EPTT-2020-0103
NUMERICAL STUDY OF INCLINED DENSE JET DISPERSION IN STAGNANT ENVIRONMENTS

Pedro Henrique Augusto Mota
Marcelo Maia Ribeiro Damasceno
Aristeu da Silveira Neto
João Marcelo Vedovoto
UFU - Universidade Federal de Uberlândia, Pós-graduação de Engenharia Mecânica - FEMEC/UFU. Av. João Naves de Ávila, 2121, Bairro Santa Mônica, 38400-902, Uberlândia, MG, Brasil.,
pmota@ufu.br, marcelomrd@ufu.br, aristeus@ufu.br, vedovoto@ufu.br

Abstract. *A numerical investigation of the dispersion of inclined dense jets in stagnant environments was performed. In order to numerically simulate the advective and diffusive transport of the dense jet, Computational Fluid Dynamics (CFD) techniques were considered. The simulations were executed in the MFSim software using the Large Eddy Simulation (LES) approach. The modelling of the advective and diffusive scalar transport equation was conducted based on two sub-grid scalar flux models with constant and dynamically calculated turbulent Schmidt numbers. Moreover, a methodology was considered to model a turbulent inflow boundary condition. The Adaptive Mesh Refinement method (AMR) was considered to dynamically refine the domain according to the jet dispersion and to reduce computational costs. The LES simulations results were compared with experimental data of a turbulent dense jet discharging into a stagnant environment. The simulations performed for the different SGS models with no perturbations showed precise results for the fluid dynamics. However, the scalar dispersion could not be predicted accurately. The addition of perturbations helped to improve the transitional effects of the jet and to achieve better results for the scalar dispersion. Finally, the present work demonstrates a robust and precise methodology to evaluate the dense jet dispersion phenomena, which could also be applied to analyze the impact of brine discharge operation on the marine environment.*

Keywords: *LES, dense jet, turbulent inflow boundary, AMR*

1. INTRODUCTION

The inclined dense jet phenomenon is observed in several industrial operations. In desalination plants, this phenomenon creates an increase in salinity in regions close to the disposal site. Since the increase in salinity is associated with an imbalance in the marine ecosystem, the study of each parameter associated with the disposal of this high salinity solution is necessary. In order to estimate conditions that lead to greater dilution, experimental strategies are usually employed. In addition, computational fluid dynamics (CFD) techniques show precise information for the study of this transport phenomenon (Malcangio and Petrillo, 2010; Palomar *et al.*, 2012).

In desalination plants operations, the inclined dense jet is set upwardly and located near the seabed. For this discharge orientation, the high salinity jet achieves a maximum rise height due to the jet's initial moment and buoyancy forces. After the jet achieves its maximum height, the jet then falls back to the seabed. In addition to it, the jet inclination leads to disperse this high salinity fluid away from the disposal site, making it possible to evaluate the mixing phenomenon along its trajectory (Abessi and Roberts, 2015).

Several authors have studied the dynamics of this mixing phenomenon considering single inclined jets and different dense jet configurations experimentally. Zeitoun *et al.* (1970) evaluated the dense jet dispersion phenomenon for different nozzle angles (30º, 45º, 60º, 90º) and densimetric Froude numbers from 8 to 52. In order to analyze the influence of each parameter, the jet trajectory and the dilution at the jet centerline maximum height were obtained. Zeitoun *et al.* (1970) reported that for the same densimetric Froude number the highest dilution was observed for a jet angle of 60º.

Roberts *et al.* (1997) performed experiments of 60° inclined dense jets and determined the jet geometrical and dilution characteristics of the phenomena. The dilution characteristics were obtained at the jet centerline maximum height and at the return point. Cipolina *et al.* (2005) and Shao and Law (2010) analyzed the dispersion and dilution characteristics of negatively buoyant jets with inclinations of 30° to 60° considering the jet centerline maximum height and the return point. While Cipolina *et al.* (2005) considered a densimetric Froude up to 206 and Shao and Law (2010) considered densimetric Froude numbers up to 100. Moreover, Shao and Law (2010) investigated the effect of bed proximity.

The single dense jet phenomenon has been investigated with the application of different Computational Fluid Dynamics (CFD) techniques. The analysis conducted by these numerical approaches has become important due to the detailed and accurate capacity to reproduce the phenomenon if compared to Integral models (Palomar *et al.,* 2012). In order to study the phenomena, Gildeh *et al.* (2014) and Zhang *et al.* (2016) applied different CFD methods to solve the transport equations using the OpenFOAM® software.

The turbulence closure models considered by Gildeh *et al.* (2014) were the realizable k-ε and LRR models. The geometrical and flow properties of the inclined buoyant jet were determined and compared with experimental data. For both methods, an accurate prediction was obtained for the jet geometry and dilution characteristics, in which the LRR model conducted to slightly better results. Zhang *et al.* (2016) considered LES with both Smagorinsky and Dynamic Smagorinsky SGS models to simulate numerically 45° inclined dense jets. Considering experimental data, the geometrical characteristics of the jet were precisely predicted. However, the simulation under predicted dilution around 20% compared to experimental results. In comparison to integral models analyzed by Palomar *et al.* (2012), the turbulence closure model applied in his simulations showed better precision.

In the present study, due to its accuracy, the LES approach was applied to numerically simulate a submerged inclined dense jet with 30° inclination. In addition to it, the influence of a white noise generator was considered in the modeling of the turbulent inflow conditions. The application of a random inflow condition was made necessary to impose a more physically consistent boundary condition at the inlet.

2. METHODOLOGY

2.1. Physical model

In the present work, the schematic representation of the domain is shown in Fig. 1. It can be observed from Fig. 1 that the jet is located near the tank floor and the dimensions established for the length, height and width of the domain were defined according to the jet geometrical characteristics. Moreover, important parameters on the inclined jet are: velocity (U_0), jet discharge density (ρ_s) and diameter (D). In order to estimate jet dispersion characteristics, the jet densimetric Froude number (Fr_d) can be considered. The densimetric Froude number represents the ratio of inertial and buoyancy and is obtained by:

$$Fr_d = \frac{U_0}{\sqrt{g_0' D}} \tag{1}$$

$$g_0' = \left(\frac{\Delta\rho_0}{\rho_a}\right) g \tag{2}$$

in which g is gravitational acceleration, $\Delta\rho_0 = \rho_s - \rho_a$, and ρ_a is the water density before the jet release. Other important characteristics of this phenomenon is the momentum (L_m) and source length scales (L_Q). Furthermore, the transport phenomena is analyzed considering: maximum terminal rise height y_t, jet centerline peak horizontal and vertical distances (x_m, y_m), and return location (x_r). Moreover, on the return location and at the centerline peak the dilution is taken in consideration to establish the jet dilution and, consequently, the environmental impacts.

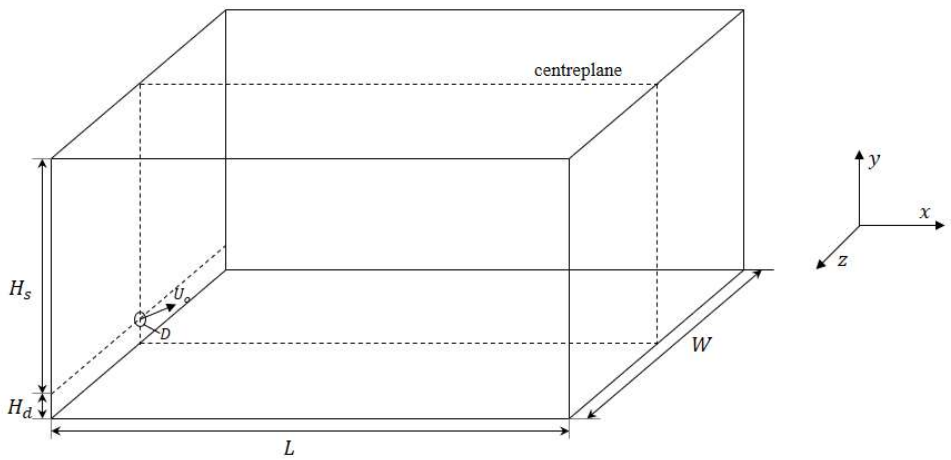

Figure 1. Schematic representation of the computational domain.

With the purpose to simulate the advective and diffusive transport of the dense jet, the following conditions were considered: monophasic flow; incompressible flow; the physical properties were defined constant but the density was considered variable due to the brine dispersion; the fluid was assumed Newtonian or, in other words, the shear stress was proportional to the strain rate; and the flow was considered isothermal.

2.2. Mathematical model

The turbulence closure model considered to study the inclined dense jet phenomena is based on the methodology proposed by Smagorinsky (1963). This approach is known as Large Eddy Simulation (LES). The filtered continuity, momentum and concentrations transport equations for LES are:

$$\frac{\partial \bar{\rho}}{\partial \mathrm{t}}+\frac{\partial(\bar{\rho}\tilde{u}_i)}{\partial x_i}=0, \tag{3}$$

$$\frac{\partial(\bar{\rho}\tilde{u}_i)}{\partial t}+\frac{\partial(\bar{\rho}\tilde{u}_i\tilde{u}_j)}{\partial x_j}=-\frac{\partial \bar{p}}{\partial x_j}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \tilde{u}_i}{\partial x_j}+\frac{\partial \tilde{u}_j}{\partial x_i}\right)\right]+\bar{\rho}g_i-\frac{\partial(\tau_{ij})}{\partial x_j}, \tag{4}$$

$$\frac{\partial(\bar{\rho}\tilde{c})}{\partial t}+\frac{\partial(\bar{\rho}\tilde{u}_j\tilde{c})}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\Gamma\frac{\partial \tilde{c}}{\partial x_j}\right]-\frac{\partial \mathrm{Q_j}}{\partial x_j}. \tag{5}$$

In Eqs. 1-3 ∂ is the differential operator, ρ is the fluid density, u_i is the component i of the velocity vector, t is the time, x_i is the spatial component in the i direction, p is the pressure, μ is the dynamic viscosity of the fluid, g_j is the gravitational acceleration, c is the scalar concentration, Γ is the scalar diffusivity, $\tau_{ij}=\bar{\rho}\widetilde{u_i u_j}-\bar{\rho}\tilde{u}_i\tilde{u}_j$ represents the SGS Reynolds stresses, $Q_j=\bar{\rho}\widetilde{c u_j}-\bar{\rho}\tilde{c}\tilde{u}_j$ represent the SGS scalar flux, the overbar indicates time averaged variables and the tilde indicates spatially filtered variables.

In the turbulence closure model developed by Smagorisnky (1963), the following formulations to model the SGS stress tensor and the SGS turbulent concentration flux are considered:

$$\tau_{ij}=-2\mu_t\tilde{S}_{ij}, \tag{6}$$

$$\mathrm{Q_j}=-\frac{\mu_t}{Sc_t}\frac{\partial \tilde{c}}{\partial x_j}, \tag{7}$$

where μ_t is the turbulent viscosity, $\tilde{S}_{ij}$ is the rate of strain tensor for the resolved scale and Sc_t is the turbulent Schmidt number. In this turbulence closure model, the Sc_t can be evaluated as a constant of 0.7 (Zhang *et al.*, 2016). Therefore, the only variable remaining is the turbulent viscosity, which can be determined by:

$$\mu_t=\rho(C_s\Delta)^2|\tilde{S}|, \tag{8}$$

$$|\tilde{S}|=\sqrt{2\,\tilde{S}_{ij}\tilde{S}_{ij}}, \tag{9}$$

where Δ is the LES filter width that is obtained by the mesh and C_s is the Smagorinsky constant that can be calibrated between values of 0.05 and 0.30.

The Smagorinsky model was detailed and can be applied to model the dense jet phenomena. Another approach considered to analyze this transport phenomenon was the one proposed by Germano *et al.* (1991) and further modified by Lilly (1992). In the approach developed by Lilly (1992), the same equations are applied to model the flow. However, the C_s is calculated dynamically via the application of a second filter in the Eq. 2.

In order to model a turbulent inflow boundary condition, an approach called white noise generator was considered. The white noise generator consists of the superimposition of a white noise over the mean velocity profile. In this method, it is applied a random signal with zero mean and unity variance to impose determined variations to the mean velocity profile.

2.3. NUMERICAL MODEL

The simulations were performed in a reduced domain of dimensions 0.64×0.32×0.16 m^3 (L×W×H) and the initial height of the jet was specified at $0.01\ m$ from the tank floor. Moreover, other parameters of discharge were: jet diameter of 6.5 mm, jet initial velocity of $0.356\ m/s$, discharge angle of 30°, jet density of $1017.8\ kg/m^3$, and local density of $998.0\ kg/m^3$. For the fluid properties, it was defined a molecular viscosity of $0.001\ Pa.s$ and a scalar diffusivity of $1.48\ x\ 10^{-9}\ m^2/s$.

The turbulence closure models applied to study the dense jet phenomenon were the Dynamic sub-grid model proposed by Germano *et al.* (1991) and the Smagorinsky sub-grid model. The mixing and dispersion phenomena were studied considering three simulations:

- Simulation 1: Dynamic sub-grid model, a constant Sc_t of 0.6 without turbulent inflow condition.
- Simulation 2: Dynamic sub-grid model, a constant Sc_t of 0.3 without turbulent inflow condition.
- Simulation 3: Smagorinsky sub-grid model ($C_s = 0.10$), a constant Sc_t of 0.6 with a turbulent inflow condition. A fluctuation of 20% of the average velocity profile was considered in the x and y directions.

As aforementioned, the problem was simulated in the MFSim software. Thus, a finite volume multilevel method was considered for the equations' discretization. In addition, the implementation was carried out through multi-block structured meshes with variable time steps. Regarding the numerical methodology, the Fractional Steps Method was selected for the pressure-velocity coupling. The scheme adopted for the spatial discretization of advective terms in the transport equations was the Barton scheme. Navier-Stokes and the scalar transport equations were solved using a multigrid-multilevel solver. The convergence criterion of 10^{-6} was set for the continuity, momentum, and scalar equations. At last, the time step was adjusted to ensure a Courant number of 0.5.

In order to model this physical phenomenon with lower computational resources, the resort to adaptive mesh refinement method (AMR) was considered. The advantage of applying this strategy is to update a non-uniform mesh according to the flow dynamics and the dense jet dispersion, considering a certain parameter as an indicator function. The vorticity and the scalar, being the last related to fluid density, are used as a refinement criterion in this case study. Thus, it was stablished a mesh size of 1.0 mm for Simulation 1 and 2 and a mesh size of 0.65 mm for Simulation 3.

The simulations were performed up to 24 s, with a real time computing duration exceeding 20 days with 32 cores and were carried out in the Fluid Mechanics Laboratory at the Federal University of Uberlândia, Brazil.

3. RESULTS AND DISCUSSION

As mentioned before, the dispersion phenomena was simulated considering the AMR method to refine regions of interest. Taking into account the vorticity and the scalar properties, the mesh was updated according the plume dispersion. In Fig. 2 it is possible to observe the contours of density and the mesh for 1.0 s and 16.0 s of discard.

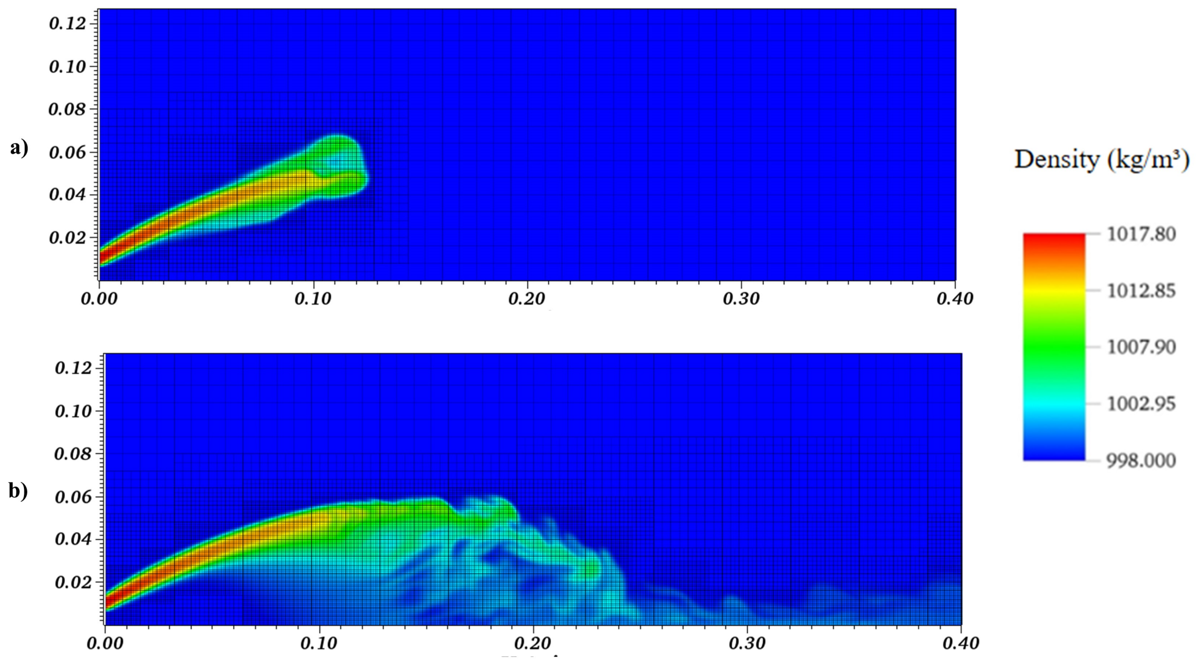

Figure 2. Instantaneous density contours in Simulation 2. a) $t = 1.0\ s$; b) $t = 16.0\ s$.

From Fig. 2, it can be affirmed that the computational cost at the initial stages of discharge was lower due to the lower number of computational cells required to simulate the phenomenon at that time. It is important to mention that, in simulation 1 and 2, approximately 11 million computational cells were necessary to simulate the phenomenon after it achieved the statistical permanent regime. Moreover, approximately 20 million computational cells were necessary in simulation 3.

The results obtained can be observed in Tab. 1. In order to analyze the phenomena it was considered: x_m and y_m are the horizontal and vertical locations of the centerline peak, y_t is the terminal rise height, x_t is the return point location. The dilution was evaluated at the centerline peak (S_m) and at the return point location (S_r). These variables were considered to obtain the coefficients that are commonly applied to compare experimental and numerical results. The experimental values defined in Tab. 1 were taken from the study performed by Gildeh *et al.* (2014) considering a 30° inclined dense jet. Furthermore, the relative errors of each coefficient, taking as reference the experimental data, is showed between parentheses.

Table 1. Comparison of experimental and numerical results.

Coefficients	Experiments	Simulation 1	Simulation 2	Simulation 3
x_m/L_m	2.00	2.0410 (1.02)	2.2205 (1.11)	1.959 (0.98)
y_m/L_m	0.70	0.6858 (0.98)	0.7021 (1.01)	0.686 (0.98)
x_r/L_m	3.50	3.3749 (0.96)	3.7553 (1.07)	3.339 (0.96)
y_t/L_m	1.13	1.1103 (0.99)	1.0613 (0.94)	1.061 (0,94)
S_m/Fr	0.65	0.1902 (0.29)	0.1978 (0.30)	0.332 (0.51)
S_r/L_m	1.27	0.8009 (0.63)	0.7551 (0.59)	0.972 (0.77)

From Tab. 1 it can be affirmed that precise results were obtained considering the jet centerline trajectory. The geometrical parameters associated with the centerline peak, return point of the jet centerline and the highest height achieved by the plume were normalized by the length L_m. Considering these locations the better prediction was obtained applying the Dynamic sub-grid model with a $Sc_t = 0.6$. Moreover, in the simulation with a $Sc_t = 0.3$, the coefficients showed that the geometrical characteristics were slightly over-predicted. In contrast, the geometrical coefficients were slightly under-predicted by the use of a turbulent inflow condition. It is important to mention that for Simulation 3, a calibration on the Smagorinsky constant (C_s) could be considered. With the calibration of the C_s, it would be possible to achieve better results considering these geometrical parameters.

The dilution normalized by the densimetric Froude number was evaluated at the jet centerline peak and at the return point location. It is important to observe that the dilution was obtained by the initial concentration at the source over the local concentration. The LES simulations with no inlet turbulent condition under-predicted considerably the dilution coefficient at the return point and at the peak of the centerline trajectory. Moreover, the imposition of a greater turbulent diffusivity on the scalar transport equation in simulation 2 did not affect the scalar transport phenomena. The results obtained considering a turbulent inflow condition showed the best results. In Fig. 3, it is possible to observe how a turbulent inflow condition changes the jet dispersion phenomena.

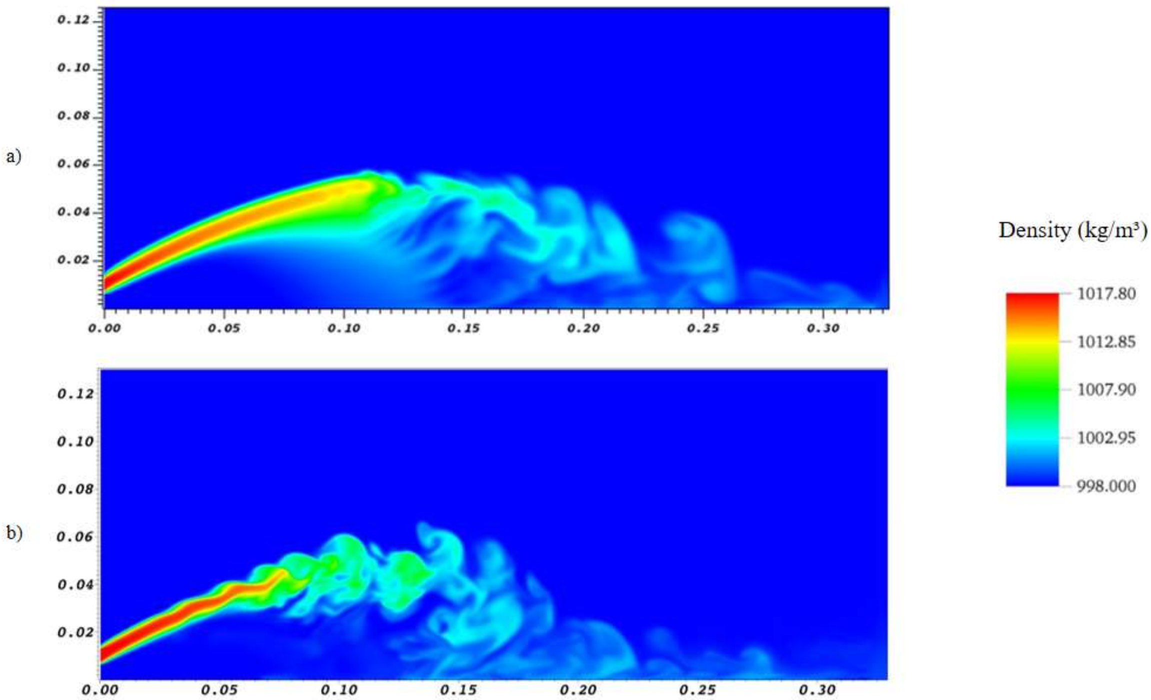

Figure 3. Instantaneous density contours. a) Simulation 1; b) Simulation 3.

It is possible to affirm based on Fig. 3 that the white noise imposed on the inlet (Simulation 3) resulted in a less diffusive flow. Due to this dispersion characteristic, the transition stage occurs closer to the jet inlet and a higher dilution is observed at the jet centerline trajectory peak.

It is important to mention that the geometrical coefficients on Table 1 were calculated considering the velocity average field. In addition to it, a greater distance on the return point location considering the density and velocity average fields was observed only for Simulation 3. For the simulations 1 and 2, this difference on the return point location considering the density and velocity average fields was negligible.

In order to establish the environmental impact of an inclined jet dispersion operation in desalination plants, the results observed in this study are satisfactory. It is important to mention that low values of dilution are directly correlated to higher environmental impacts.

4. CONCLUSIONS

The phenomena of inclined dense jet was modelled considering LES approaches and a sub-grid scalar flux model with constant turbulent Schmidt number. It was noted that the use of turbulent inflow conditions at the inlet is necessary to obtain better results considering experimental data as reference. In other words, it is not possible to accurately reproduce the phenomena without applying a turbulent inflow condition, since the jet became too diffusive without it. The results show that the methodology applied in the software MFSim is adequate to study these dispersion phenomena and accurate information of the phenomena can be calculated to ensure lower environmental impacts.

5. ACKNOWLEDGEMENTS

The authors would like to thank the support provided by CAPES and PETROBRAS.

6. REFERENCES

Abessi, O., Roberts, P.J.W., 2015, "Effect of nozzle orientation on dense jets in stagnant environments". Journal of Hydraulic Engineering, v. 141.

Cipolina, A., Brucato, A., Grifasi, F., Nicosia, S., 2005, "Bench-scale investigation of inclined dense jets". Journal of Hydraulic Engineering, v. 131, pp. 1017-1022.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1991, "A dynamic sub-grid-scale eddy viscosity model". Physics of Fluid Dynamics, 3, pp. 1760-1765.

Gildeh, H.K., Mohammadian, A., Nistor, I. and Qiblawey, H., 2014, "Numerical modeling of 30º and 45º inclined dense turbulent jets in stationary ambient". Environmental Fluid Mechanics, 15.

Lilly, D., 1992, "A proposed modification of the Germano subgrid-scale closure method". Physics of Fluids A: Fluid Dynamics, v. 4.

Malcangio, D. and Petrillo, A., 2010, "Modeling of brine outfall at the planning stage of desalination plants". Desalination, 254, No. 1, pp. 114-125.

Palomar, P., Lara, J., Losada, I., Rodrigo, M. and Alvarez, A., 2012, "Near field brine discharge modelling part 1: Analysis of commertial tools". Desalination, 290, pp. 14-27.

Roberts, P.J.W., Ferrier, A., Daviero, G., 1997, "Mixing in inlined dense jets". Journal of Hydraulic Engineering, v. 123, pp. 693-699.

Shao, D., Law, A., 2010, "Mixing and boundary interactions of 30° and 45° inclined dense jets". Environmental Fluid Mechanics, v. 10, pp. 521-553.

Smagorinsky, J., 1963, "General circulation experiments with primitive equations". Monthly Weather Review, Amrican Meteorological Society", v. 91, n. 3, pp. 99-164.

Zeitoun, M. A. mcHilhenny, W.F., Reid, R.O., 1970, "Conceptual designs of outfall systems for desalination plants". Research and development progress Office of Saline Water. United States Department of the Interior,Washington D.C.

Zhang, S., Jiang, B., Law, A. W., Zhao, B., 2016, "Large Eddy Simulation of 45º inclined dense jets". Environmental Fluid Mechanics, 16, pp. 101-121.

7. RESPONSIBILITY NOTICE

The authors are the only ones responsible for the printed material included in this paper.