International Journal of Heat and Mass Transfer 123 (2018) 89–96

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

# Experimental and numerical modeling of heat transfer in directed thermoplates

Imane Khalil *, Ryan Hayes, Quinn Pratt, Christopher Spitler, Daniel Codd

*Shiley-Marcos School of Engineering, 5998 Alaclá Park, University of San Diego, San Diego, CA 92110, United States*

ARTICLE INFO

*Article history:*
Received 12 December 2017
Received in revised form 21 February 2018
Accepted 22 February 2018

*Keywords:*
FLUENT
Computational fluid dynamics
Heat exchanger
Dimpleplate
Experiment validation
Reynolds number

ABSTRACT

We present three-dimensional numerical simulations to quantify the design specifications of a directed thermoplate expanded channel heat exchanger, also called dimpleplate. Parametric thermofluidic simulations were performed independently varying the number of spot welds, the diameter of the spot welds, and the thickness of the fluid channel within the laminar flow regime. Results from computational fluid dynamics simulations show an improvement in heat transfer is achieved under a variety of conditions: when the thermoplate has a relatively large cross-sectional area normal to the flow, a ratio of spot weld spacing to channel length of 0.2, and a ratio of the spot weld diameter with respect to channel width of 0.3. Experimental results performed to validate the model are also presented.



## 1. Introduction

Thermoplates are efficient heat transfer devices used in a variety of engineering practices. While many methods for manufacturing thermoplates have existed for some time, only recently have computational methods been used to diagnose the often complex flow present inside such devices [1–3]. Garg and Maji, and others have modeled the fluid flow and heat transfer sinusoidal channels with complex velocity fields, flow separation and re-attachments [4–8]. Thermoplate devices with regular, yet non-sinusoidal, spot welds are excellent candidates for parametric analysis using computational fluid dynamics simulations given their complex geometry-dependant flow structure.

The subject of this study is a thermoplate comprised of two sheets of Inconel 625 seam-welded together, with a pattern of spot welds distributed along the fluid direction as shown in Fig. 1. The space between the sheets is hydraulically expanded to a known thickness using a parallel plate guide fixture. The device uses a single fluid, in the liquid phase. This particular heat exchanger is a prototype test-section of a larger design intended for use in a specialized concentrated solar power receiver [9]. This application requires a compact, formable profile with high operating efficiency and minimal temperature gradient between the irradiated wall and heat transfer fluid. Preliminary designs utilize water as the working fluid, with laminar flowrates from 0.5 to 5.0 g/s, pressures ranging from 1 to 50 bar, and outlet temperatures of 80–250 °C.

As opposed to standard thermoplates which often fill an entire sheet with one or more fluids, we consider a directed flow through a single channel. The presence of periodic spot welds, and the complex curvature created during the inflation procedure result in flow structures which undermine the application of analytic solutions.

We present a case-study for this particular breed of thermoplate by varying critical components of the geometry such as the height of the channel, denoted $\delta$, the diameter of the spatially-periodic spot welds, denoted $D_s$, and the spacing between subsequent spot welds, given by $d_s$.

For fully developed laminar flow inside ducts, $Nu$ is constant and relatively low. Hence there are large wall-fluid temperature differences, $\Delta T$, between the outer wall and the fluid [10]. The motivation for our work is to enhance the heat transfer by increasing $Nu$.

First, we present details of the FLUENT Computational Fluid Dynamics (CFD) model used in this study, then we discuss the experiments performed to validate the FLUENT model, followed by the results from independently varying geometric parameters, and concluding with a discussion of the potential for an improved design.

## 2. Modeling

In this section we address the specifics of developing the model for the thermoplate. First we discuss the theory behind the flow

* Corresponding author.
*E-mail address:* ikhalil@sandiego.edu (I. Khalil).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.02.093
0017-9310/

**Nomenclature**

| | | | |
|---|---|---|---|
| $\delta$ | channel height (mm) | $Re$ | Reynolds number (based on $D_H$) |
| $l$ | channel length (mm) | $Nu$ | Nusselt number |
| $w$ | channel width (mm) | $P$ | pressure (Pa) |
| $D_s$ | spot weld diameter (mm) | $v$ | velocity (cm/s) |
| $d_s$ | spot weld spacing (mm) | $T$ | temperature (°C) |
| $D_H$ | hydraulic diameter (mm) | | |
| $\bar{h}$ | average heat transfer coefficient (W/m$^2$·°C) | | |

inside the thermoplate. We then address the construction of the thermoplate model using the commercial software packages SolidWorks (2016) and ANSYS-FLUENT (17.0) [11]. Due to the complexity of the part, the thermoplate geometry was constructed using the SolidWorks sheet-metal tools and then exported to the ANSYS environment for meshing, and finally FLUENT is used for thermofluidic calculation.

### 2.1. Theory

The thermoplate in question uses water in the liquid phase as the working fluid. The FLUENT setup included the pressure-based solver with the energy and incompressible laminar flow models. Therefore, the following equations are used in the FLUENT solver:

The continuity equation,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0, \tag{1}$$

the momentum equation,

$$\frac{\partial}{\partial t}(\rho \vec{U}) + \nabla \cdot (\rho \vec{U}\vec{U}) = -\nabla P + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g}, \tag{2}$$

the total energy equation,

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{U}(\rho E + \rho)) = \nabla \cdot \left(k \nabla T - h\vec{J} + (\bar{\bar{\tau}} \cdot \vec{U})\right), \tag{3}$$

where

$$E = h - \frac{P}{\rho} + \frac{U^2}{2}, \tag{4}$$

and enthalpy for incompressible flow is

$$h = \int_{T_{\text{ref}}}^{T} C_p dT + \frac{P}{\rho}, \tag{5}$$

Furthermore, Reynolds number is calculated,

$$Re = \frac{4\dot{m}}{\pi D_H \mu}. \tag{6}$$

In the equations above $t$ is time, $\rho$ is density, $\vec{U}$ is velocity, $P$ is pressure, $k$ is the thermal conductivity, $\vec{J}$ is the diffusion flux, $T$ is temperature and $T_{\text{ref}} = 298.15$ K, $\bar{\bar{\tau}}$ is the stress tensor, $C_p$ is the specific heat, $\mu$ is viscosity, and $\dot{m}$ is the mass flowrate.

The performance of the heat exchanger is calculated based on the temperature drop between the inlet and outlet. The performance is evaluated after varying the channel height, $\delta$, the spot weld diameter, $D_s$, and the weld spacing, $d_s$. The changes in geometry are generalized with the following ratios,

$$\text{Length Ratio} = \frac{d_s}{l} \quad \text{and} \quad \text{Width Ratio} = \frac{D_s}{w}. \tag{7}$$

Furthermore, the average hydraulic diameter, $D_H$, is calculated in SolidWorks. The details of the model geometry are discussed in the following section.

Only the fluid inside the channel is present for CFD simulation. The Inconel sheet is represented by a wall boundary condition. Furthermore, because the sheets are inflated from the mid-plane, we can apply a symmetry boundary-condition on the bottom face of the model, as shown in Fig. 3. Thus, the FLUENT model consists of only half of the fluid-domain.

The spots themselves are not modeled as they are part of the Inconel wall, and have no direct-contact with the fluid. Although

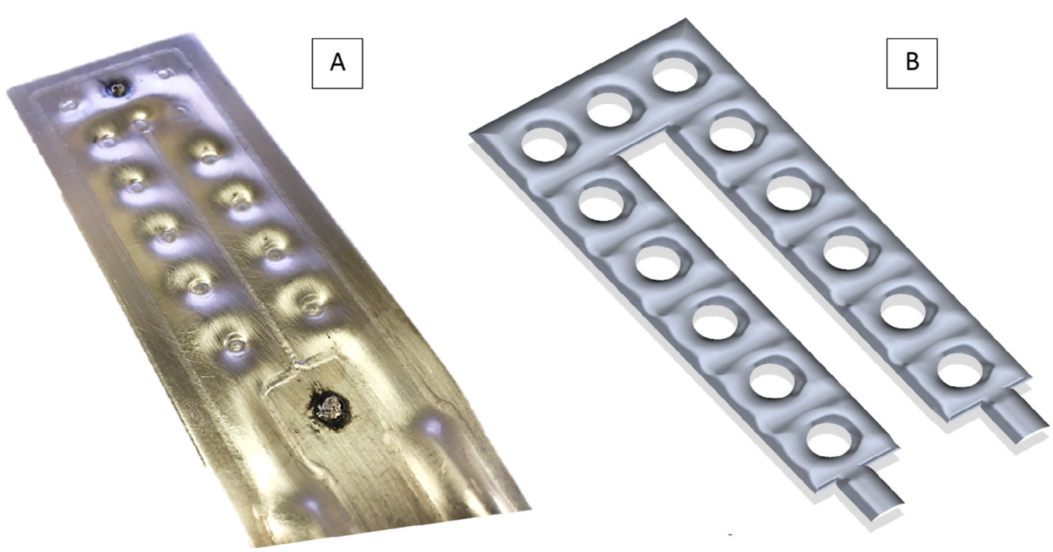


**Fig. 1.** Figure (A) shows the actual thermoplate used in experiments, the dark marks on the surface are weld locations of thermocouples and should be ignored. Figure (B) shows the SolidWorks model created to validate CFD simulations.