

MODELING OF TURBULENT FLOWS IN RECTANGULAR DUCTS OF CONSTANT SECTION USING OPENFOAM

RAQUEL FARIA^{1,2}, ALMERINDO D. FERREIRA², A. M. G. LOPES², ANTONIO C. M. SOUSA³

¹ Polytechnic Institute of Coimbra, ISEC, Department of Mechanical Engineering - Rua Pedro Nunes - Quinta da Nora, 3030-199 COIMBRA, Portugal, e-mail: rfaria@isec.pt, web: <http://www.isec.pt>

² ADAI-LAETA, Department of Mechanical Engineering, University of Coimbra - Polo II, 3030-788 COIMBRA, Portugal, e-mail: almerindo.ferreira,antonio.gameiro@dem.uc.pt, web: <http://www.uc.pt>

³ Department of Mechanical Engineering, University of New Brunswick - P.O. Box 4400, Fredericton, NB E3B 5A3, Canada, e-mail: asousa@unb.ca, web: <http://www.unb.ca/fredericton/engineering/depts/mechanical/people/sousa.html>

Keywords: OpenFOAM, Rectangular duct, Pressure coefficient, Friction coefficient, *k-w* SST turbulence model

Introduction

The increasing interest in turbulent flows in non-circular ducts, which are present in many important technical applications, such as heat exchangers, air conditioning systems and rotary machinery, among others, has led to a considerable research effort aiming the understanding and characterization of these flows. The most common configurations are the triangular and the rectangular cross-section shapes [1-2]

In engineering applications, it is quite common to consider the fully developed flow characteristics in non-circular ducts to be similar to those flowing on pipes of circular cross-section ([1-3]), with the cross-section being described by its hydraulic (or equivalent) diameter (D_h).

It is well established that, for fully developed flow, the pressure drops linearly in the stream wise direction, being pressure frequently presented in the form of the dimensionless pressure coefficient (C_p):

$$C_p = \frac{p - p_{ref}}{\frac{1}{2} \rho U^2} \quad (1)$$

where p (Pa) is the static pressure at the point of interest, p_{ref} (Pa) is the reference static pressure, ρ (kg/m^3) is the density of the fluid and U (m/s) is the axial mean velocity.

Assuming ducts of constant cross section, the pressure drop between two sections can be used to evaluate the mean friction coefficient [1]. Therefore, the Darcy's friction factor (C_f) can be used, which is defined by the following expression:

$$C_f = \frac{D_h \cdot \frac{\partial p}{\partial x}}{\frac{1}{2} \rho U^2} \quad (2)$$

being $\frac{\partial p}{\partial x}$ the pressure gradient in the axial direction.

The numerical simulations, performed within this work, were implemented in the open source OpenFOAM [4] CFD toolbox, being nowadays very popular in industrial engineering as well as in the scientific community. Several reasons led to the choice of this code, such as no limitations for parallel computing and being open for adaptation and development, bringing great flexibility and suitability for the users, contrarily to commercial computational fluid dynamics (CFD) codes.

This work is framed in the study of the applicability of Irwin [5] probes in incompressible flow through rectangular ducts and on its use under pressure gradient conditions. In the present work, simulations performed with OpenFOAM, and also with ANSYS CFX, are compared against experimental results.

Experimental Setup

In the present study, a rectangular duct with a constant width of 0.12 meters was used. A movable top wall permitted to vary the height, and four different aspect ratios (AR) were considered (1:4.00, 1:2.67, 1:2.00 and 1:1.03).

Figure 1 schematically depicts the experimental apparatus. A centrifugal fan was used to move the air into the 5 meters long plywood rectangular duct. The duct has an initial flow developing section followed by the so-called test section; both sections have the same length (2.5 m). Fifteen pressure taps were initially considered in the test section, aiming the selection of 4 averaging locations. On each averaging location, the mean pressure was obtained by using 4 pressure taps (one in each side of the duct, as indicated by the dashed lines in Figure 1).

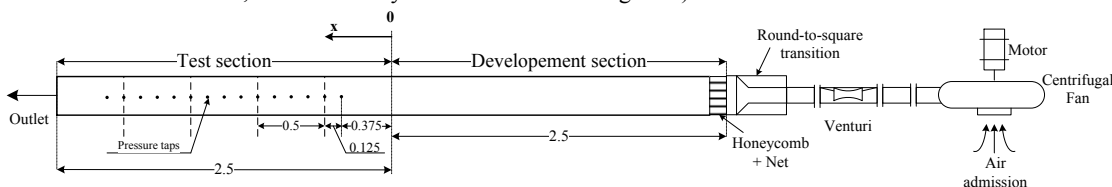


Figure 1: Schematic of experimental apparatus – Top View (Dimensions in meters)

Numerical Setup

For the OpenFOAM [4] simulations, several structured grids were tested (generated using the blockMesh utility) with increasing resolution to evaluate the mesh dependency. The standard solver simpleFOAM was employed, assuming steady state and incompressible conditions, to solve the RANS equations based on the finite-volume (FVM) discretization method. The maximum allowable absolute residual for the linear solvers was 1E-05. The k- ω SST (Shear Stress Transport) [6] turbulence model was employed with the automatic near-wall treatment [7]. A turbulence intensity of 5% was considered at the inlet.

The following boundary conditions were applied at the inlet: zero gradient for pressure and fixed values for the other properties. At the outlet, zero gradient was assumed for all, except pressure, which was set equal to experimental value. Symmetry conditions were imposed in two of the domain boundaries, due to the geometry (rectangular duct) of the domain (only a quarter was simulated). Walls were treated as smooth, with a no-slip condition. Similar conditions and identical mesh sizes were adopted for the ANSYS CFX simulation.

Results

The results discussed here are only for the duct aspect ratio 1:2.00, which is representative of all other cases. Figure 2 depicts the C_p distribution along the test section. The experimental results are the mean values calculated at each the averaging section, and the computational distributions are a result from a mesh sensitivity study, performed in OpenFOAM (Figure 2.a)). Then a comparison between experimental data, CFX and OpenFOAM (mesh with best agreement, $y^+=56.7$) can be established (Figure 2.b)).

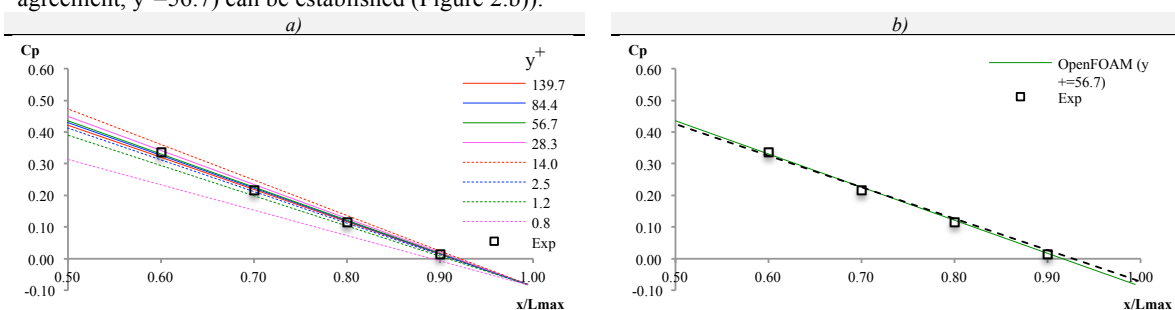


Figure 2: Pressure coefficient distribution. a) Mesh study in OpenFOAM (various values of y^+) and b) Experimental results against predicted values by OpenFOAM and CFX

In fully developed flow conditions, the relation between pressure gradient and friction coefficient is obtained through a force balance. Assuming that pressure drops linearly, a constant value is obtained in the test section. Table 1 shows the C_f values and the respective deviation of computational predictions.

Table 1: Friction coefficient predicted and deviations from experimental results

	C_f	Deviation (%)
<i>Experimental</i>	0.0173	-
<i>OpenFOAM</i>	0.0168	2.84
<i>CFX</i>	0.0159	8.20

Acknowledgements

The authors thank all those involved in the organisation of OFW11 and to all the contributors that will enrich this event.

References

- [1] J.P. Hartnett, J.C.Y. Koh, and S.T. McComas, "A Comparison of Predicted and Measured Friction Factors for Turbulent Flow Through Rectangular Ducts," *Journal of Heat Transfer*, vol. C 84, pp. 82-88, February 1962.
- [2] A. Melling and J. H. Whitelaw, "Turbulent flow in a rectangular duct," *Journal of Fluid Mechanics*, vol. 78, no. Part 2, pp. 289-315, 1976.
- [3] S. Ahmed and E. Brundrett, "Turbulent flow in non-circular ducts. Part 1 - Mean flow properties in the developing region of a square duct," *International Journal of Heat and Mass Transfer*, vol. 14, pp. 365-375, 1971.
- [4] OpenCFD, OpenFOAM: The Open Source CFD Toolbox. User Guide Version 3.0.1, OpenCFD Limited, , Dec. 2015.
- [5] H.P.A.H. Irwin, "A simple omnidirectional sensor for wind-tunnel studies of pedestrian-level winds," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 7, pp. 219-239, 1981.
- [6] F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, pp. 1598-1605, August 1994.
- [7] F.R. Menter and T.E. Esch, "Elements of Industrial Heat Transfer Predictions", *Keynote lecture at the 16th Brazilian Congress of Mechanical Engineering*, Brazil, 2001.