CFD simulation of dense gas dispersion in neutral atmospheric boundary layer with OpenFOAM

³ Vu Tran · E. Y. K. Ng · Martin Skote

5 Received: date / Accepted: date

4

Abstract In this study, MOST (Monin-Obukhov Similarity Theory) is used to 6 specify the profiles of velocity, turbulent kinetic energy (k) and eddy dissipa-7 tion rate (ϵ) in ABL (Atmospheric Boundary Layer) flow. The OpenFOAM stan-8 dard solver *buoyantSimpleFoam* is modified to simulate neutrally stratified ABL. 9 The solver is able to obtain equilibrium ABL. For gas dispersion simulation, 10 buoyantNonReactingFoam is developed to take into accounts fluid properties 11 change due to temperature, buoyancy effect and variable turbulent Schmidt num-12 ber. The solver is validated for dense gas dispersion in wind tunnel test and field 13 test of LNG (Liquefied Natural Gas) vapour dispersion in neutrally stratified ABL. 14

15 Keywords LNG dispersion \cdot CFD \cdot ABL \cdot OpenFOAM

School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore E-mail: tranlevu001@e.ntu.edu.sg

Martin Skote

School of Aerospace, Transport and Manufacturing, Cranfield University, UK

Vu Tran, E. Y. K. Ng

16 1 Introduction

¹⁷ Many human activities are affected by the atmospheric boundary layer (ABL). ¹⁸ This is also where most air pollution phenomena occur. Understanding of the ¹⁹ processes taking place in the ABL has attracted various research studies. Some ²⁰ typical applications of ABL related research topics are wind engineering, urban ²¹ flows, weather forecast, air pollution and risk assessment of hazardous material ²² spills in industrial sites

One hazardous dense gas is liquefied natural gas (LNG), which is an effec-23 tive solution for long-distance natural gas transfer. LNG has become the preferred 24 option for international trading of natural gas. However, LNG storage, handling, 25 transportation are exposed to serious risks for humans, equipment and the environ-26 ment due to thermal hazards associated with combustion events such as pool fire, 27 vapour cloud fire, explosion or rapid phase transition. Safety assessment and haz-28 ard mitigation methods should be applied to lower the possibilities of catastrophic 29 disaster relating to the LNG industry. The scope of this study is constrained to 30 the discussion of dense gas dispersion when released into the ABL. 31

Computational Fluid Dynamics (CFD) is increasingly being used in simula-32 tion of ABL flows. Open source CFD tool is a more powerful research tool in 33 comparison to proprietary software because of its flexibility to incorporate new 34 implementation of fields calculation and post-processing. OpenFOAM is an open 35 source CFD software package that attracts users from both industry and academia. 36 Using a general CFD code such as OpenFOAM for simulating ABL flow and gas 37 dispersion also encourages research sharing and reusing code in this specific field 38 where in-house code is usually adopted. 39

An important task before modelling gas dispersion in the ABL is obtaining 40 the crrect ABL flow prior to the release of gas source. One approach to achieve 41 this is using equilibrium ABL, i.e. zero stream-wise gradients of all variables, as 42 a steady state ABL flow. For neutral ABL, Richards and Hoxey (1993) proposed 43 appropriate boundary conditions of mean wind speed and turbulence quantities 44 for the standard $k - \epsilon$ model based on Monin-Obukhov similarity theory (MOST). 45 These profiles were derived assuming constant shear stress with height and were 46 used to model ABL as horizontally homogeneous turbulent surface layer (HHTSL). 47 However, HHTSL was hard to achieve mostly due to the ground boundary condi-48 tions (Yang et al., 2009), which manifested in a decay of velocity profile due to a 49 spike in the turbulent kinetic energy close to the ground. However, consistency be-50 tween wall boundary conditions, turbulence model with associated constants and 51 numerical schemes was shown to achieve HHTSL (Jonathon and Christian, 2012; 52 Parente et al., 2011; Yan et al., 2016). These authors adopted proprietary CFD 53 software for their simulation. Applying these implementations in open-source CFD 54 code also require extensive modifications of the source code to successfully simu-55 late equilibrium ABL. OpenFOAM was previously used for atmospheric buoyant 56 (Flores et al., 2013) and dense gas dispersions (Mack and Spruijt, 2013; Fiates 57 et al., 2016; Fiates and Vianna, 2016). However, the validation of these solvers 58 in simulation of equilibrium ABL was not reported. Therefore, the atmospheric 59 turbulence might not be correctly solved throughout the computational domain. 60 In this study, MOST is used to model the profiles of velocity, turbulent kinetic 61 energy (k) and eddy dissipation rate (ϵ) of ABL according to an approach proposed 62

by Richards and Hoxey (1993). These profiles are used as the boundary conditions
at the inlet of ABL flow simulation. OpenFOAM application buoyantSimpleFoam

is modified to simulate neutrally stratified ABL turbulence. For gas dispersion
simulation, buoyantNonReactingPimpleFoam is developed to take into account the
buoyancy effect and variable turbulent Schmidt number. The solver is validated
for dense gas dispersion cases from wind tunnel and field tests of LNG vapour
dispersion in neutrally stratified ABL.

70 2 Methodology

71 2.1 Models

⁷² The $k - \epsilon$ model is used for turbulence modelling. It is based on expression of ⁷³ turbulent dynamic viscosity μ_t by Jones and Launder (1972):

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{1}$$

Two additional transport equations for turbulence kinetic energy k and turbulence dissipation rate ϵ are required. To include the effect of buoyancy, the transport equations for k and ϵ are:

$$\frac{D}{Dt}(\rho k) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon \tag{2}$$

$$\frac{D}{Dt}(\rho\epsilon) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} G_k + C_{1\epsilon} C_{3\epsilon} \frac{\epsilon}{k} G_b - C_{2\epsilon} \rho \frac{\epsilon^2}{k}$$
(3)

⁷⁷ where ρ is the fluid density; $C_{\mu} = 0.09$, $\sigma_k = 1$, $\sigma_{\epsilon} = 1.3$, $C_{1\epsilon} = 1.44$ and $C_{2\epsilon} =$ ⁷⁸ 1.92 are model constants as proposed in original paper (Launder and Spalding, ⁷⁹ 1974). The value of $C_{3\epsilon}$ is calculated using:

$$C_{3\epsilon} = tanh \left| \frac{v}{u} \right| \tag{4}$$

where v, u are vertical and horizontal velocity accordingly.

 G_k is production of turbulence kinetic energy due to the mean velocity gradients. G_b is the buoyancy source term:

$$G_b = -\frac{\mu_t}{\rho P r_t} (\boldsymbol{g} \cdot \boldsymbol{\nabla} \rho) = -C_g \frac{\mu_t}{\rho} (\boldsymbol{g} \cdot \boldsymbol{\nabla} \rho)$$
(5)

where $C_g = 1/Pr_t$ is used as a model constant to take into account the userdefined value of turbulent Prandtl number Pr_t . g is gravitational vector.

Energy, heat and transport properties are determined by a set of thermophysical models (Greenshields, 2017) in OpenFOAM. This set defines mixture type, transport and thermodynamic properties models, choice of energy equation variable and equation of states.

The fluid in a simulation is defined as a mixture of fixed compositions. Enthalpy is chosen as energy equation variable. Transport and thermodynamic properties are determined using models based on the density ρ , which are calculated from pressure and temperature fields. Polynomial functions of order N are used to relate transport property μ , the specific heat c_p and density ρ with temperature field T:

$$\mu = \sum_{i}^{N-1} a_{\mu i} T^{i}$$

$$c_{p} = \sum_{i}^{N-1} a_{c_{p}i} T^{i}$$

$$\rho = \sum_{i}^{N-1} a_{\rho i} T^{i}$$
(6)

where $a_{\mu i}$, $a_{c_p i}$ and $a_{\rho i}$ are the polynomials coefficients.

95 2.2 Boundary conditions

96 2.2.1 ABL air inlet

⁹⁷ MOST ha been validated for the surface layer of ABL by many empirical stud-⁹⁸ ies (Foken, 2006). It assumes horizontally homogeneous and quasi-stationary flow ⁹⁹ field, i.e. profiles of flow variables are only varying in the vertical direction and ¹⁰⁰ their vertical fluxes are assumed constant. The inlet boundary conditions proposed ¹⁰¹ by Richards and Hoxey (1993) based on MOST are widely used in CFD study of ¹⁰² atmospheric flow. The velocity, turbulent production rate k and dissipation rate ϵ ¹⁰³ profiles in vertical direction z are written as:

$$u(z) = \frac{u_*}{\kappa} \ln \frac{z + z_0}{z_0}$$

$$k = \frac{u_*^2}{\sqrt{C\mu}}$$

$$\epsilon(z) = \frac{u_*^3}{\kappa(z + z_0)}$$
(7)

where z_0 is aerodynamic roughness length, u_* is friction velocity, C_{μ} is $k - \epsilon$ model constant.

These profiles are implemented in OpenFOAM as atmBoundaryLayer class and its subclasses. Required parameters are flow and vertical direction, reference velocity, reference height and aerodynamic roughness length. The friction velocity is calculated as:

$$u_* = \frac{\kappa * u_{ref}}{\ln((z_{ref} + z_0)/z_0)}$$
(8)

Parente et al. (2011) presented an elaborate procedure to ensure the consistency
for arbitrary inlet profile of turbulent kinetic energy k. Instead of altering model

constants as Yang et al. (2009), the effect of non-constant k on momentum and ϵ equation can be characterised by deriving an equation for C_{μ} :

$$C_{\mu}(z) = \frac{u_*^4}{k(z)^2} \tag{9}$$

Source terms are added to k and ϵ transport equations to ensure equilibrium condition:

$$S_{k} = \frac{\rho u_{*}\kappa}{\sigma_{k}} \frac{\partial}{\partial z} \left((z+z_{0}) \frac{\partial k}{\partial z} \right)$$

$$S_{\epsilon} = \frac{\rho u_{*}^{4}}{(z+z_{0})^{2}} \left[\frac{(C_{\epsilon 2} - C_{\epsilon 1})\sqrt{C_{\mu}}}{\kappa^{2}} - \frac{1}{\sigma_{\epsilon}} \right]$$
(10)

Richards and Norris (2011) revisited the problem of modelling the HHTSL by deriving the inlet profiles directly from the conservation and equilibrium equations. This allows various inlet profiles to be specified by varying the turbulence model constants. For standard $k - \epsilon$ models, the inlet profiles of velocity and turbulence properties are identical to Equation (7). However they suggested to change the von-Karman constant κ according to model constants as:

$$\kappa = \sqrt{(C_{\epsilon 2} - C_{\epsilon 1})\sigma_{\epsilon}\sqrt{C_{\mu}}} \tag{11}$$

Using the standard $k - \epsilon$ model constants, we obtain $\kappa_{k-\epsilon} = 0.433$.

123 2.2.2 Wall boundary conditions

¹²⁴ In CFD, the below approximation is used to calculate wall shear stress:

$$\tau_w = \nu_t \frac{\partial u}{\partial n} |_w \approx \nu_t \frac{(u_P - u_w)}{y_P} \tag{12}$$

where y_P is distance to wall of the wall adjacent cell. Subscript w and P denote field value evaluated at wall and wall adjacent point respectively.

However, this approximation is inaccurate when wall velocity gradient is significantly larger than velocity difference between the adjacent cell and the wall. This is the case for most ABL flows. Turbulent kinematic viscosity ν_t wall function is used to calculate the wall shear stress τ_w from the wall velocity difference. To take into account the aerodynamic roughness length z_0 , the calculation of turbulent kinematic viscosity at wall adjacent cell is:

$$\nu_t = \frac{\kappa u_* y_P}{\ln\left(\frac{y_P + z_0}{z_0}\right)} \tag{13}$$

where friction velocity u_* can be calculated from a simple relation derived by Launder and Spalding (1974), assuming that generation and dissipation of energy are in balance:

$$u_* = C_{\mu}^{1/4} k^{1/2} \tag{14}$$

 ϵ wall function is used to calculate value of ϵ at wall adjacent cell ϵ_P as:

$$\epsilon_P = \frac{C_{\mu}^{0.75} k^{1.5}}{\kappa y_P + z_0} \tag{15}$$

The wall is usually defined as non-slip condition where velocity is zero. However, to account for the effect of aerodynamic roughness length, a new boundary condition for velocity is implemented in OpenFOAM as:

$$u_P = \frac{u_*}{\kappa} \ln\left(\frac{y_P + z_0}{z_0}\right) \tag{16}$$

140 2.2.3 Top, side and outlet boundaries

At the outlet boundary, the flow is assumed fully developed and unidirectional.
All flow variables are supposed to be constant at this boundary.

The top and side of the computational domain are external boundaries repre-143 senting the far fields of flow. If a constant pressure is applied in these boundaries, 144 this may alter the inlet wind profile in case the prescribed pressure is not matched 145 with the boundary velocity (Luketa-Hanlin et al., 2007). The zero gradient bound-146 ary condition, which set normal velocity to zero and all others variables are set 147 equal to the inner values, or symmetry condition can be used at the top and side 148 boundaries to reserve the wind profile and eliminate the effect of changing the 149 inlet profiles. 150

Hargreaves and Wright (2007) showed that zero gradient velocity at the top boundary resulted in a decay of velocity downstream, due to the extraction of energy at the wall with respect to the wall shear stress. A driving shear stress, zero flux of turbulent kinetic energy and a flux of dissipation rate ϵ are imposed at the upper boundary:

$$\frac{du}{dz} = \frac{u_*}{\kappa z}$$

$$\frac{\mu_t}{\sigma_\epsilon} \frac{d\epsilon}{dz} = -\frac{\rho u_*^4}{\sigma_\epsilon z}$$
(17)

156 2.3 Numerical tool and data sets

¹⁵⁷ OpenFOAM is an open source CFD software package based on finite volume

¹⁵⁸ method, co-located variables and unstructured polyhedral meshes. In this study,

159 buoyantSimpleFoam is used to simulate ABL turbulence. The application buoyantNonReactingFoam

is developed based on rhoReactingBuoyantFoam solver, previously used for dense gas dispersion by Fiates et al. (2016), to simulate atmospheric turbulence under neutral stability for dispersion of dense gas continuous source in flat terrain. buoyantNonReactingFoam uses polynomial thermophysical models to account for the change of fluid properties due to temperature. The solver takes into account the buoyancy effect and the variable turbulent Schmidt number. Algorithms used in these two solvers are presented in Algorithm 1 and 2.

Algorithm 1 buoyantSimpleFoam solver algorithm

1: Initializing variables such as: time variables, mesh, solution control, fields and continuity

errors

- 2: while $t < t_{end}$ do
- 3: Solving momentum equation
- 4: Solving energy equation for enthalpy and correcting thermal properties
- 5: Solving pressure correction equation for p_{rgh} and calculating pressure field
- 6: Correcting turbulent properties
- 7: Writing fields
- 8: end while

Algorithm 2 buoyantNonReactingFoam solver algorithm

- 1: Initializing variables such as: time variables, mesh, solution control, fields and continuity errors
- 2: while $t < t_{end}$ do

3:	$t+=\Delta t$
4:	Solving continuity equation for density
5:	while PIMPLE outer correctors do
6:	Solving momentum equation
7:	Solving species transport equation
8:	Solving energy equation for enthalpy and correcting thermal properties
9:	while PIMPLE inner correctors do
10:	Solving pressure correction equation for p_{rgh} and calculating pressure field
11:	Solving continuity equation for density
12:	Calculating time step continuity errors
13:	end while
14:	Correcting turbulent properties
15:	end while
16:	Writing fields
17:	end while

Boundary conditions used for ABL flows are developed as new libraries in
 OpenFOAM. These include velocity inlet, turbulent kinetic energy, dissipation
 rate inlet and wall boundary conditions.

A set of full scale field tests and experimental wind tunnel tests for LNG dispersion model validation was reported in (Ivings et al., 2013). Most data of these tests were available in REDIPHEM database (Nielsen and Ott, 1996). The data contains physical comparison parameters of each test. These are maximum arcwise concentration, i.e. the maximum concentration across an arc at the specified distance from the source and point-wise concentration, i.e. the concentration at specific sensor locations. Two wind tunnel data DA0120 and DAT223 are used to validate OpenFOAM solver in prediction of dense gas dispersion over a flat, unobstructed terrain in simulated neutral ABL. In these tests, continuous source of SF₆ gas was released in flat terrain without obstructions. For field test, we select Burro9, which is continuous LNG spills under neutral ABL.

181 3 Neutral ABL simulation

182 3.1 Domain and mesh

- A 2D domain of $5000 \text{ m} \times 500 \text{ m}$ with the resolution of 500×50 cells is used for the
- ¹⁸⁴ simulation of neutral ABL over flat terrain. The mesh is uniform in stream-wise
- direction and stretched in vertical direction with the expansion ratio of 1.075.

186 3.2 Numerical setting

¹⁸⁷ The boundary conditions of the cases are represented in Table 1.

 ${\bf Table \ 1} \ {\rm Boundary \ conditions \ for \ neutral \ ABL \ simulation}$

ABL inlet	profiles of k, u, ϵ	Eq. (7)	
ABL outlet	zeroGradient for all variables		
	<pre>fixedValue for static pressure</pre>		
ABL side	zeroGradient for all variables		
ABL top	zeroGradient for all variables		
	<code>fixedFlux/zeroGradient</code> for u and ϵ	Eq. (17)	
ABL bottom	noSlip for u		

ABL parameters used to define inlet variable profiles are listed in Table 2 according to the reference case of Hargreaves and Wright (2007):

Table 2 ABL parameters using for neutral ABL simulation

$u_* (m/s)$	z_0 (m)	$u_{ref} \ (m/s)$	z_{ref} (m)
0.625	0.01	10	6

¹⁹⁰ Steady state simulation is employed using buoyantNonReactingSimpleFoam de-

¹⁹¹ scribed in previous section. OpenFOAM discretisation schemes, velocity-pressure

¹⁹² coupling algorithm as well as linear solvers are listed below:

- 193 Time schemes: steadyState
- 194 Gradient schemes: Gauss linear
- ¹⁹⁵ Divergence schemes: Gauss limitedLinear 1
- ¹⁹⁶ Surface normal gradient schemes: corrected
- 197 Laplacian schemes: Gauss linear corrected
- ¹⁹⁸ Interpolation schemes: linear
- ¹⁹⁹ Solving algorithm: SIMPLE

200 - Linear solver for p: GAMG with DICGaussSeidel preconditioner

- 201 Linear solver for U, h, k, epsilon: PBiCGStab with DILU preconditioner
- 202 Residual control is set at three order of magnitude for pressure and four order
- of magnitude for other variables such as U, k, ϵ and h. Modification of $k \epsilon$
- (Equation (11)) are used to simulate neutral ABL and comparing with standard
- $_{205}$ models. These three cases are summarised in Table 3
- Different levels of inlet kinetic energy are obtained by altering C_{μ} according
- ²⁰⁷ to Equation (9). The source term by Pontiggia et al. (2009) is implemented us-

Turbulence model	standard $k-\epsilon$	
	modified $k-\epsilon$	Eq. (10)
Wall functions	nutkWallFunction for ν_t	
	<code>epsilonWallFunction</code> for ϵ	
	kqRWallFunction for k	

Table 3 Turbulence models setting for neutral ABL simulation

ing Equation (10). Two values of default value $C_{\mu} = 0.09$ and $C_{\mu} = 0.017$ are simulated and compared with Monin-Obukhov theory.

210 3.3 Results and discussion of neutral ABL simulations

Modification of $k - \epsilon$ models achieve the matched results as shown in Figure 1.

²¹² Including source terms as in Equation (10) is sufficient to compensate terms deflec-

 $_{^{213}}$ tion from calculation of von-Karman constant $\kappa_{k-\epsilon}=4.3$ from model constants

(Equation (11)) and $\kappa = 4.1$ used in Monin-Obukhov theory.

Results from modelling different turbulence kinetic energy by varying C_{μ} are 215 presented in Figure 2. The profiles of velocity and dissipation rate are perfectly 216 matched with the Monin-Obukhov profiles. In the $C_{\mu}\,=\,0.017$ simulation, the 217 value of k near ground is smaller than the theoretical value, however, the kinetic 218 energy level is matched with the theory at greater height. The smaller value of k at 219 the wall adjacent cell is due to the wall function, where wall treatment used with 220 the default $C_{\mu} = 0.09$ is implemented. However, the overall results are acceptable 221 for verifying the proposed model in simulating different levels of kinetic energy. 222



Fig. 1 Comparing velocity, turbulent kinetic energy and turbulent dissipation rate profiles at the outlet boundary from simulation of neutral ABL using standard $k - \epsilon$ (kEps), modified $k - \epsilon$ (kEpsMod) turbulence model and MOST inlet profiles (MOST)



Fig. 2 Comparing velocity, turbulent kinetic energy and turbulent dissipation rate profiles from simulations of different kinetic energy levels by varying $C_{\mu} = 0.09$ (KEpsCmu09) $C_{\mu} =$ 0.017 (KEpsCmu017) and MOST inlet profiles (MOST)

²²³ 4 Dense gas dispersion in wind tunnel tests

²²⁴ 4.1 Numerical setting

The effect of the turbulent Schmidt number Sc_t is investigated in dense gas dispersion. Three test cases are summarised in Table 4. The effect of turbulent models is examined by applying the modified $k - \epsilon$ which is already validated in simulating ABL over flat terrain in Section 3.

Table 4 Turbulent Schmidt number Sc_t in Hamburg tests

	Case 1	Case 2	Case 3
Sc_t	1	0.7	0.3
Label (Fig. 4, 10)	FOAM_ORIG	FOAM_Sc07	FOAM_Sc03

Firstly, the steady simulation using bouyantSimpleFoam is performed to estab-229 lish the steady ABL flow prior to the dense gas release. The solver which includes 230 buoyancy effects bouyantSimpleFoam is used to account for of density stratifica-231 tion in dense gas flow. The atmospheric inlet profiles are specified by MOST with 232 parameters in Table 5. Standard $k - \epsilon$ with modifications is used to study the 233 ability to simulate the ABL with each model. Secondly, the transient simulation 234 is performed using steady simulation solutions as initial fields. A modified version 235 of rhoReactingBouyantFoam is studied to model multi-species flow where mixture 236 considered are air and dense gas SF_6 . The wind tunnel tests were conducted in 237 isothermal condition, therefore constant thermal and transport properties are used 238 for both gases. 239

_

	Unit	DA0120	DAT223
Substance		SF_6	SF_6
Density	kg/m^3	6.27	6.27
z_0	m	0.0001	0.0001
Wind speed	m/s	0.54	0.74
Reference height	m	0.00718	0.01367
Ambient temperature	^{0}C	20	20
Source diameter	m	0.07	0.07
Spill rate	kg/s	0.0001743	0.000872

 $\textbf{Table 5} \hspace{0.1 cm} \text{Hamburg flat, unobstructed test case parameters (Nielsen and Ott, 1996)}$

 $_{\rm 240}$ $\,$ In simulations of DA0120 and DAT223 tests, the discretisation schemes and linear

²⁴¹ solver setting are identical to the simulation of neutral ABL (Section 3).

242 4.2 Results and discussion of gas dispersion in wind tunnel tests

243 4.2.1 Peak concentration prediction

The steady state plumes at ground level of DAT0120 and DAT223 tests are plotted in Figure 3. Under higher release volume flow rate and higher wind speed, DAT223 plume is wider and is spreading further downstream than the DAT0120 plume.

The predicted and measured peak gas concentration are compared at several distances from the spill in Figure 4. Turbulent Schmidt number Sc_t has significant effect in predicting dense gas dispersion. The original **rhoReactingBouyantFoam** code, with assumption of species diffusivity equals to viscosity, is shown to overpredict concentration with a factor of three. The modified code takes into account the variable species diffusivity Sc_t by reading this parameter from user input. The



Fig. 3 DA0120(up) and DAT223(below) ground level contours of SF_6 mass fraction

value of $Sc_t = 0.3$ is shown to yield a perfect match with the experimental data. However, there is a slightly acceptable over-predicted species concentration at a point near the source release.

Results from the DAT223 simulation are presented in Figure 5. Satisfactory over
 predicted peak concentration is similar to DA0120 case.

258 4.2.2 Point-wise concentration

Figure 6 presents gas concentration at the downwind distance X = 1.84 of the DA0120 test. The simulation can reproduce the averaged gas concentration. The first incidence time of gas concentration is earlier than observed in experiments. However, the time for reaching averaged maximum concentration is well predicted.

263 4.2.3 Statistical model evaluation

Statistical Performance Measures (SPMs) are means to compare prediction parameters and the measured ones for model evaluation. The SPM chosen should reflect



Fig. 4 Peak concentration for DA0120 test



Fig. 5 Peak concentration for DAT223 test



Fig. 6 Concentration at X = 1.84 of DA0120 test

the bias of these predictions. In the context of LNG vapour dispersion model evaluation, Ivings et al. (2013) proposed five SPMs including mean relative bias (MRB), mean relative square error (MRSE), the fraction of predictions within the factor of two of measurements (FAC2), geometric mean bias (MG) and geometric variance (VG). Definition and acceptability criteria for each SPMs are presented in tabular form as Table 6 where C_m , C_p are the measured and simulated concentration, respectively, and \overline{A} denotes the mean operation of variable A.

Statistical performance of OpenFOAM results are compared with the specialised commercial code for gas dispersion FLACS in Table 6. FLACS results are extracted from (Hansen et al., 2010). The performance of current OpenFOAM code is considerably better than FLACS. In fact, FLACS is based on the porosity distributed resistance (PDR) approach. Therefore, this modelling of the boundary layer close to solid surfaces might contribute to the outperformance of the Open279 FOAM model in the comparison. In conclusion, even though larger tests were

- validated in FLACS, the proposed model in OpenFOAM is a promising tool for
- ²⁸¹ further investigation of atmospheric dense gas dispersion.

SPM	MRB	RMSE	FAC2	MG	VG
Definition	$\overline{\left(\frac{C_m - C_p}{0.5(C_m - C_p)}\right)}$	$\overline{\left(\frac{(C_m-C_p)^2}{0.25(C_m+C_p)^2}\right)}$	$\frac{C_m}{C_p}$	$\exp\left(\overline{\ln\frac{C_m}{C_p}}\right)$	$\exp\left(\overline{\left(\ln\frac{C_m}{C_p}\right)^2}\right)$
Acceptable range	[-0.4, 0.4]	< 2.3	[0.5, 2]	[0.67, 1.5]	< 3.3
Perfect value	0	0	1	1	1
FLACS (Hansen et al., 2010)	0.25	0.29	0.89	1.34	1.61
FOAM	-0.06	0.02	1.07	1.06	1.02

 Table 6
 Statistical performance measures of Hamburg unobstructed tests

²⁸² 5 LNG vapour dispersion in field tests

283 5.1 Numerical setting

The steady simulation uses the atmospheric inlet specified by MOST. Standard $k - \epsilon$ with modifications is used to study the ability to simulate the ABL with each model. All required meteorological parameters are tabulated in Table 7, where u_{ref} and T_{ref} are air velocity and temperature at the height of 2 m respectively.

The transient simulation is divided into two steps. The first step is during the spill duration, i.e. from the time of zero to when the spill ends. The second step is after the spill stops to the end time of simulation. The gas inlet is treated as a ground boundary in this later step.

The gas inlet condition is usually obtained from separate source term modelling. There is not much information about the vaporisation of LNG from the ex-

perimental data. Therefore, uncertainty arises at the setting of this condition. Mass 294 flux of LNG or the LNG vaporization rate is used to derive source term of LNG 295 spilling. Luketa-Hanlin et al. (2007) reviewed a number of experiments conducted 296 to estimate the LNG vaporization rate of the spill on water, the range of this value 297 varied between approximately 0.029 to 0.195 kg m⁻² s⁻¹. In the case of Burro test, 298 the simulated vaporisation rate is assumed to be $m_{LNG}^{\prime\prime} = 0.167 \,\mathrm{kg}\,\mathrm{m}^{-2}\,\mathrm{s}^{-1}$. Den-299 sity of LNG vapour is similar to that of CH₄ at boiling point $\rho_{LNG} = 1.76 \text{ kg m}^{-3}$ 300 (Luketa-Hanlin et al., 2007). The spill diameter is derived from the vaporization 301 rate, reported spill mass m_{spill} and duration t_{spill} : 302

$$D_{spill} = \sqrt{\frac{4m_{spill}}{\pi m_{LNG}' t_{spill}}} \tag{18}$$

³⁰³ The volume spill rate is used as gas inlet condition:

$$\dot{V}_{spill} = \frac{m_{spill}}{\rho_{LNG} t_{spill}} \tag{19}$$

The LNG spill variables used in simulation are also tabulated in Table 7.

Table 7 Burro9 tests meteorological and gas release parameters

Parameters	u_{ref}	u_*	z_0	T_{ref}	t_{spill}	D_{spill}	\dot{V}_{spill}
Unit	m/s	m	m/s	Κ	s	m	m^3/s
	5.7	0.252	2E-4	308.55	79	32.2	77.17

³⁰⁵ 5.2 Results and discussion of LNG vapour dispersion

306 5.2.1 The steady simulation

Profiles of velocity and turbulence quantities are sampled at the **outlet** boundary and compared with Monin-Obukhov theory profiles which are used as inlet boundary conditions. The steady state simulation of ABL with $k - \epsilon$ reveals that wind velocity and turbulence profiles are accurately reproduced as presented in Figure 7. The success of the modified $k - \epsilon$ model proves that the proposed model can adequately reproduce the Monin-Obukhov ABL profiles in the full scale simulation.



Fig. 7 Comparing ABL profiles at outlet boundary of Burro9 simulation using modified $k - \epsilon$ model (kEpsMod) and MOST inlet profiles (MOST)

314 5.2.2 Mesh sensitivity study

Maximum concentration at the arcs of 57 m, 140 m, 400 m and 800 m downwind are used as performance parameters for the mesh sensitivity study. Three meshes $_{\rm 317}$ $\,$ with refined factors as summarised in Table 8 are used to simulate LNG gas dis-

 $_{\tt 318}$ $\,$ persion under adiabatic thermal wall condition. Results from four peak arc-wise

 ${\bf Table \ 8} \ {\rm Burro \ test \ computational \ domain \ and \ mesh \ parameters}$

³¹⁹ concentrations are plotted in Figure 8.

	Mesh 1	Mesh 2	Mesh 3	
Domain region	[(-150, 0, 50), (850, 300, 50)]			
Refined region	[(-100, 0, 5), (400, 100, 5)]			
Mesh size (m)	10	5	2	
Mesh refined size (m)	5	2	1	



Fig. 8 Result of Burro9 mesh sensitivity study

Increasingly mesh refinements help to resolve maximum concentration more
 accurately. The difference of gas concentrations between meshes are significantly

reduced with refinement. Due to computational restriction, no further mesh is
used for mesh sensitivity study and Mesh 3 parameters (Table 8) is chosen for the
following study.

325 5.2.3 Ground heat transfer sensitivity study

Three different models of heat transfer from the ground are used to study their effect on the numerical results, which are summarised in Table 9. For constant heat flux case, the value of 200 W/m^2 is used.

Table 9 Wall thermal boundary conditions in Burro tests

	Case 1	Case 2	Case 3
Heat transfer model	Adiabatic wall	Constant Heat Flux	Wall temperature
Label (Fig. 9)	Adiabatic	fixedFlux	fixedTem

The effect of ground heat in predicting peak gas concentration is plotted in Figure 9.

The adiabatic case results in a better prediction of experimental data than the fixed flux and fixed temperature cases. However, all simulations yield underpredicted results. This may be due to that the buoyancy effect is over-predicted and consequently the gas concentration is zero in the fixed flux case at downwind arcs (at 400 and 800 m).



Fig. 9 Result of Burro9 ground heat transfer study

$_{336}$ 5.2.4 Turbulence Schmidt number, Sc_t , sensitivity study

Two values of $Sc_t = 1$ and $Sc_t = 0.3$ are used for studying the sensitivity of the proposed model in predicting the maximum gas concentration. Results are compared in Figure 10.

 $Sc_t = 0.3$, which was used previously in wind tunnel dense gas dispersion is shown to be appropriate for accurate prediction of maximum gas concentration at the 57 m array and 140 m array. Further downwind, at 400 m array and 800 m array, there is no significant difference between the two values.



Fig. 10 Result of Burro9 turbulent Schmidt number Sc_t study

344 5.2.5 Isosurface contour

The vertical isosurface contours at X = 140 are illustrated in Figure 11. Underpredicted cloud height are revealed in all tests indicating that the cloud buoyancy is not correctly solved.

Horizontal isosurface contours at height Z = 1 are shown in Figure 12. The gas 348 concentration contour is plotted side by side with the contour from experiment 349 data, where the left is the result of interpolating concentration at some concen-350 tration data points (presented in plots by black dot points), the right is from 351 experimental data. Overall, the cloud height is considerably well predicted but 352 the cloud width is over-predicted. Furthermore, it can be seen that the gas moves 353 downwind slower than experimental data, which under-estimates the downwind 354 spreading of the gas cloud. 355



 $\label{eq:Fig.11} {\bf Fig. 11} \ {\bf Vertical isosurface at X} = 140, {\bf Top: Simulation, Bottom: Experimental data of Burro9} \\ {\bf test}$



Fig. 12 Horizontal isosurface at Z = 1; Top: Simulation, Bottom: Experimental data of Burro9 test

356 5.2.6 Concentration predictions

FDS (Fire Dynamics Simulator) (McGrattan et al., 2013) is a low Mach number code using the LES turbulence model. The computational domain is discretised into a connected rectilinear mesh. The governing equations are discretised using finite-difference method. A second-order scheme is used for space discretisation and an explicit second-order Runge-Kutta scheme for time discretisation. OpenFOAM concentration results are compared with FDS data extracted from (Mouilleau and Champassith, 2009).

The comparison of OpenFOAM, FDS, and experimental results for Burro9 test is shown in Figure 13. FDS is over-predicted, while OpenFOAM is under-predicted. However, OpenFOAM is accurate in prediction at 800 m arc.



Fig. 13 Maximum arc-wise gas concentrations $\tt Burro9$ test

Figure 14 is the plot of gas concentration at $1\,\mathrm{m}$ elevation at $140\,\mathrm{m}$ down-367 wind of Burro9 experiment (EXP) and simulations using the developed solver 368 (FOAM) and FDS (FDS). For the developed solver result, the peak concentra-369 tion is under-estimated while the temporal trend of changing concentration gen-370 erally shows good agreement with validation data. The concentration magnitude 371 is fairly matched except during local maximum/minimum durations. Also shown 372 in Figure 14 is the result from FDS simulation which is generally over-predicted. 373 However, the developed solver cannot capture the fluctuation, while FDS yields 374 fluctuating gas concentration over the time period. This is an advantage of LES 375 over RANS turbulence model. The over-prediction of FDS may be due to that a 376 constant coefficient Smagorinsky model was adopted in the simulation. However, 377 the dynamic Smagorinsky model was shown to improve the gas dispersion predic-378 tion (Ferreira Jr. and Vianna, 2016). This indicates that it would be a promising 379 approach to use LES in order to enhance the performance of the developed solver. 380

381 5.2.7 Statistical model evaluation

Overall statistical performance of OpenFOAM results are compared versus FLACS with data extracted from (Hansen et al., 2010) in Table 10. The predictions do not match all SPMs. However, some important SPMs are within the acceptable range. All gas concentrations are within a factor of two (FAC2=1) and better than FLACS (FAC2 = 0.94).



Fig. 14 Point concentration at 140 m of Burro9 test

Table 10 Statistical performance measures of Burro tests

	MRB	RMSE	FAC2	MG	VG
FLACS (Hansen et al., 2010)	0.16	0.12	0.94	1.18	1.14
FOAM Burro9	0.44	0.23	1	0.63	1.28

387 6 Conclusions

A solver is developed to reproduce horizontal homogeneous atmospheric surface layer in neutrally stratified ABL using OpenFOAM. The empirical atmospheric boundary layer model MOST is used to specify the inlet boundary conditions for velocity, turbulent kinetic energy and dissipation rate. Flow variable profiles at outlet boundary are successfully maintained and consistent with their profiles at the inlet boundary. This demonstrates the effectiveness of the solver in simulating the horizontal homogeneous atmospheric surface layer. It can also predict different
levels of ABL turbulence kinetic energy.

A solver for ABL gas dispersion simulation taking into account buoyancy effect, variable turbulence Schmidt number and ground heat transfer is developed using the OpenFOAM platform. In the study of dense gas dispersion in neutral simulated ABL, the model is successfully validated by reproducing maximum gas concentration. SPMs from simulation results are better than those from the specialized commercial software for gas dispersion, FLACS.

In the study of LNG accidental release, a dense cold gas vapour dispersion in 402 ABL with three ground heat transfer assumptions are simulated and compared 403 with the full scale field measurements. The gas peak concentration is used as val-404 idation parameters. Adiabatic wall assumes zero heat flux from ground to the gas 405 cloud, whereas, the fixed temperature model assumes isothermal ground where the 406 ground temperate remains unchanged when in contact with the cold gas cloud. The 407 real heat flux to the gas cloud would be in between these two cases. The other 408 model assuming a fixed flux of heat to the gas cloud is also included. Of the three 409 ground heat transfer models, adiabatic wall gives the closest prediction of gas 410 peak concentration. The model is shown to accurately predict vertical buoyancy 411 while the cloud spreading downwind is under-predicted. SPMs from the simula-412 tion results are compared with the LES code FDS and specialized dispersion code 413 FLACS, showing that the solver is more accurate in predicting gas concentra-414 tion in neutrally stratified ABL. Further investigation is required to validate the 415 OpenFOAM solver in ABL with thermal stratification. 416

417	References
418	Ferreira Jr ES, Vianna SSV (2016) LARGE EDDY SIMULATION COMBINED
419	WITH EQUIVALENT DIAMETER FOR TURBULENT JET MODELLING
420	AND GAS DISPERSION
421	Fiates J, Vianna SSV (2016) Numerical modelling of gas dispersion us-
422	ing OpenFOAM. Process Safety and Environmental Protection 104,
423	Part:277–293, DOI $http://doi.org/10.1016/j.psep.2016.09.011$, URL
424	http://www.sciencedirect.com/science/article/pii/S0957582016302105
425	Fiates J, Santos RRC, Neto FF, Francesconi AZ, Simoes V, Vianna SSV (2016) An
426	alternative CFD tool for gas dispersion modelling of heavy gas. Journal of Loss
427	Prevention in the Process Industries 44:583–593, DOI 10.1016/j.jlp.2016.08.002,
428	URL https://www.scopus.com/inward/record.uri?eid=2-s2.0-
429	84994155203 & doi = 10.1016% 2 Fj. jlp. 2016.08.002 & partner ID = 40 & md5 = f15971 c9 c9332 b320 e618 d86 adf 80351 for the state of the state o
430	Flores F, Garreaud R, Muñoz RC (2013) CFD simulations of
431	turbulent buoyant atmospheric flows over complex geometry:
432	Solver development in OpenFOAM. Computers & Fluids 82:1–
433	13, DOI http://dx.doi.org/10.1016/j.compfluid.2013.04.029, URL
434	http://www.sciencedirect.com/science/article/pii/S0045793013001795
435	Foken T (2006) 50 Years of the Monin–Obukhov Similarity Theory. Boundary-
436	Layer Meteorology $119(3):431-447$, DOI $10.1007/s10546-006-9048-6$, URL
437	https://doi.org/10.1007/s10546-006-9048-6
438	Greenshields CJ (2017) OpenFOAM user guide version 5. URL

439 https://cfd.direct/openfoam/user-guide/

440	Hansen OR, Gavelli F, Ichard M, Davis SG (2010) Validation of FLACS against ex-
441	perimental data sets from the model evaluation database for LNG vapor disper-
442	sion. Journal of Loss Prevention in the Process Industries 23(6):857–877, DOI
443	$10.1016/{\rm j.jlp.2010.08.005},$ URL http://dx.doi.org/10.1016/j.jlp.2010.08.005
444	Hargreaves D, Wright N (2007) On the use of the k-epsilon model
445	in commercial CFD software to model the neutral atmospheric
446	boundary layer. Journal of Wind Engineering and Industrial Aero-
447	dynamics 95(5):355–369, DOI 10.1016/j.jweia.2006.08.002, URL
448	http://www.sciencedirect.com/science/article/pii/S016761050600136X
449	http://linkinghub.elsevier.com/retrieve/pii/S016761050600136X
450	Ivings MJ, Lea CJ, Webber DM, Jagger SF, Coldrick S (2013) A protocol for
451	the evaluation of LNG vapour dispersion models. Journal of Loss Prevention
452	in the Process Industries 26(1):153–163, DOI 10.1016/j.jlp.2012.10.005, URL
453	http://dx.doi.org/10.1016/j.jlp.2012.10.005
454	Jonathon S, Christian M (2012) k-epsilon simulations of the neutral atmospheric
455	boundary layer: analysis and correction of discretization errors on practical grids.
456	International Journal for Numerical Methods in Fluids $70(6)$:724–741, DOI
457	doi:10.1002/fld.2709, URL https://doi.org/10.1002/fld.2709
458	Jones WP, Launder BE (1972) The prediction of laminarization with a two-
459	equation model of turbulence. International Journal of Heat and Mass Trans-
460	fer 15(2):301–314, DOI https://doi.org/10.1016/0017-9310(72)90076-2, URL
461	http://www.sciencedirect.com/science/article/pii/0017931072900762
462	Launder BE, Spalding DB (1974) The numerical computation of tur-
463	bulent flows. Computer Methods in Applied Mechanics and Engineer-

 $_{464}$ ing 3(2):269–289, DOI http://dx.doi.org/10.1016/0045-7825(74)90029-2, URL

465	http://www.sciencedirect.com/science/article/pii/0045782574900292
466	Luketa-Hanlin A, Koopman RP, Ermak DL (2007) On the application of com-
467	putational fluid dynamics codes for liquefied natural gas dispersion. Journal of
468	Hazardous Materials 140(3):504–517, DOI 10.1016/j.jhazmat.2006.10.023
469	Mack A, Spruijt MPN (2013) Validation of OpenFoam for heavy gas dis-
470	persion applications. Journal of Hazardous Materials 262:504–516, DOI
471	10.1016/j.jhazmat.2013.08.065
472	McGrattan K, Hostikka S, McDermott R, Floyd J, Weinschenk C, Overholt K
473	$\left(2013\right)$ Fire dynamics simulator, user's guide. NIST special publication 1019:20
474	Mouilleau Y, Champassith A (2009) CFD simulations of at-
475	mospheric gas dispersion using the Fire Dynamics Simulator
476	(FDS). Journal of Loss Prevention in the Process Industries
477	22(3):316–323, DOI https://doi.org/10.1016/j.jlp.2008.11.009, URL
478	http://www.sciencedirect.com/science/article/pii/S0950423008001496
479	Nielsen M, Ott S (1996) A collection of data from dense gas experiments. Tech.
480	rep., Risø National Laboratory, Roskilde

Parente A, Gorlé C, van Beeck J, Benocci C (2011) Improved k–epsilon
model and wall function formulation for the RANS simulation of
ABL flows. Journal of Wind Engineering and Industrial Aerodynamics 99(4):267–278, DOI https://doi.org/10.1016/j.jweia.2010.12.017, URL
http://www.sciencedirect.com/science/article/pii/S016761051100002X

Pontiggia M, Derudi M, Busini V, Rota R (2009) Hazardous gas dispersion:
A CFD model accounting for atmospheric stability classes. Journal of hazardous materials 171(1-3):739–747, DOI 10.1016/j.jhazmat.2009.06.064, URL
http://www.sciencedirect.com/science/article/pii/S0304389409009844

490	Richards PJ, Hoxey RP (1993) Appropriate boundary conditions for
491	computational wind engineering models using the k-epsilon turbulence
492	model. Journal of Wind Engineering and Industrial Aerodynamics 46-
493	47 (Supplement C):145–153, DOI https://doi.org/10.1016/0167-6105 (93)90124-
494	7, URL http://www.sciencedirect.com/science/article/pii/0167610593901247
495	Richards PJ, Norris SE (2011) Appropriate boundary conditions for computational
496	wind engineering models revisited. Journal of Wind Engineering and Industrial
497	$\label{eq:action} Aerodynamics \ 99(4): 257-266, \ DOI \ https://doi.org/10.1016/j.jweia.2010.12.008, \ and \ begin{tabular}{lllllllllllllllllllllllllllllllllll$
498	$\label{eq:url:likelihood} {\rm URL\ http://www.sciencedirect.com/science/article/pii/S0167610510001418}$
499	Yan BW, Li QS, He YC, Chan PW (2016) RANS simulation of neutral atmospheric
500	boundary layer flows over complex terrain by proper imposition of boundary
501	conditions and modification on the k-epsilon model. Environmental Fluid Me-
502	chanics 16(1):1–23, DOI 10.1007/s10652-015-9408-1
503	Yang Y, Gu M, Chen S, Jin X (2009) New inflow boundary conditions for
504	modelling the neutral equilibrium atmospheric boundary layer in computa-
505	tional wind engineering. Journal of Wind Engineering and Industrial Aerody-
506	namics 97(2):88–95, DOI http://dx.doi.org/10.1016/j.jweia.2008.12.001, URL

507 http://www.sciencedirect.com/science/article/pii/S0167610508001815

CERES Research Repository

School of Aerospace, Transport and Manufacturing (SATM)

Staff publications (SATM)

CFD simulation of dense gas dispersion in neutral atmospheric boundary layer with OpenFOAM

Tran, Vu

2019-08-02 Attribution-NonCommercial 4.0 International

Tran V, Ng EYK, Skote M. (2019) CFD simulation of dense gas dispersion in neutral atmospheric boundary layer with OpenFOAM. Meteorology and Atmospheric Physics, Volume 132, April 2020, pp. 273–285 https://doi.org/10.1007/s00703-019-00689-2 Downloaded from CERES Research Repository, Cranfield University