Research the heat transfer simulation analysis of water heater pipeline flow channel system

Shu-Lung wang[1], Yi-Feng Chang[2], Ay-Su[3], Yi-Ting Wang[4]

[1]Department of Mechanical Engineering, Taoyuan Innovation Institute of Technology, Taoyuan Taiwan (R.O.C.)
[2,3]Department of Mechanical Engineering, Yuan Ze University, Taoyuan, Taiwan(R.O.C.)

Abstract— *This study is research of gas water heater cycling system with piping the heat transfer simulation analysis of water heater pipeline flow channel system the mold flow software of numerical calculation is FLUENT6.2 the geometric shape and computational grid for quarter bend, When the fluid passes through the shape changed flow field, under the effect of pressure distribution and geometric shape change of the flow field, there will be a recirculation zone. In the flow field of a quarter bend, To understand the structure of the flow field and flow field of temperature a analysis.*

Keywords— Gas Water Heater Cycling System, Quarter Bend, Flow Field, Numerical Calculation.

I. INTRODUCTION

In recent years, with the sharp rise of crude oil prices, the occurrence of the financial crisis, global warming, and climatic anomalies, various countries push energy saving and carbon reduction, and the importance of earth's water resources is relatively enhanced. As living standards are greatly improved, the water heater has become a requisite for life. A water heater is comprised of a burner, igniter, heat exchanger, valve body, and safety device. The heat is transmitted by heating cold water through the heat exchanger, in order to export hot water the design of a water heater circulation system using a high heat capacity medium for heat absorption (sinking) often results in heat flow blockage on the contact surface of any two objects. This contact thermal resistance has been discussed in depth in literature, e.g. Kenneth W. B. et al. [1], Miroslav Colic et al. [2], and R. L. D. Cane et al. [3].

[4]When the storage of fossil fuels become less and less, the application of solar energy gets more and more important. However, the solar thermal heating systems on the market are currently costly, heavy, and bulky. This thesis introduces the design of a novel solar collector, which unites the advantages of a flat plate collector and an evacuated tube collector to form a "flat plate evacuated solar collector". The research studies the feasibility of this kind of solar collector; the study methods include the numerical simulations of fluid flow associated with heat transfers and the experiments on a real flat plate evacuated solar collector. The results show that the efficiency of the flat plate evacuated solar collector lies between the evacuated tube collector and flat plate collector and this novel solar collector is worth further study.

This paper presents the design of a self-sufficient control system using model-based control method for a heat pump water heating (HPWH) system with a photovoltaic/thermal (PVT) evaporator. Using commercially available photovoltaic modules, making flow channel in the back to form a photovoltaic/thermal module and utilize the module as an evaporator of heat pump water heater. When the power needed to operate the compressor of heat pump water heater is supplied by a PVT module. Therefore, the system can operate normally achieve self-sufficiency that does not require external power. Then, the simulation model of photovoltaic/thermal evaporator and heat pump water heating system is designed and developed on the MATLAB/Simulink environment. Through actual measurement of environmental parameters for different weather conditions were simulated [5].

II. EXPERIMENTAL DESIGN AND METHOD

This study Discuss the heat transfer simulation analysis of water heater pipeline flow channel system.The application is simulated by FLUENT 6.2 CFD software, the water heater cold water inlet pipe is analyzed; within 30 cm of the position there is a leftward quarter bend and a horizontal straight pipe (where piping corrosion occurs), the scouring effect of turbulent flow at 20 cm away from the next forward horizontal quarter bend and at the inlet is apparently the primary cause of corrosion. While the position is at a relatively low temperature, high flow is one of the causes of corrosion.

III. EXPERIMENTAL RESULTS AND DISCUSSION

This study Discussion the heat transfer simulation analysis of the gas water heater pipeline flow channel system In this study, as the medium is in static state when the flow rate decreases, and the pitting corrosion rate is higher than that in high flow rate medium, the high flow rate can reduce the probability of pitting corrosion. Therefore, in analysis of corrosion, the effect of flow field speed in the circulating pipe of the flow field in the pipe wall is simulated by numerical calculation, and according to the aforesaid conditions. First, the effect of a quarter bend in the circulation system pipeline on the flow field pattern is discussed; in the numerical procedure, the mold flow software of numerical calculation is FLUENT6.2 the geometric shape and computational grid for quarter bend, the distribution of computational grids is as shown in Figure1 and the total number of grids is 67639. When the fluid passes through the shape changed flow field, under the effect of pressure distribution and geometric shape change of the flow field, there will be a recirculation zone. In the flow field of a quarter bend, when the fluid passes through the curved tube, due to the flow field inertia and centrifugal force, the fluid on the outer wall of curved tube is extruded, thus, the outside pressure is high. As the flow field on the inner wall is under the same effect, the pressure is lower than the outer wall, forming the pressure profile in Figure 2. As the pressure distribution is non-uniform in the curved tube, it results in pressure difference. Meanwhile, due to the pressure difference between inner and outer walls, the outer high pressure flow field generates a velocity increment normal to the wall surface. The closer to the inner tube wall, the faster the flow field is formed of velocity components under normal and tangential directions. Therefore, the flow field velocity vector pattern and distribution in Figures 3 and 4. are formed. The closer the flow field is to the downstream of the curved tube, the higher the velocity near the inner wall. Through the curved tube, due to the inertial effect, a reversed velocity is formed on the inner tube wall, in order to maintain mass conservation, and the vortex in Figure 5. is formed.

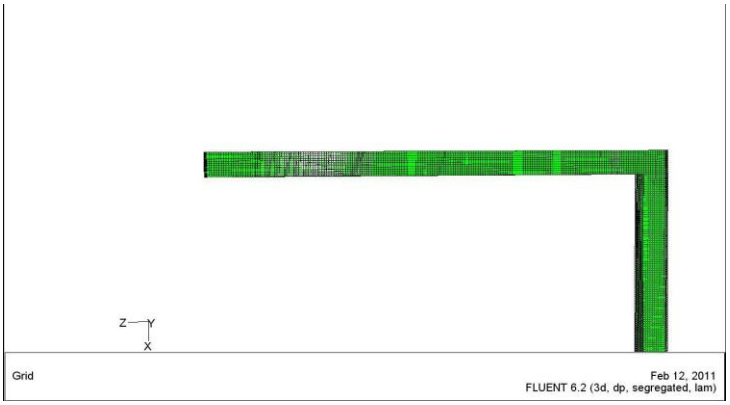

FIGURE 1 MESH OF DISTRIBUTION DIAGRAM

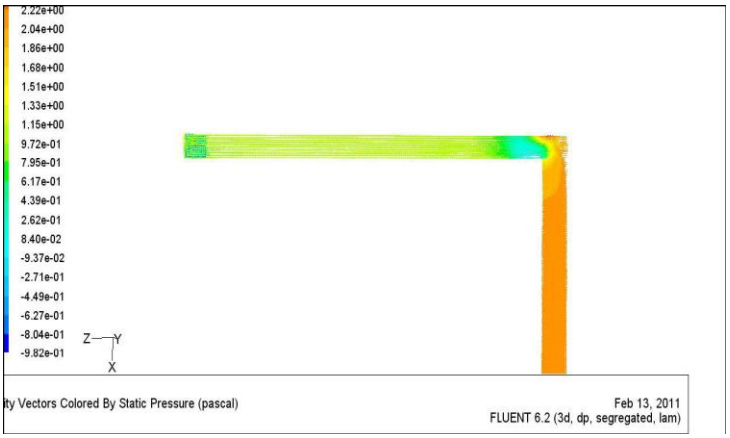

FIGURE 2. PATTERN OF FLOW FIELD PRESSURE VECTOR

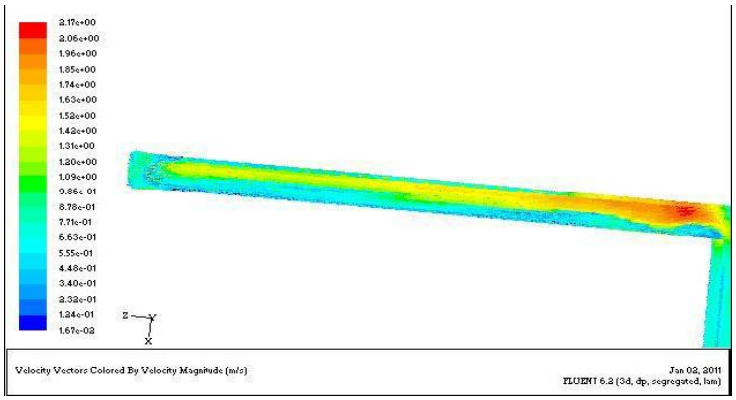

FIGURE 3. PATTERN OF FLOW FIELD VELOCITY VECTOR

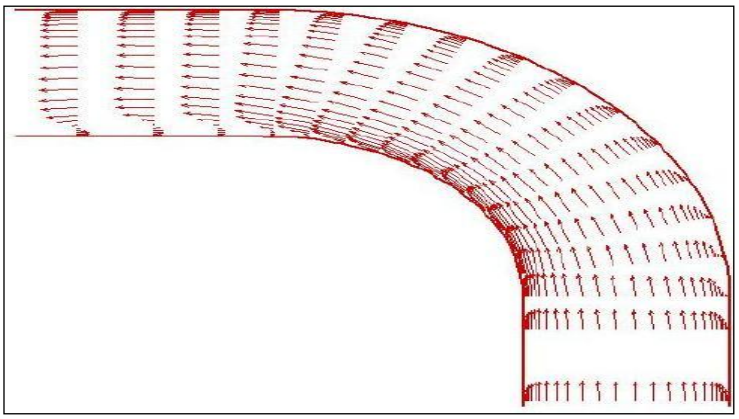

FIGURE 4. FLOW FIELD VELOCITY VECTOR DISTRIBUTION

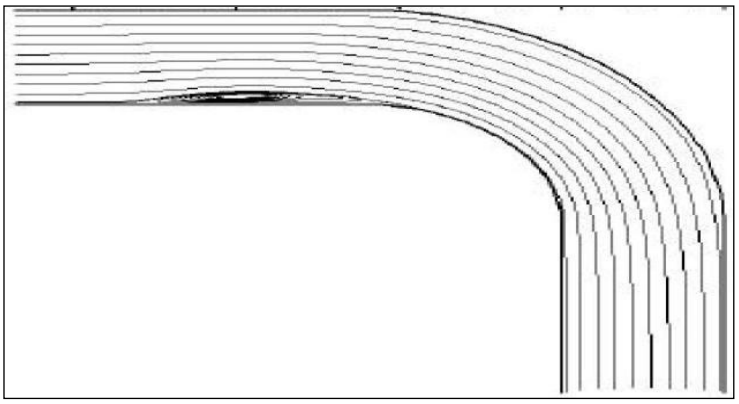

FIGURE 5. CURVED TUBE VORTEX DISTRIBUTION

IV. CONCLUSION

As the design of the pipeline exhaust at high temperature can remove the water vapor pressure inside the thermal circulation pipeline, clear the flow path, and is corrosion resistant.

REFERENCES

[1] Kenneth W. B. and Marianna A. B., "Laboratory studies on magnetic water treatment and their relationship to a possible mechanism for scale reduction". Desalination 109, pp. 131-148, 1997.

[2] Miroslav Colic, Dwain Morse., "The elusive mechanism of the magnetic memory of water". A: Physicochemical and Engineering Aspects 154, 167-174, 1999.

[3] R. L. D. Cane, S. B. Clemes and D. A. Forgas, "Heati n g recovery heat pump operating experiences, ASHRAE" Trans., 100(2), 165-172,1994.

[4] Chih-Chien Huang A Study on Flat Plate Evacuated Solar Collector Master's Thesis, Graduate School of Dayeh University, Mechanical Engineering, 2009.(inchinese)

[5] Bo-Jhih LinSelf-sufficiency Control System Design for Photovoltaic/Thermal (PVT) Assisted Heat Pump Water Heater Master's Thesis, Graduate School of Mechanical Engineering, Taipei technological University, electrical Mechanical Engineering, 2014.(in chinese)