Design, Simulation, Fabrication, and Experimental Analysis of a Double Pipe Heat Exchanger

Mohammed Almomen, Jan Pinpin, and Lince Rumainum

Department of Engineering and Physics, University of Central Oklahoma, Edmond, OK 73034

Abstract

The design of heat-transfer equipment involves a trade-off between the two conflicting goals of low capital cost (high overall heat-transfer coefficient, small heat-transfer area) and low operating cost (small stream pressure drops). Optimal designs thus involve the constraints of capital and energy costs, which are constantly changing. In this project, we develop a computer interface similar to commercial computer software packages used for heat exchanger design, the underlying computer program calculates and optimize the size of heat exchangers within the constraints of capital and energy costs; particular emphasis is on the design of a double pipe heat exchanger. Heat transfer simulation using ANSYS® Fluent®, in addition to engineering experimentation were also conducted to confirm the efficiency and reliability of the proposed designs.

Introduction

Heat exchangers are devices that facilitate the exchange of heat between two fluids that are at different temperatures while keeping them from mixing with each other. Some examples and applications of the heat exchangers can be found in car radiators, air conditioners, heaters, and refrigerators to name a few. As for the project, the focus is mainly in a double pipe heat exchanger. There are two configurations of fluid flow in the double pipe heat exchanger, parallel and counter flow. The connection of the pipes in the double pipe heat exchanger can be classified into two configurations, a series connection, as shown in Fig. 2 and a parallel connection.

Deliverables

In this project the deliverables are divided into four sections:

1. Computer-based user interface,
2. Designing the heat exchanger,
3. Numerical simulation of heat transfer inside the heat exchanger, and
4. Construction and experimentation on the heat exchanger.

These four deliverables are essential to one another to compare the result for the given constrictions.

Computer-Based User Interface

The interface is designed to be user friendly. It allows anyone to use it as long as the necessary inputs are specified. The inputs for the user interface are listed below:

- ✓ Inlet and outlet temperature of the hot fluid,
- ✓ Inlet temperature of cold fluid,
- ✓ Mass flow rate of both fluids,
- ✓ Selections for both fluids,
- ✓ Selections for inner pipe,
- ✓ Maximum pressure inside both pipes,
- ✓ Configurations of the pipe, and the option for the fins.

The interface is shown in Fig. 1. Beside displaying the input and output, it also displays the information of what the users *should not* do (or an error message will appear) and also some information they need to know. After running the program, they will then be given the result which would give the least amount of number of stages with pressure less than the maximum pressure drop specified, which include:

- ✓ outlet temperature of the cold fluid,
- ✓ total length of the pipe,
- ✓ number of stages,

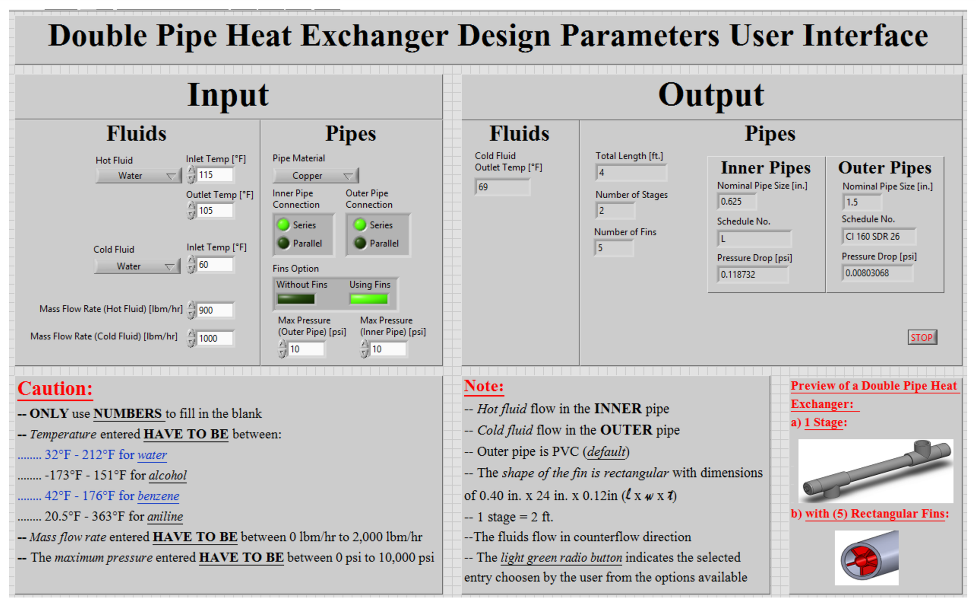

Fig 1: User Friendly Interface in LabVIEW with Sample Results

- ✓ standard pipe size (the Nominal Pipe Size and Schedule Number) of both pipes,
- ✓ pressure drop of both pipes, and
- ✓ number of fins.

The nominal pipe size and schedule number will make it easier for the user to build their own heat exchanger. In the end of the program a message box will appear to give the users the option to saved the output into a word document.

Designing the Heat Exchanger

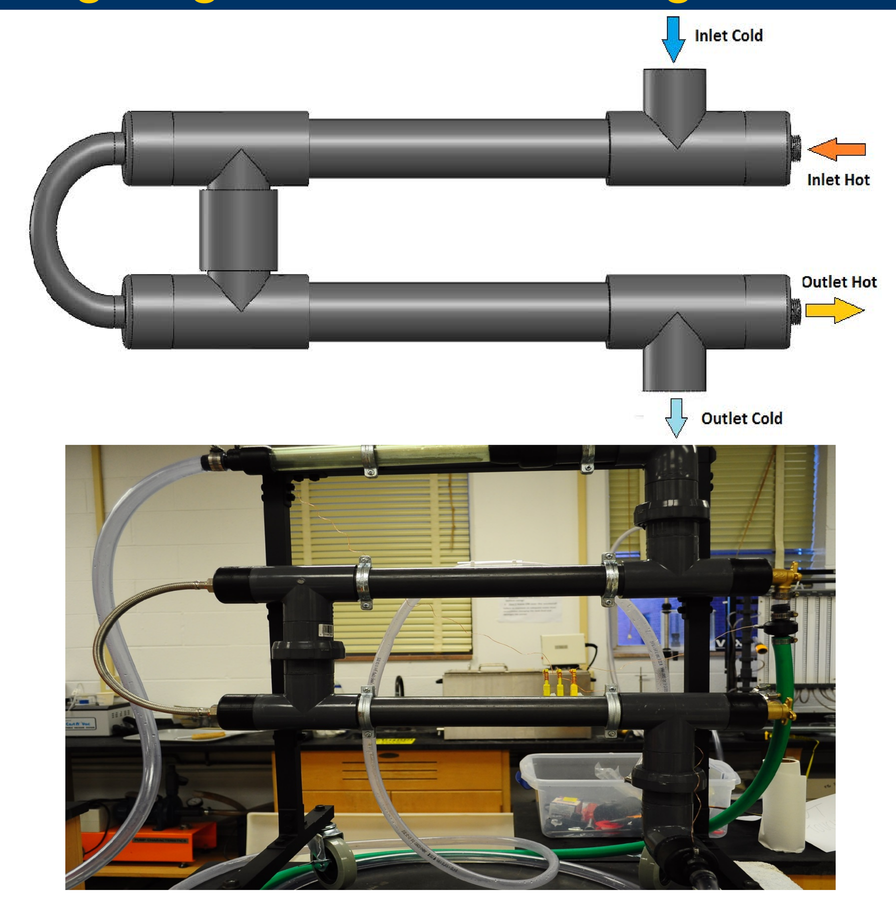

Fig. 2: Two Stages Design of the Double Pipe Heat Exchanger and the Physical Model

The final design is a series-series connection heat exchanger, where the fluid will flow through one stage at a time (shown in Fig. 2). Each stage of the inner pipe is connected using threaded return bend while each stage of the outer pipe will be connected with a union for easy maintenance purposes. The flow of the fluids will be counter-flow, where the fluids flow the opposite direction of each other. The details of the final design of the double pipe heat exchanger is shown in Fig. 3.

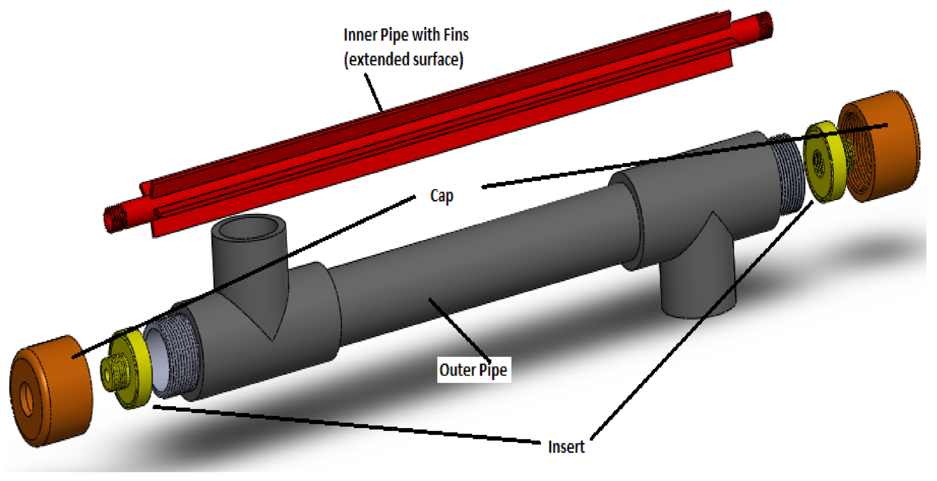

Fig. 3: Details of a 1 – Stage of the Double Pipe Heat Exchanger

ANSYS® Fluent®

The ANSYS Fluent is used to solve the conservation of mass equations, Navier-Stokes equation, and the energy equation. ANSYS is used to model heat transfer inside the heat exchanger. The double pipe heat exchanger design is transferred from the Solidworks software. The focus is on

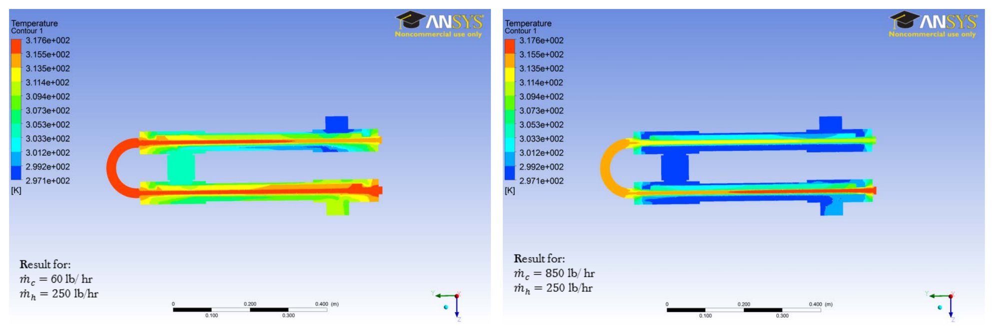

Fig. 4: The Cross Temperature of the Double Pipe Heat Exchanger

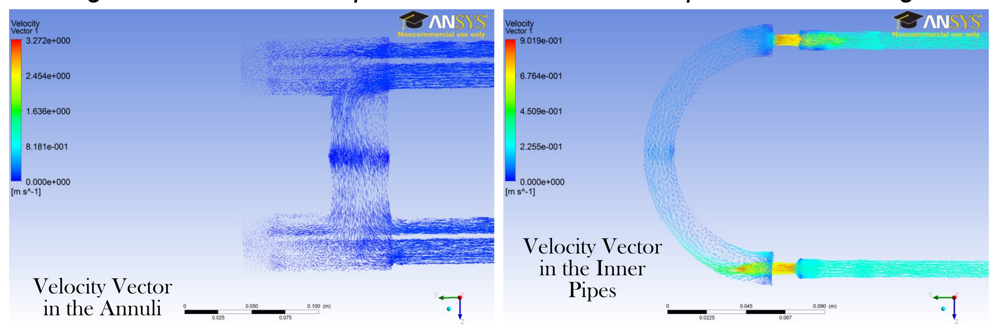

Fig. 5: The Velocity Vector of the Double Pipe Heat Exchanger

the temperature profile of the fluids throughout the pipes. Fig. 4 shows the fluid heat transfer inside the Solidworks model of the double pipe heat exchanger in Fig. 2 with fixed mass flow rate of the hot fluid and different mass flow rate of the cold fluid. The change in temperature of the inner fluid as well as the outer fluid are represented with the change in color, dark blue being the coldest and red as the hottest. The ANSYS Fluent also give the velocity vector of the fluid inside each pipe (shown in Fig. 5). After cross-checking the results of the ANSYS Fluent simulation with the design code used in LabVIEW and approving the results because of their similar behavior of results (the results are either both increases or both decreases) (shown in Fig. 6), then experiment is conducted.

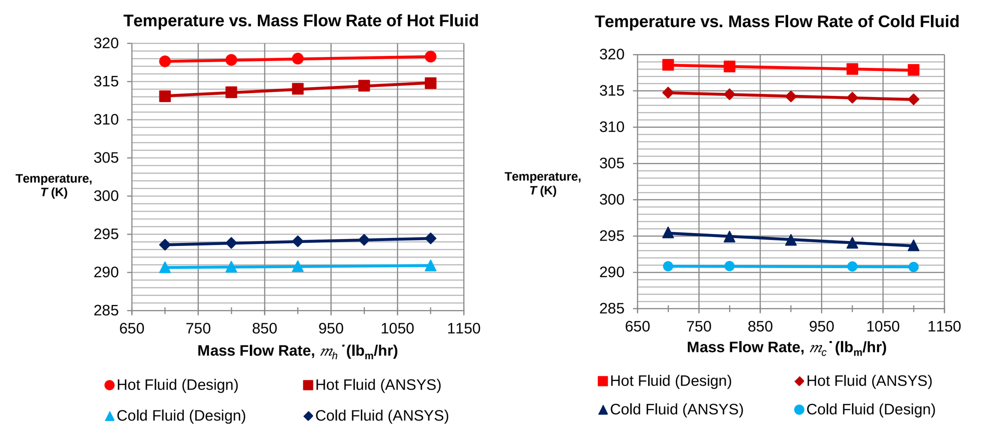

Fig. 6: Temperature vs. Mass Flow Rates Results of the Code Design and ANSYS® Fluent®

Experiment & Results

For the experimental procedure, thermocouples are used to measure the temperatures of the fluids. First a calibration charts for the thermocouple is built using DAQ 6008 from National Instruments, voltage amplifier, and LabVIEW software. The LabVIEW block diagram and circuit used for the data reading and calibration are shown in Fig. 8 and Fig. 9, where Fig. 10 is the experimental result for the calibration, which shows a proportional relationship between temperature and

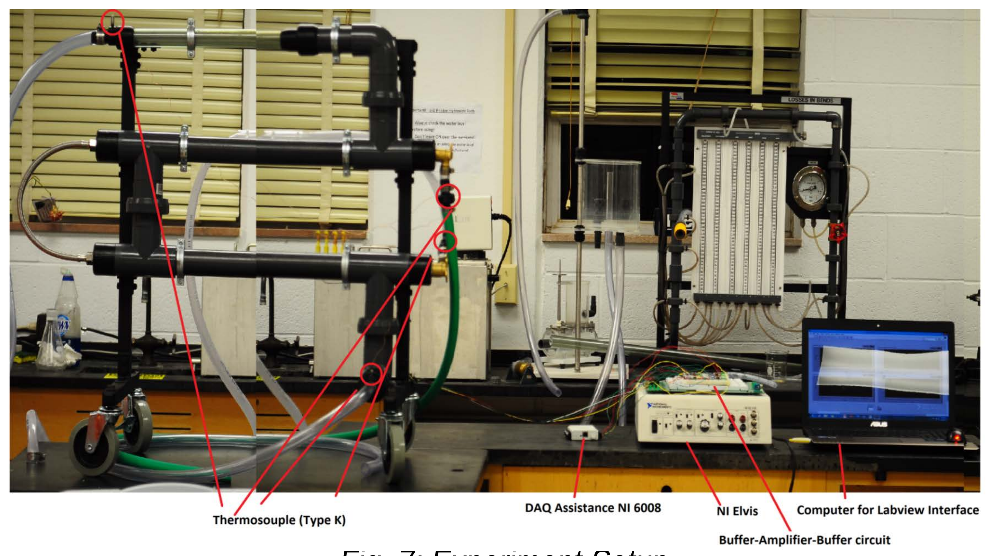

Fig. 7: Experiment Setup

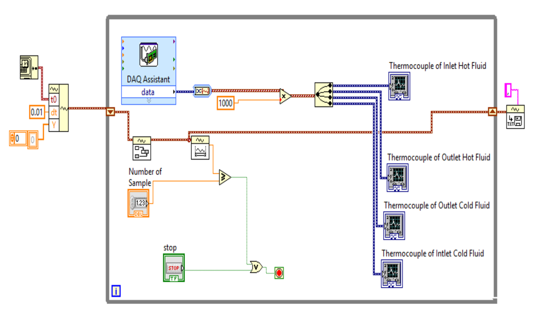

Fig. 8: LabVIEW Block Diagram

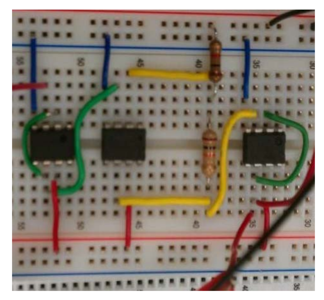

Fig. 9: Voltage amplifier Circuit

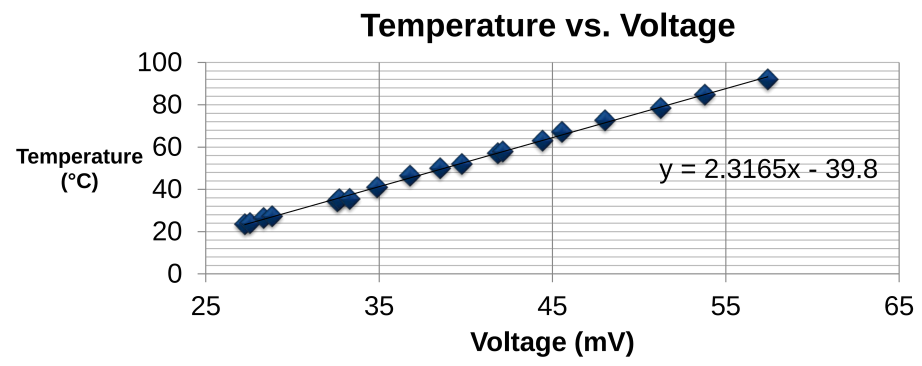

Fig. 10: Temperature vs. Voltage Results

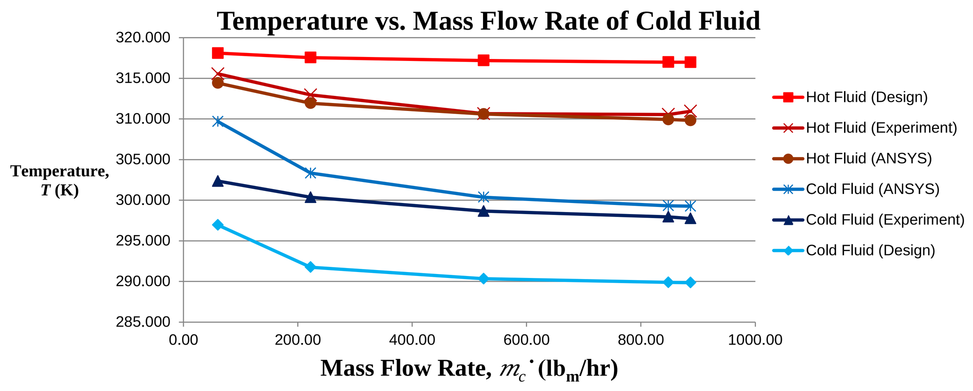

Fig. 11: User Interface Results

voltage. There are four thermocouples used in the experiment. Each pair of thermocouple is used for the inlet and outlet temperatures for one fluid. And after running an experiment for a fixed mass flow rate of hot fluid and changing the mass flow rate of cold fluid, the simulation and design code were modified accordingly to see the behavior of all three cases of the deliverables. Fig. 11 were constructed from those results. Even though it shows several some difference in the data but the behavior of each data is consistent.

Future Works

Further testing would be conducted with different mass flow rate for the cold fluid to further analyze the data. Also another experiment will be conducted with the cold fluid in fixed mass flow rate and the hot fluid in varying mass flow rate. A better thermocouple could be used that might give a more accurate result. For the cap and inserts, since the material currently used is 3D printed and somehow porous, a better alternative is under research in order to improve the sealing of the device.

References

Bergman, T. Incropera, F.P., Dewitt, D/P. and Lavine A., Fundamentals of Heat and Mass Transfer, 2007, 7 edition, pg 299-302.

Janna, W. (2010). Design of fluid thermal systems . (3rd ed.).

Kuppan, T. (2000). Heat exchanger design handbook. CRC Press.

Munson, B., Young, D., Okkiishi, T., & Huebsch, W. (2009). Fundamentals of fluid mechanics . (6th ed.). Wiley, John & Sons, Incorporated.

Shah, R., & Sekulic, D. (2003). Fundamentals of heat exchanger design. Wiley.

Serth, R. (2007). Process heat transfer: Principles and applications . Academic Press.

Contact Information

Dr. A. Ait-Moussa

Department of Engineering and Physics
Howell Hall 118A4
University of Central Oklahoma

100 N. University Drive
Edmond, OK 73034
USA

Tel: (405) 974-5293
Fax: (405) 974-3812
Email: aaitmoussa@uco.edu