Numerical Study of a Manifold Design for Heat Pipe Solar Collectors

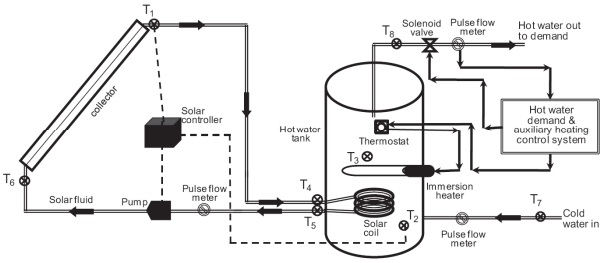
Feixiang Ren and Advisors: Junling Hu and Linfeng Zhang
Department of Mechanical Engineering
University of Bridgeport, Bridgeport, CT

Abstract

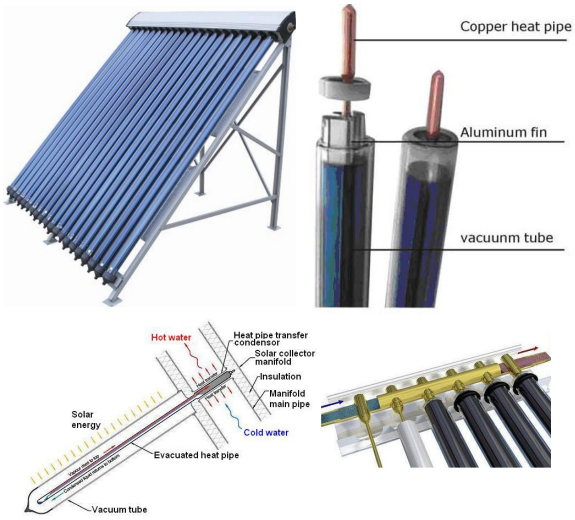
This project studies an active solar water heating system which uses glass evacuated tube solar collectors. The heat pipes inside the evacuated tubes transfer the absorbed solar energy to water in the manifold through the copper headers of heat pipes. A numerical model will be developed to simulate the heat transfer and fluid flow inside the manifold under different solar radiation flux levels and fluid flow rates. The simulation results provide understanding of the heat transfer and fluid flow patterns inside the manifold and will be used to help optimize the manifold design.

The Heat Pipe Collector System

Solar water heating is the most popular way of utilizing solar energy. It is widely used to supply hot waters for both households and larger organizations such as hotels, factories, hospitals, etc. A typical solar water heating system consists of a solar collector system, a hot water storage tank, a control unit, and a pump system.



Solar water heating system [1]



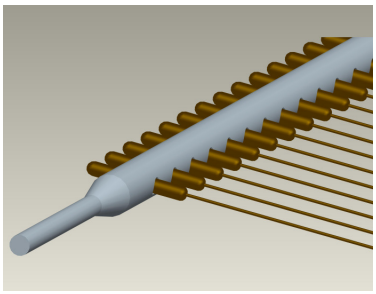
The heat pipe collector and manifold [2]

References

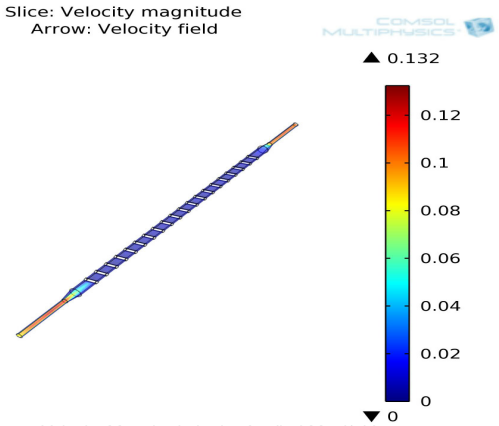
[1] L.M. Ayompe, et al, Validated TRNSYS model for forced circulation solar water heating systems with flat plate and heat pipe evacuated tube collectors, Applied Thermal Engineering 31 (2011) 1536-1542.
[2]http://www.seabirdsolar.com/scripts/heat_pipe_panel_photos.php

Simulation of Manifold

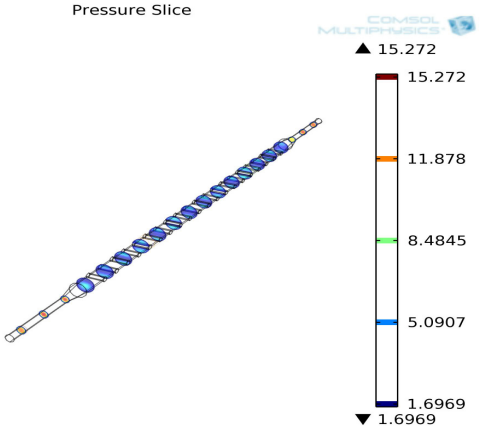
Manifold is the heat exchanger for a heat pipe solar collector. It houses the condensers of the heat pipes. The solar energy absorbed is transfer through heat pipe to the condensers where heat is transferred to the water flowing through the manifold. This project studied a copper manifold connected with twenty evacuated tubes through twenty heat pipe condensers. This heat transfer and fluid flow is simulated in COMSOL to study the effects of the operating and design parameters.



3D Geometry of the Manifold with the Heat Pipe



Velocity Magnitude in the Applied Manifold



Pressure Slices in the Applied Manifold

A fluid model is built in COMSOL to simulate the flow entering from the inlet, passing the twenty heat pipe condensers and exiting at the outlet. A heat transfer model will be added to in the future models to simulate the combined heat transfer and fluid flow in the manifold.