dissipation analysis has confirmed that for this simplified case, the heatsink can provide sufficient cooling to prevent damage to the flow meter. It has been shown that radiation is important in this case and cannot be neglected. The simplified CFD model could be used to investigate the effects of the fin geometry on the thermal performance of the heatsink. In addition, effects of length and diameter of the tube could be explored. Further simulations could be conducted using more complex geometry and boundary conditions, taking into account the curvature and surface temperature of the pipe itself, as well as the flow meter above the heatsink, which would all have effects on the flow and thus the heat transfer.

References

1. Sadiq, H., Wong, M. B., Tashan, J., Al-Mahaidi, R. & Zhao, X. L. *Determination of steel emissivity for the temperature prediction of structural steel members in fire*. J. of Materials in Civil Engng.;25(2), 167-173, http://dx.doi.org/10.1061/(asce)mt.1943-5533.0000607, (2013).
2. Ozisik, M.N., *Heat Transfer, A Basic Approach*. New York: McGraw-Hill; (1985)
3. Popiel C.O., Wojtkowiak J., Bober K. *Laminar free convective heat transfer from an isothermal vertical slender cylinder.* Experimental Thermal & Fluid Science. 32:607-613. https://doi.org/10.1016/j.expthermflusci.2007.07.003, (2007).
4. S.W. Churchill, H.H.S. Chu. *Correlating equations for laminar and turbulent free convection from a vertical flat plate*. Int. J. Heat Mass Transfer; 18:1323–1329, (1975).
5. T. Cebeci. Laminar-free-convective-heat transfer from the outer surface of a vertical slender circular cylinder, in: Proc. 5th Int. Heat Transfer Conf., vol. 3. Paper NC 1.4, Tokyo; pp. 15–19, (1974).