International Journal of Heat and Mass Transfer 65 (2013) 123–130

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Solution of an inverse convection problem by a predictor–corrector approach

Joseph R. VanderVeer, Yogesh Jaluria *

Department of Mechanical and Aerospace Engineering, Rutgers University, 98 Brett Rd., Piscataway, NJ 08854, United States

ARTICLE INFO

Article history:
Received 8 May 2013
Accepted 19 May 2013
Available online 28 June 2013

Keywords:
Inverse problems
Computational heat transfer
Convection

ABSTRACT

A predictor–corrector method for solving inverse convection problems has been developed and tested against both numerical and experimental data. The method was applied to the simple convection problem of a two-dimensional plume in a crossflow. Crossflow velocities up to 1.0 m/s. The plume was generated by electrically heating a copper plate to a temperature up to 425 K. The method attempts to predict both the source strength, and the source location, with a self imposed requirement on sampling and simulation data points. The samples and simulations required are found to be 5 and 2 respectively. Tests based on simulation alone indicate the methodology has a source strength prediction error of less than 1%, and less than 6% for source location. Experimental tests bring the overall error up to 5% for source strength and 10% for source location. This study indicates the potential of the methodology and demonstrates some of its limitations. The approach can be extended to applied areas such as environmental flows, room fires, and thermal management systems.

1. Introduction

Inverse heat transfer problems are often of greater interest to engineers than the associated forward problems. An example of this consideration is the temperature distribution on the wall of an optical fiber drawing furnace. The temperature at the center of the furnace is relatively easy to measure using an instrumented graphite rod. The temperature at the walls of the furnace is very difficult to measure due to the cylindrical shape of the furnace, inaccessibility, and high temperatures. Issa et al. [1] developed a regularization technique to use the center line temperature to predict the furnace wall temperature. Thus an inverse problem was solved and optimization was used to make the final result essentially unique.

Many methods are available to solve inverse problems. Several books [2–5] and articles [6–11] have been published on the subject. However few articles cover the inverse convection problems, and those that do often cover it as an aside to radiative or conduction problems. Of the few articles available, often a simple yet effective method is used to solve the inverse convection problem. That method is to iterate the simulation until it matches the known data, often requiring many iterations. This would be time consuming in large and complicated domains. One such example is the paper by Liu et al. [12], in which they use inverse convection methods to determine thermal profiles in a slot vented enclosure. They use this iterative approach requiring 20 to 30 iterations to achieve less than 1% error.

Knight et al. [13] used an iterative experimental–numerical data driven approach, coupled with a response surface, to predict the temperature and velocity of a jet in a crosswind. This method helps in predicting which data points need to be sampled in between iterations. They found that the methodology was able to predict jet velocity within experimental uncertainty and source temperature within 9%. However, the second stage of their approach over-predicted source temperature by as much as 23% [13,14].

The present study is a continuation of the work initiated by Knight et al. [13] and Ma et al. [14]. Preliminary work was covered in [15], with the sole purpose of being able to predict the source location as well as source strength. The added complexity of the unknown source location warranted an overall simplification of the problem from a jet in a crosswind to a plume in a crosswind. The overall goal is to predict, within acceptable error, both the location and source strength of the plume. For this methodology to be most useful, the number of sample points must be limited to a select few, and the number of simulations also must be kept small.

2. Experimental system

Most inverse solution methodologies require a fundamental understanding of the forward problem. This particular method is

---

* Corresponding author. Tel.: +1 732 445 3652; fax: +1 732 445 3124.
E-mail addresses: jaluria@soemail.rutgers.edu, jaluria@jove.rutgers.edu (Y. Jaluria).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.05.055

Nomenclature

b,m	model parameters
$C_1,C_2,C_{1\epsilon},C_\mu,\sigma_k,\sigma_\epsilon$	$k-\epsilon$ model coefficients
d	number of simulations
E	thermal energy
F	minimization function
I	turbulence intensity
k	turbulence kinetic energy
l	turbulence length scale
M_W	molecular weight
n	number of sample locations
P	pressure
P_{rt}	turbulent Prandtl number
T	temperature
t	time
U	free stream velocity
u,v	velocity components
X,Y	normalized coordinates
x,y	coordinates

Greek symbols

ϵ	turbulence dissipation rate
λ	thermal conductivity
μ	dynamic viscosity
μ_t	eddy viscosity
ϕ	normalized temperature $\phi=\frac{T-T_\infty}{T_S-T_\infty}$
ρ	density

Superscripts

$'$	instantaneous
$-$	ensemble averaged
$*$	predictor stage, alternative heat flux eqn.

Subscripts

∞	free stream
A,B	data set A,B
i,j,k	index
mod	modified
P	predicted
S	source

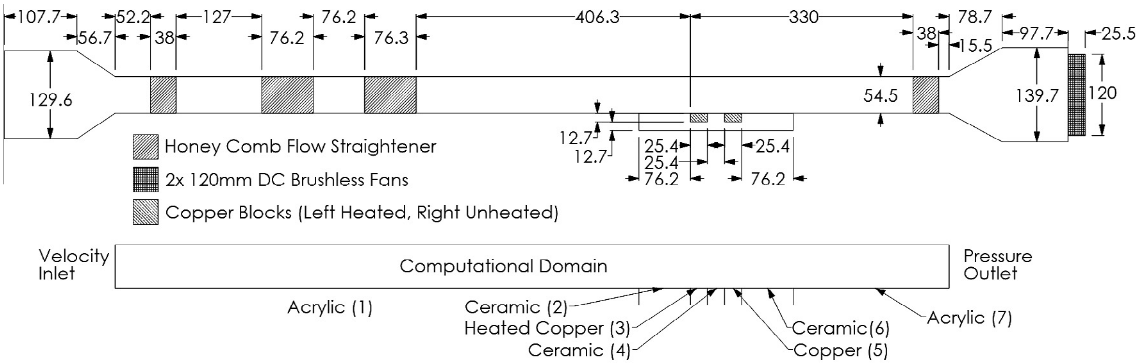

Fig. 1. Schematics of the wind tunnel and the computational domain [15].

no different. As previously described, the forward problem is a plume of heated air in a crosswind. A small region heated to a given temperature T_S is subjected to a flow of velocity U_∞, which is perpendicular to the direction of the vertical flow induced by thermal buoyancy.

The wind tunnel test section dimensions are $54.5\times305\times254$ mm, and can produce velocities in the range of $0-5.0$ m/s. Fig. 1 is a schematic of the experimental wind tunnel including the computational domain. All dimensions are in millimeters and the depth into the page is 305 mm. The heated section uses a resistance type electric heater to heat a copper block, encouraging a uniform heated surface, which is 25.4 mm wide. The maximum temperature of the heated section is limited to a maximum of 450 K due to material limitations of the wind tunnel. This creates a limitation to the maximum free stream velocity within the tunnel as the maximum heat input gets overwhelmed by the free stream above 1.0 m/s. That is to say the thermal plume is difficult to detect using the methods described here. The temperatures are measured using a K-type thermocouple mounted on a two-dimensional stage for motion within the plane of the figure. The X-direction is in the direction of the free stream, with zero at the upstream edge of the heated block. The Y-direction is in the direction of the plume, with zero at the surface of the heater.

A Pitot-static tube is used to determine the free stream velocity. The tube is attached to a NIST traceable differential pressure sensor from Omega (PX655–0.1DI), which has a full scale reading of 0.1 inches of water. The pressure sensor has an error of 0.05% of full scale reading. This results in a maximum of 3% error of the calculated velocity, which is at most 0.018 m/s for the applicable velocities.

The temperature is measured using a K-type thermocouple probe mounted on an X–Y traversing stage, and recorded using a National Instruments data acquisition board. Samples over several days yield a maximum difference of 7% outside the plume, and a maximum of 2% within the plume, indicating good repeatability.

3. Numerical simulations

The numerical simulations were carried out using the software package Ansys Fluent version 13[16]. The Navier–Stokes equations were solved using a three dimensional, steady state, realizable $k-\epsilon$ model with enhanced wall effects. The three-dimensional model is employed due to a limitation of Fluent, solid–solid conduction is not modeled at all in two-dimensional conjugate heat transfer problems, and the desire to ensure two dimensionality