MODELLING AND SIMULATION OF A VARIABLE COOLING SYSTEM FOR DOMESTIC WATER PURIFIER

SHAILESH S. SENGAR, PRATEEK MITTAL & SORABH

Manav Rachna International Institute of Research and Studies, Faridabad, Haryana, India

ABSTRACT

This paper presents a case; where in, a variable cooling system for the domestic purifier is modeled and tested under various temperature conditions. The variation in the thermal and flow parameter has been observed for the various temperature conditions and the required flow rate of water inside the heat exchanger is analyzed. This device is intended to be used as an attachment to the domestic water purifier, especially in India, as most of the regions face harsh and long summers in this part of the world. This analysis provides a strong foundation for the design and development of the cooling system. Also, the paper discusses the need for a small cooling system in the domestic house hold. The analysis has been done using ANSYS™ Fluent for the modeling and simulation purpose.

KEYWORDS: Variable Cooling System, Heat Exchanger, CFD & Thermo-Fluid Analysis

Received: Mar 09, 2018; **Accepted:** Mar 29, 2018; **Published:** Apr 17, 2018; **Paper Id.:** IJMPERDAPR2018166

1. INTRODUCTION

The original heat exchanger consisted of a rectangular thin plate, which is made up of Aluminum and a Copper pipe with semi- circular cross- section. The length of pipe is approximately 1 meter and radius of pipe cross- section is 5 centimeter. The plate size is 30cm×12.5cm. But, for the sake of simplification in our analysis, the bottom plate is removed and its temperature is given as boundary condition for the pipe. A three dimensional geometry of pipe was prepared in SOLIDWORKS as shown in Figure. 1.

Figure 1: Solid Works Model of the Tubing Geometry

The sharp edges near pipe bent are provided with fillet radius, to make the flow smooth. The geometry is named as *flat plate* for bottom, *curved plate* for top semicircular section, *inlet* and *outlet*. The working space inside the pipe is named interior. Once the geometry is made and named, it is imported to ANSYS ICEM-CFD for meshing. Tetra-dominated unstructured meshing is done for the *interior* geometry, whereas, the surfaces have triangular meshing as shown in Fig. 2 and 3. It has total no. of elements 676156 and 141795 nodes.

Figure 2: ICEM CFD Mesh

Figure 3: Magnified View of Mesh

Figure 4: Polyhedra Mesh

While solving the cases in FLUENT, this kind of mesh was creating problem of divergence. So, a different mesh type, called *Polyhedra* was implemented as shown in Fig. 3. Once the meshed geometry is prepared, it is imported into ANSYS FLUENT for solution. The geometry is scaled into meter from mm. 3D, and steady, pressure-based FLUENT solver is used. As in the heat exchanger process, heat transfer generally takes place via conduction and convection modes, and the *Radiation* is kept off while keeping the *Energy* solver on. The Reynolds no. was calculated for internal flow, and it came out to be greater than 4000, hence, the flow inside heat exchanger is turbulent. *Realizable $k - \epsilon$ turbulent model* is used for flow simulation with Enhanced-wall treatment for near wall. Pressure gradients and thermal effects are active.

2. MATERIALS

Two materials were used from FLUENT database viz. water for interior and copper for plates.

3. BOUNDARY CONDITIONS

For *flat plate* no-slip B.C. is provided for momentum, and -5°C (268 K) temperature is given, and also 3mm thickness is assigned to it. For *curved plate*, heat flux B.C. is taken to be zero to keep it insulated, while no-slip B.C is given for momentum. Its thickness is kept zero. The *outlet* is assigned pressure-outlet B.C. with zero gauges and back flow temperature is at NTP i.e. 300 K. For *inlet*, there are three cases for velocity inlet Boundary Condition. Different values of

mass flow rate are assigned for all cases separately, to get required value of temperature at outlet.

4. MODELING AND EXPERIMENTATION

There are three cases prepared for the required temperature at outlet and total temperature drop inside the heat exchanger.

Case 1: 20°C Temperature Drop

Inlet is given 35°C (308 K) temperature and for 0.01 kg/s mass flow rate, the outlet temperature came out to be 15°C. Hence, we obtained the required temperature drop of 20°C. Following figures 5 and 6 show contours of temperature and convective heat transfer coefficient, respectively for this case.

Figure 5: Contours of Static Temperature for 20°C Temperature Drop

Figure 6: Contours of Convective Heat Transfer Coefficient for 20°C Temperature Drop

Case 2: 15°C Temperature Drop

Inlet is given 35°C (308 K) temperature and for 0.04 kg/s mass flow rate, the outlet temperature came out to be 20°C. Hence, we obtained the required temperature drop of 15°C. Following figures 7 and 8 show the contours of temperature and convective heat transfer.

Figure 7: Contours of Static Temperature for 15°C Temperature Drop

Figure 8: Contours of Convective Heat Transfer Coefficient for 15°C Temperature Drop

Case 3: 10°C Temperature Drop

Inlet is given 35°C (308 K) temperature and for 0.1 kg/s mass flow rate, the outlet temperature came out to be 25°C. Hence, we obtained the required temperature drop of 10°C. Following figures 9 and 10 show contours of temperature and convective heat transfer coefficient, respectively for this case.

Figure 9: Contours of Convective Heat Transfer Coefficient for 10°C Temperature Drop

Figure 10: Contours of Convective Heat Transfer Coefficient for 10°C Temperature Drop

The axial velocity contour plot is not showing significant separation, so the time spent by any fluid particle inside the conduit shall not vary much from the calculated value, using the flow rate (see figures below 11 and 12).

Figure 11: Contours of Axial Velocity for Mass Flow Rate of 0.01 kg/s

Figure 12: Contours of Axial Velocity for Mass Flow Rate of 0.1 kg/s

5. RESULTS AND DISCUSSIONS

As the given flow is fully developed, the boundary layer thickness cannot change with the distance along the stream. Now, if the flow rate is increased, the boundary layer thickness will reduce, which consequently increases the temperature gradient, but it also depends upon the velocity. The following equation [1] depicts the relation between temperature drop and Prandtl no.

$$\rho A v C_p \Delta T = -k A \frac{\partial T}{\partial y}$$

$$\Delta T = -\frac{k}{\rho C_p v} \frac{\partial T}{\partial y}$$

The effective change in temperature drop is decided Prandtl No., which in this case, is greater than 1 ($Pr = 7.56$). So, the momentum diffusivity is seven times more effective than thermal diffusivity, so velocity has more impact here. Also, from the above equation, the temperature drop is inversely proportional to the velocity [2]. So, by increasing the velocity, the temperature drop decreases. The results obtained during the simulation are summarized below in Table 1 and 2.

Table 1: Results Obtained after Simulation

Quantity	Case 1	Case 2	Case 3
T- outlet (°C)	15	20	25
ΔT (°C)	20	15	10
h (W/m ² °C)	2065.7	5065.6	11353.2
Q (flow rate in ml/s)	10	40	100
ΔP (Pa)	650.51	4970.7	20875.9

Table 2: Flow Rates during Various Temperature Drops as Observed during the Simulation

Q (flow rate in ml/s)	10	15	20	40	100
T- outlet (°C)	15	17	18	20	25
ΔT (°C)	20	18	17	15	10

REFERENCES

1. Cengel Yunus A., Ghajar Afshin J., *Fundamentals of Heat and Mass Transfer*, McGraw Hill, 5th Edition, 2017.
2. Seshadri C.H., Patankar F.B., *Elements of Fluid Mechanics*, Prentice Hall (India), 1972.