

# **An Evaluation of Computational Fluid Dynamics for Modelling Buoyancy- Driven Displacement Ventilation**

Malcolm J. Cook BSc. MSc.

This thesis is submitted to De  
Montfort University, Leicester, in  
partial fulfilment of the requirements  
for the degree of Doctor of  
Philosophy

August 1998

*To Mom and Dad who have always encouraged  
me and supported my work*

*“Only after the last tree has been cut down,  
Only after the last river has been poisoned,  
Only after the last fish has been caught,  
Only then will you find that money cannot be eaten”*

*Cree Indian Prophecy*

## Abstract

Environmental concerns and demands for energy efficiency have prompted building designers to reconsider natural ventilation rather than mechanical ventilation and air conditioning. Natural ventilation can be brought about either by wind pressure or by buoyancy forces generated by temperature differences. This research addresses buoyancy-driven flows in a displacement ventilation regime.

Buoyancy-driven displacement ventilation occurs when thermal plumes, which form above heat sources such as occupants and electrical equipment, cause warm, buoyant air to accumulate in the upper part of a space forming a stratified layer. This layer drives a flow out of openings at high level which then draws in fresher air through low level openings. The design of such ventilation strategies must ensure that the layer of stratified air is able to drive a sufficient airflow rate and that the lower level of the stratification remains above the occupants' breathing zone.

Computer models can be used to assist with the design of naturally ventilated buildings. One such tool which has seen increased use with the advent of faster, more affordable, desk-top computers, is *Computational Fluid Dynamics (CFD)*. CFD is a detailed airflow modelling technique which solves the governing equations of air motion to give predictions of, primarily, pressure, velocity and temperature at many locations throughout the geometry under consideration. However, literature to-date points to short-comings in the capabilities of CFD for modelling buoyancy-driven flows.

The aim of this research was to evaluate the accuracy with which CFD is able to model buoyancy-driven flows. This has been done by defining three simple benchmark cases for which both analytical results and salt bath modelling measurements exist. These fundamentally different modelling techniques enable two distinct mechanisms for evaluation. The benchmarks comprise rectangular spaces with point and line sources of buoyancy (heat input) on the floor and openings to the outside air in the top and bottom. The CFD simulations were carried out using the code CFX which comprises solution and modelling techniques (such as turbulence modelling) which are typical of many commercially available CFD programs.

The program gave favourable predictions for the level of the horizontal interface between the warm and cool air, the temperature change across the interface, and the air change rates and flow patterns. Some discrepancies existed in modelling the buoyant plumes produced above the heat sources when using the standard  $k - \varepsilon$  turbulence model. These were reduced when employing the more recent  $k - \varepsilon$  model based on renormalisation group theory.

The work has shown that careful control of the solution process is needed when modelling buoyancy-driven flows and that care is needed when specifying the conditions at the boundaries separating internal and external air. The work offers guidance to others wishing to use the benchmarks and suggests how these techniques could be used for modelling buoyancy-driven flow regimes in real buildings.

# Acknowledgements

This research was supported with funding from De Montfort University, Leicester, and was undertaken within the Environmental Computer Aided Design and Performance (ECADAP) group, and more recently the Institute of Energy and Sustainable Development (IESD).

I would like to thank my director of studies, Prof. Kevin Lomas (De Montfort University) whose support, sound guidance, and meticulous attention to detail throughout this research has been greatly appreciated. I would also like to thank my second supervisors Prof. Andrew Howarth (University of Nottingham) and Dr. Martin Crane (De Montfort University) whose complimentary backgrounds ensured a varied and balanced approach to my research. I am also indebted to Dr. Geoff Whittle (Simulation Technology) and Prof. Paul Linden (University of Cambridge and University of San Diego, California) for their enlightening discussions and for spending many hours advising on the direction of the research and later checking the accuracy of my work.

I would like to thank all of the staff at the IESD for their moral support and encouragement, and for their assistance with IT matters.

It is difficult for me to find words which adequately express my feelings of thanks towards my wife Lisa, whose incredible patience and unfailing loving support, has, without doubt, ensured the completion of this thesis.

# Contents

Abstract	iii
Acknowledgements	iv
Contents	v
Nomenclature	x
Glossary and Abbreviations	xiv
<b>1. Introduction</b>	<b>1</b>
1.1 Preamble	2
1.2 Aims of the Research	4
1.3 Research Methodology and Thesis Content	4
<b>2. Natural Ventilation and Computational Fluid Dynamics</b>	<b>6</b>
2.1 Preamble	7
2.2 The Need for Ventilation	7
2.3 Ventilation - A Brief Historical Review	8
2.4 Ventilation - Some Modern Practice Examples	11
2.5 Modelling Natural Ventilation	15
2.5.1 Empirical Air Tightness Method	16
2.5.2 Simplified Theoretical Models	16
2.5.3 Zonal Models	16
2.5.4 Salt Bath Modelling	18
2.5.5 Plume Theory	19
2.5.6 Wind Tunnel Testing	19
2.5.7 Computational Fluid Dynamics	20
2.5.8 Advantages and Disadvantages of each Modelling Technique	21
2.6 Use of Computational Fluid Dynamics for Modelling Building Airflows	23
2.7 Selecting CFD Software	27
2.8 Summary	27
<b>3. The Theory and Salt Bath Modelling of Buoyancy-Driven Displacement Ventilation</b>	<b>29</b>
3.1 Preamble	30
3.2 Buoyancy Sources	30
3.3 Qualitative Description of Buoyancy-Driven Displacement Ventilation	32
3.4 Governing Equations for a Plume Emanating from a Point Source of Buoyancy	34

3.5	Quantitative Description of Buoyancy-Driven Displacement Ventilation	37
3.5.1	Flow from a Point Source of Buoyancy	37
3.5.2	Flow from a Line Source of Buoyancy	40
3.6	Salt Bath Modelling	41
3.7	Validity of Salt Bath Models	46
3.8	Methods for Evaluating the CFD Code	48
3.9	Summary	50
<b>4.</b>	<b>Mathematical Models of CFX</b>	<b>51</b>
4.1	Preamble	52
4.2	The Governing Equations of CFD	52
4.3	Heat Transfer	55
4.4	Buoyancy	55
4.5	Turbulence Modelling	56
4.5.1	Background	56
4.5.2	Mathematical Description of Turbulence Using Reynolds Averaging	57
4.5.3	Eddy Viscosity Models	58
4.5.4	The Standard $k - \varepsilon$ Model in CFX	61
4.5.5	The RNG $k - \varepsilon$ Model in CFX	62
4.5.6	Second Order Closure Models	63
4.6	Mathematical Description of Boundaries	64
4.6.1	Wall Boundaries	64
4.6.1.1	Wall functions for velocity	65
4.6.1.2	Wall functions for enthalpy	67
4.6.2	Pressure Boundaries	67
4.6.2.1	Inflow through a pressure boundary	68
4.6.2.2	Outflow through a pressure boundary	68
4.6.2.3	Other notes on pressure boundaries	68
4.6.3	Symmetry Plane Boundaries	68
4.6.4	Inter Block Boundaries	69
4.7	Fluid Properties	69
4.8	Summary	70
<b>5.</b>	<b>Two Dimensional Modelling of Buoyancy-Driven Displacement Ventilation</b>	<b>71</b>
5.1	Preamble	72
5.2	Definition of Benchmark 1	72
5.3	Initial Simulations	73

5.3.1	Setting Boundary Conditions	73
5.3.2	Enthalpy and Buoyancy Reference Temperatures	75
5.3.3	Specification of Fluid Properties	76
5.3.4	Mesh Definition	76
5.3.5	Convergence Criteria	77
5.3.6	Obtaining a Solution	78
5.4	Analysis of Benchmark 1 Results	81
5.4.1	Discussion of the Flow Field	81
5.4.2	Vertical Density Profile	85
5.4.3	Interface Properties	86
5.4.4	Plume Behaviour	87
5.5	Mesh Investigation	92
5.6	Effect of Turbulence Model	95
5.7	Effect of Sizes and Positions of Openings	100
5.8	Effect of Space Dimensions	102
5.9	Effect of Source Strength on the Flow	105
5.10	Investigation of Exterior Space Modelled	108
5.11	Summary	111
<b>6.</b>	<b>Three Dimensional Modelling of Buoyancy-Driven Displacement Ventilation: Line Source</b>	<b>113</b>
6.1	Preamble	114
6.2	Definition of Benchmark 2	115
6.3	Representing Benchmark 2 in CFX	116
6.4	Analysis of Benchmark 2 Results	118
6.4.1	Initial Simulations	118
6.4.2	Analysis of the Displacement Flow Pattern in Benchmark 2	119
6.4.3	Mesh Investigation	123
6.4.4	Plume Properties	123
6.4.4.1	Plume Entrainment	123
6.4.4.2	Volume Flux in the Plume	124
6.4.4.3	Buoyancy in the Plume	126
6.4.5	Effect of Changes in Source Strength	127
6.4.5.1	Flow Pattern	127
6.4.5.2	Volume Flux in the Plume	129
6.4.5.3	Buoyancy in the Plume	129
6.4.6	Effect of Changes in Opening Area	129
6.5	Comparison of Benchmarks 1 and 2	132
6.6	Summary	134
<b>7.</b>	<b>Three Dimensional Modelling of Buoyancy-Driven Displacement Ventilation: Point Source</b>	<b>135</b>

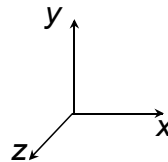
7.1	Preamble	136
7.2	Definition of Benchmark 3	136
7.3	Representing Benchmark 3 in CFX	136
7.4	Analysis of Benchmark 3 Results	137
7.4.1	Initial Simulations	137
7.4.2	Analysis of the Displacement Flow Pattern in Benchmark 3	137
7.4.3	Plume Properties	142
7.4.3.1	Plume Entrainment	142
7.4.3.2	Volume Flux in the Plume	143
7.4.3.3	Buoyancy in the Plume	145
7.4.4	Effect of Changes in Source Strength	145
7.4.4.1	Flow Pattern	145
7.4.4.2	Volume Flux in the Plume	148
7.4.4.3	Buoyancy in the Plume	148
7.4.5	Effect of Changes in Opening Area	150
7.5	Summary	150
<b>8.</b>	<b>Discussion of Results and Suggestions for Further Work</b>	<b>152</b>
8.1	Preamble	153
8.2	Guidance on How to Model the Benchmark Cases	153
8.2.1	Boundary Conditions	153
8.2.2	Mesh Definition	155
8.2.3	Convergence Criteria	157
8.2.4	Obtaining a Solution	158
8.3	Accuracy of the Benchmark Results	160
8.3.1	Two Dimensional Modelling	163
8.3.2	Flow Pattern	163
8.3.3	Plume Modelling	165
8.3.4	Interface Parameters and Effect of Opening Size	168
8.3.5	Turbulence Modelling	172
8.3.6	Effect of Source Strength	177
8.4	Application to Real Buildings	179
8.4.1	Guidance	179
8.4.2	Reliability of Results	182
8.4.3	Application Example	183
8.5	Suggestions for Further Work	186
8.5.1	Current Benchmarks	186
8.5.2	New Benchmarks	187
<b>9.</b>	<b>Conclusions</b>	<b>189</b>
9.1	General Conclusions	190
9.2	The Benchmarks	190
9.3	Modelling Buoyancy-Driven Flows	191

9.4 Accuracy and Reliability	192
9.5 Closing Remarks	193
<b>References and Bibliography</b>	<b>194</b>
<b>Appendix A Selecting CFD Software</b>	
<b>Appendix B Additional Plume Theory</b>	
<b>Appendix C Tensor Notation</b>	
<b>Appendix D Solving the Governing Equations in CFX</b>	

## Nomenclature

(-)	dimensionless variable
$A^*$	effective opening area ( $m^2$ )
$Ar$	Archimedes number (-)
$a_u$	total area of upper openings ( $m^2$ )
$a_l$	total area of lower openings ( $m^2$ )
$B$	strength of buoyancy source ( $m^4/s^3$ )
$C$	empirical constant in point source plumes (-)
$C_1, C_2, C_3, C_\mu$	turbulence model constants (-)
$C_d$	discharge coefficient (-)
$C_p$	specific heat capacity (J/KgK)
$c$	momentum theorem constant (-)
$D$	empirical constant in line source plumes (-)
$D_\phi$	diffusivity ( $\Gamma_\phi/\rho$ ), ( $m^2/s$ )
$d_f$	length scale for false time-steps (m)
$E, E_{He}$	wall function constants (-)
$G'$	plume buoyancy ( $m/s^2$ )
$G'_H$	hypothetical value of $G_T$ at $y = H$ ( $m/s^2$ )
$g$	acceleration due to gravity ( $m/s^2$ )
$g'$	reduced gravity or buoyancy ( $m/s^2$ )
$g'_h$	change in buoyancy across the interface, or stratification strength ( $m/s^2$ )
$H$	height of space (m)
$He$	total enthalpy (J/Kg)
$He^+$	normalised enthalpy (-)
$He_{ref}$	enthalpy reference value (J/Kg)
$He_{res}$	enthalpy residual (Watts)
$He_w$	enthalpy at wall (J/Kg)
$h$	height of interface (m)
$he$	static enthalpy (J/Kg)

$J_{He}$	enthalpy flux at wall ( $J/m^2s$ )
$k$	turbulent kinetic energy ( $m^2/s^2$ )
$l$	length scale (m)
$M$	volume flux in plume ( $m^3/s$ )
$P, p$	pressure (Pa)
$Pe$	Peclet number (-)
$P_w$	pressure difference due to wind (Pa)
$P_s$	pressure difference due to stack effect (Pa)
$Q$	heat flux ( $W/m^2$ )
$q$	mass flux (Kg/s)
$q_v$	airflow rate ( $ach^{-1}$ )
$Ra$	Rayleigh number (-)
$Re$	Reynolds number (-)
$r$	radial distance (m)
$S_\phi$	source term in $\phi$ equation (varies)
$T$	temperature (K)
$t$	time (s)
$t_f$	false time-step (s)
$u_i$	velocity tensor = ( $u, v, w$ ) (m/s)
$u^+$	normalised velocity (-)
$u^{par}$	velocity parallel to wall (m/s)
$V$	velocity scale (m/s)
$v_f$	velocity scale for false time-steps (m/s)
$v_G$	Gaussian axial plume velocity (m/s)
$v_T$	'Top-hat' axial plume velocity (m/s)
$W$	heat input (Watts)
$x_i$	spatial coordinate tensor = ( $x, y, z$ ) (m):



$y^+$  normalised distance (-)

$y_0^+, y_{He}^+$	wall function constants (-)
$\alpha$	plume entrainment (-)
$\beta$	coefficient of expansion ( $K^{-1}$ )
$\beta_0$	turbulence model constant (-)
$\Delta H_{e_h}$	change in enthalpy across the interface (J/Kg)
$\Delta T_h$	change in temperature across the interface (K)
$\delta_{ij}$	kronecka delta = $\begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j \end{cases}$ (-)
$\varepsilon$	dissipation of turbulent kinetic energy ( $m^2/s^3$ )
$\phi$	arbitrary variable (varies)
$\Gamma_{eff}$	effective diffusivity (Kg/ms)
$\Gamma_{He_t}$	eddy diffusivity for enthalpy (Kg/ms)
$\Gamma_\phi$	diffusion coefficient for $\phi$ (Kg/ms)
$\eta_0$	turbulence model constant (-)
$\kappa$	wall function constant (-)
$\lambda$	thermal conductivity (W/mK)
$\mu, \mu_{eff}$	total (effective) dynamic viscosity (Kg/ms)
$\mu_l$	laminar viscosity (Kg/ms)
$\mu_t$	turbulent (or eddy) viscosity (Kg/ms)
$\theta$	azimuth angle (radians)
$\rho$	density ( $Kg/m^3$ )
$\sigma_{He}$	turbulent Prandtl number for enthalpy (-)
$\sigma_k$	turbulent Prandtl number for $k$ (-)
$\sigma_\varepsilon$	turbulent Prandtl number for $\varepsilon$ (-)
$\sigma_{ij}$	shear stress tensor (Pa)
$\tau$	wall shear stress (Pa)
$\nu$	kinematic viscosity ( $\mu/\rho$ ), ( $m^2/s$ )
$\xi$	normalised interface height = $h / H$

### **subscripts**

<i>Bref</i>	buoyancy reference
<i>Eref</i>	enthalpy reference
ext	external (ambient)
G	Gaussian value (see Appendix B)
int	internal
o	reference value
T	'top-hat' value (see Appendix B)
L	values expressed as a quantity per metre length

### **superscripts**

$k - \varepsilon$	value pertaining to standard $k - \varepsilon$ turbulence model
RNG	value pertaining to RNG $k - \varepsilon$ turbulence model

## **Glossary and Abbreviations**

ach <sup>-1</sup>	air changes per hour
AIVC	Air Infiltration and Ventilation Centre
BRE	Building Research Establishment
CFD	Computational Fluid Dynamics
CFDS	Computational Fluid Dynamics Services
CIBSE	Chartered Institute of Building Services Engineers
Control Volume	Discrete volumes of space in the computational domain over which the transport equations are solved
ESRU	Energy Systems Research Unit
PLEA	Passive and Low Energy Architecture