International Journal of Heat and Mass Transfer 95 (2016) 1019–1025

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effect of fan-generated swirl on turbulent heat transfer and fluid flow in a pipe

J.M. Gorman [a,*], E.M. Sparrow [a], S. Ilamparuthi [a], W.J. Minkowycz [b]

[a] Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55455, USA
[b] Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago, IL 60607, USA

ARTICLE INFO

Article history:
Received 10 October 2015
Received in revised form 18 December 2015
Accepted 18 December 2015
Available online 15 January 2016

Keywords:
Fan generated swirl
Rotating fan
Turbulent pipe flow
Heat transfer
CFD
Numerical simulation

ABSTRACT

Fan-generated swirl can be characterized as chaotic and strongly mixed. It differs from controlled swirl with respect to those characteristics that tend to increase both the rate of heat transfer and the pressure drop. In the investigation described here, a rotating fan simulation, which takes full account of the rotation of the fan blades and of the true nature of the delivered flow, was implemented with high fidelity. The fan output is delivered directly to the inlet of a round pipe. It is believed that this is the first time that such a realistic approach has been used. The air delivered to the pipe has a temperature different from that of the pipe wall. The swirl component is shown to give rise to a significant enhancement of heat transfer, from a factor of two to 50% between $x/D = 0$ and 20. Similar increases of the wall shear were also found to result from the swirl. As a comparison case, a blower curve for the fan in question was used. Since blower curves omit the swirl component, only a uniform axial flow can be extracted. The swirl effects on heat transfer and shear stress were obtained by comparing the results for the actual rotating-fan-provided flow with those for the blower curve model. In the absence of swirl, the axial distributions of the local heat transfer coefficient and wall shear displayed a local undershoot, thereby supporting limited earlier reporting of such happenings in simpler situations.

1. Introduction

In a commonly encountered situation, fluid is ducted to a heat exchanger from a fan, blower, or pump that may be somewhat distant from the inlet of the exchanger. In that case, the attributes of the flow entering the exchanger are affected both by the swirling nature of the fan-produced flow and its subsequent interactions with the delivery duct. In another practical situation, the fan may be situated at the inlet of a pipe where heat transfer is taking place and blows its swirling discharge directly into the pipe. Both of these situations serve to motivate the research that is reported in this paper.

The specific situation to be dealt with here is an axial fan with *rotating blades* situated at the upstream end of a round pipe and blowing into the pipe without the presence of an intervening flow-modifying medium. The temperature of the inflowing fluid is different from the temperature of the pipe wall, thereby creating heat transfer.

There is a rich literature on the use of inserted swirl-producing objects to enhance heat transfer in pipes. Among the utilized methods are twisted-tape inserts [1–5] and stationary or fluid-driven rotating blades [6–10]. On the other hand, a literature search failed to produce any published articles which dealt with a fan with rotating blades discharging flow into a round pipe where heat transfer was occurring. The swirl produced by the aforementioned inserted objects differs from that created by moving, rotating fan blades.

The method used in the analysis and solution of the just-defined problem is numerical simulation. To facilitate the work, an actual axial fan was modeled without approximation and with strict account being taken of the rotation of the blades. Special focus was directed to the friction-related decay of the swirl that is produced by axial fans. There is evidence [11] that a swirling flow entering a pipe can be long lasting.

The investigation encompasses both fluid flow and heat transfer characteristics. For fluid flow, the circumferential variations of shear and pressure were identified as functions of the downstream distance from the rotating blades. In particular, the degree of frictional resistance attributable to rotation was determined. With regard to heat transfer, the circumferential variations of the heat flux and the circumferential average of the wall heat flux were obtained, both as a function of axial position. The extent of heat transfer enhancement due to rotation was carefully documented.

---

* Corresponding author. Tel.: +1 612 625 5502.
E-mail address: gorma157@umn.edu (J.M. Gorman).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.12.038
0017-9310/

Nomenclature

A	model constant
c_p	specific heat at constant pressure
D	pipe diameter
F_1, F_2	blending functions in the SST model
$\hat{h}$	circumferentially averaged heat transfer coefficient
k	thermal conductivity
L	pipe length
k_{turb}	turbulent thermal conductivity
Nu_D	circumferential averaged Nusselt number, $\hat{h}D/k$
p	pressure
P_k	production term for the turbulent kinetic energy
Pr_{tur}	turbulent Prandtl number
q	local heat flux
S	absolute value of the shear strain rate
T	temperature
T_{bulk}	fluid bulk temperature
T_{wall}	pipe wall temperature
t	time
u_i	velocity component in the i-direction
x_i	Cartesian coordinate
x	axial coordinate

Greek letters

α	SST model constant
β_1, β_2	SST model constants
ε	turbulence dissipation
θ	circumferential coordinate
κ	turbulent kinetic energy
μ	molecular or dynamic viscosity
μ_{turb}	turbulent viscosity
ν	kinematic viscosity
ρ	fluid density
σ	diffusion coefficient
ω	specific rate of turbulence dissipation

For comparison purposes, two cases characterized by non-swirling, uniformly distributed fluid inflows were investigated. One of these was based on the use of the blower curve for the particular fluid mover in question. The other case made use of the mass flow rate delivered by the actual rotating fan but envisioned that the flow entering the pipe has a uniform velocity profile. Comparisons among these cases enabled the effect of swirl to be viewed from different perspectives.

2. The physical situation

A schematic diagram of the physical situation to be considered here is presented in Fig. 1. As seen there, an axial fan is situated at the inlet of a round pipe. The fan draws air from the upstream space and discharges it directly into the pipe inlet. The air experiences frictional interaction with the pipe wall with the outcome that the fan-imparted swirl decays. The temperature of the flow entering the pipe is different from that of the pipe wall, so that heat transfer occurs. It is expected that the heat flux would vary both circumferentially and longitudinally.

The flow is intrinsically turbulent, so that a turbulence model is a necessary part of the solution methodology. It is expected that the turbulence level would be greatest in the fan discharge and would decrease in the downstream direction. It is also reasonable to expect that the flow pattern and the heat transfer would depend on the length of the pipe into which the fan discharges.

Three different situations are to be investigated. The primary problem is the one in which the fan discharges into the pipe. For comparison purposes, two relatively simpler situations will also be considered. One of these is based on applying the blower curve for the fan in question to provide a flow rate which is uniformly distributed across the pipe inlet. That flow does not contain swirl. A second comparison case is based on utilizing the flow rate delivered to the pipe inlet by the actual rotating fan. However, for this comparison case, it is assumed that that flow rate is uniformly distributed across the pipe inlet and that there is no swirl.

The solution methodology is based on modeling and subsequent numerical simulation. The flow provided to the pipe inlet by the rotating fan is auto-determined. For the other two cases, the flow rates are as described in the preceding paragraph. Since the flow rates cannot be arbitrarily specified, the Reynolds number cannot be arbitrarily varied as an independent parameter. The pipe length was varied parametrically and was the one prescribable independent variable.

3. Numerical modeling

In the considered physical situation, the fluid flow is three-dimensional, turbulent, and unsteady. The adopted simulation software is ANSYS CFX 15.0. This software is based on discretization by means of the finite-volume method. To ensure high accuracy, a mesh independence study was performed. The study involved three different meshes, respectively encompassing 25.5×10^6, 39.7×10^6, and 55.4×10^6 nodes. To quantify the dependence of the solution on the number of nodes, two independent quantities were chosen: (a) the mass flowrate passing through the system and (b) the heat transfer rate at a representative location. For the mass flowrate, the results corresponding to the three foregoing meshes are: 0.148, 0.157, and 0.160 kg/s. The

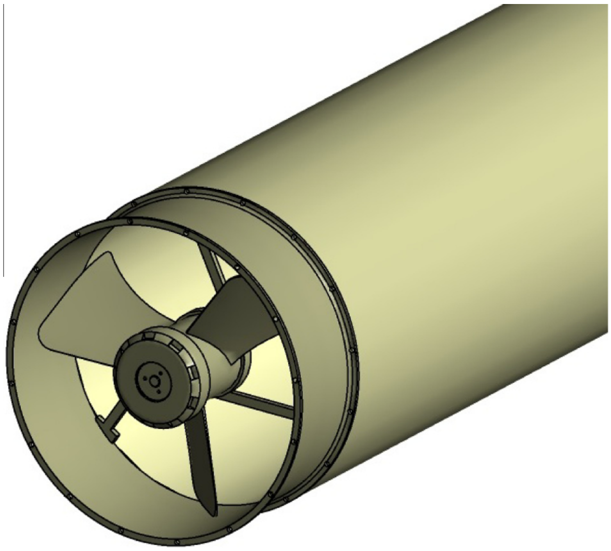

Fig. 1. Schematic diagram of the problem under consideration showing a rotating fan discharging swirling flow into a round pipe.