



The University of Bradford Institutional Repository

<http://bradscholars.brad.ac.uk>

This work is made available online in accordance with publisher policies. Please refer to the repository record for this item and our Policy Document available from the repository home page for further information.

To see the final version of this work please visit the publisher's website. Access to the published online version may require a subscription.

Link to publisher's version: [http://dx.doi.org/10.1061/\(ASCE\)HY.1943-7900.0000903](http://dx.doi.org/10.1061/(ASCE)HY.1943-7900.0000903)

Citation: Guo Y (2014) Numerical simulation of the spreading of aerated and nonaerated turbulent water jet in a tank with finite water depth. *Journal of Hydraulic Engineering*. 140(8)

Copyright statement: © 2014 ASCE. Full-text reproduced in accordance with the publisher's self-archiving policy.

1 Numerical simulation of the spreading of aerated and nonaerated turbulent water jet in
2 a tank with finite water depth

3 Yakun Guo¹

4

5 **Abstract:** Numerical simulations are carried out to investigate the spreading of two-
6 dimensional plane turbulent aerated and nonaerated jets in a tank filled with finite
7 water depth. A multiphase model is applied to simulate the problem under
8 investigation. The governing equations, their numerical scheme and the boundary
9 conditions are presented. Aerated and non-aerated turbulent jets are simulated for a
10 range of the jet velocity and width at exit, the initial air content at exit and the water
11 depth in tank. The simulated results show that a self-similar Gaussian velocity
12 distribution exists from the distance downstream being larger than five jet slot width
13 for both the aerated and nonaerated jets. Good agreement between the simulated
14 velocity profiles and available laboratory experiments is obtained. The simulated
15 slope of the jet velocity decay along the jet centreline is in good agreement with the
16 experimental measurements. The effect of air content on pressure distribution and the
17 maximum impinging hydrodynamic pressure at the tank bottom is discussed.

18

19 Key words: numerical simulation; turbulence; jet; air content

20

21

22 **Introduction**

23 Plunge pool scour generated by free trajectory jets is one of key problems in the
24 design and operation of a hydro scheme. The development of plunge pool scour can

¹Reader, School of Engineering, University of Aberdeen, Aberdeen, AB24 3UE, UK. Email:
y.guo@abdn.ac.uk.

25 endanger the foundation and abutment of a dam. The erosion of plunge pool is closely
26 related to the hydraulic energy dissipation in the plunge pool. There are many energy
27 dissipation means. For example, the high velocity water jet from a slot in a dam or
28 from a flip bucket and the waterfall over the spillway is among them. These energy
29 dissipation means have many advantages, such as economy, simple engineering
30 structure and a wide suitability for both the discharge and water depth downstream.
31 Therefore, these energy dissipation means have been widely used in the medium and
32 high dams. However they also present a challenge task to their designers. As highly
33 turbulent water jet travels through atmosphere, it entrains air into it and becomes a
34 mixture of air-water prior to impinging into plunge pool downstream. Studies showed
35 that only 10-20 percent jet energy is dissipated during the trajectory process through
36 the atmosphere (Elevatorski 1959); most jet energy is dissipated within plunge pool.
37 Therefore, understanding of the mechanism of energy dissipation within plunge pool
38 can improve the prediction of the erosion and scour. As free water jet becomes an air-
39 water two-phase flow prior to entering pool, it is important to accurately estimate the
40 effect of air entrained into the jet on the energy dissipation in plunge pool. Such
41 energy dissipation is closely related to the spreading of the jet in plunge pool. This is
42 the motivation of this study in which we aim to advance our knowledge and
43 understanding of the effect of air content on the spreading of a jet in plunge pool.

44

45 Due to its practical importance, many laboratory experiments have been conducted to
46 investigate the scour depth in plunge pool during the past decades. Several empirical
47 formulas for predicting plunge pool scour depth have been proposed based on both
48 the laboratory experiments and some prototype data (see, for example, Martins 1975;
49 Rajaratnam and Beltaos 1977; Mason 1984; Mason and Arumugam 1985; Bormann

50 and Julien 1991; Hoffmans 1998). The calculated results from these formulas,
51 however, are different from each other (Mason and Arumugam, 1985). Such
52 difference may be ascribed to the fact that most formulas only considered the effect of
53 jet fall height and discharge per unit width, the characteristic size of bed materials,
54 takeoff jet angle and tailwater depth on the scour depth, but did not take the influence
55 of air content into account when evaluating the scour depth. In practical situation,
56 turbulent free water jet becomes a two-phase flow (air-water mixture) prior to
57 entering into water downstream as it entrains considerable air into it during its
58 trajectory (Ervine *et al.* 1980). The study of Mason (1989a, b) indicated that the air
59 entrained by turbulent free water jet should be taken as an additional parameter in the
60 estimation of plunge pool scour. His study showed that the air content increased scour
61 depth. However, his formula over-estimated scour depth when it was applied to the
62 prototype data. The effect of air content on the scour depth has also been recently
63 investigated by Bollaert and Schleiss (2003b); Canepa and Hager (2003); Xu *et al.*
64 (2004) and Pagliara *et al.* (2006, 2008). The study of Xu *et al.* (2004) shows that for a
65 rectangular jet, when both the water flow rate and air-water mixture jet velocity for
66 aerated jet are the same as those of non-aerated jet, the scour depth is decreased with
67 the increasing of air content. In their comparison, to keep the same water flow rate
68 and jet velocity, aerated jet width is obviously larger than that of non-aerated jet.
69 Canepa and Hager (2003) also indicated that caution should be taken which velocity –
70 air-water mixtures or pure water – is used when evaluating the effect of air content on
71 scour depth. For rectangular jets, which are typical of spillway discharge (Puertas and
72 Dolz 2005), the formula presented by Ervine (1976) shows that the amount of air
73 entrained by free jet with high velocity and large fall height is very large. More

74 studies on plunge pool scour by a trajectory jet can be found in recent review papers
75 by Hager (2007) and Bollaert and Schleiss (2003a).

76

77 Comparing with extensive laboratory experimental studies, relatively few numerical
78 investigations have been conducted to evaluate the scour generated by free-falling jet.
79 Jia *et al.* (2001) investigated the scouring process in plunge pool using CCHE3D
80 model. Salehi Neyshabouri *et al.* (2003) carried out the similar study using a two-
81 dimensional (2D) numerical model. Both studies did not examine the effect of jet air
82 content on scour. As indicated by Jia *et al.* (2001), the pressure fluctuation, which is
83 closely related to the velocity field, plays an important role in plunge pool scour.
84 Therefore, this study is to examine the effect of jet air content on plunge pool scour
85 using numerical simulations. We will focus on simulating the velocity and pressure
86 field and spreading of aerated and non-aerated jets. To this end, a multiphase model is
87 employed and described as following.

88

89 **Multiphase model**

90 The multiphase model embedded in FLUENT (ANSYS 12.0, 2009) is applied to
91 simulate the effect of air content on the spreading of the falling water jet in plunge
92 pool. Volume of fluid (VOF) is used in the simulation. In VOF models, water
93 (primary phase) and air (secondary phase) share the same velocity and pressure field,
94 therefore, a single set of momentum and continuity equations in conservative form is
95 used to describe the flow. For convenience, a brief description is given as following.

96

97 ***Governing equations***

98 The governing equations solved for each phase in the multiphase model can be
 99 written in a Cartesian coordinate system (shown as in Figure 1) as following:

100 Continuity equation:

$$101 \quad \frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (1)$$

102 Momentum equation:

$$103 \quad \frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[(\mu + \mu_t) \cdot \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (2)$$

104 k -equation:

$$105 \quad \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon \quad (3)$$

106 ε -equation:

$$107 \quad \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (4)$$

108 where ρ , μ = density and dynamic viscosity of air-water mixture, respectively; t =
 109 time; u_i = component of velocity in the x_i -direction; p = pressure; k = turbulent kinetic
 110 energy (TKE), ε = rate of dissipation of TKE, μ_t = turbulent (or eddy) viscosity, σ_k , σ_ε
 111 = turbulent Prandtl number for k and ε , respectively; G_k = TKE produced by the mean
 112 velocity gradients, G_b = TKE produced by buoyancy.

113

114 The turbulent viscosity can be determined using the turbulent kinetic energy (k) and
 115 its dissipation rate (ε):

$$116 \quad \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (5)$$

117 The values of the constants in above equations are (Rodi 1993): $\sigma_k = 1.0$; $\sigma_\varepsilon = 1.3$;
 118 $C_\mu = 0.09$; $C_{1\varepsilon} = 1.44$; and $C_{2\varepsilon} = 1.92$.

119 The term of turbulent kinetic energy produced by the mean velocity gradients G_k can
120 be determined by

$$121 \quad G_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \quad (6)$$

122 The density and viscosity of air-water mixture is a function of the volume fraction and
123 can be determined as (ANSYS 12.0, 2009):

$$124 \quad \rho = (1 - \beta_0) \rho_w + \beta_0 \rho_a \quad (7)$$

$$125 \quad \mu = (1 - \beta_0) \mu_w + \beta_0 \mu_a \quad (8)$$

126 where β_0 = volumetric fraction of air; ρ_w, ρ_a = density of water and air, respectively;
127 μ_w, μ_a = viscosity of water and air, respectively.

128

129 *Numerical scheme*

130 Figure 1 shows the computational domain. To improve the calculation accuracy and
131 reduce the computational time, the unstructured non-uniform triangular meshes are
132 used in the computational domain. This allows the locally refining the concerned
133 regions (e.g. near the jet core and the region near the tank bottom) with small meshes
134 and has advantage of flexibly assigning meshes in the computational domain (Guo et
135 al. 2008, 2012). The sensitivity of mesh size was investigated by adapting and
136 refining the meshes until no significant changes in the solution were achieved.
137 Meanwhile, the effect of meshes on the convergence of numerical simulations was
138 also examined. The final meshes used in the simulation had 43275 (for shallow water)
139 ~58277 (for deep water) nodes and 85248 (for shallow water) ~ 115200 (for deep
140 water) cells, with the minimum 0.002m grid size near the jet core and tank bottom and
141 the maximum 0.004m grid size in other regions. The second order implicit method is
142 applied for temporal discretization, while highly stable power-law differencing is used

143 for spatial discretization of governing equations. The phase coupled SIMPLE (PC-
144 SIMPLE) is applied for pressure-velocity coupling (Vasquez and Ivanov 2000). To
145 speed up the convergence of simulation, the under-relaxation technique was used by
146 changing the under-relaxation factor during the calculation. This was done carefully
147 so that no divergence or undue instability occurred (Guo *et al.* 2007).

148

149 ***Boundary conditions***

150 At the inlet boundary, average velocity and jet slot width are specified according to
151 the laboratory experiments (Guo and Luo 1999). Turbulent kinetic energy (k) and its
152 dissipation rate (ε) at the inlet boundary are calculated as (Jing *et al.* 2009):

$$153 \quad k = 1.5(IU_0)^2 \quad (9)$$

$$154 \quad \varepsilon = c_\mu^{3/4} \frac{k^{3/2}}{l} \quad (10)$$

155 where I = turbulent intensity and taken as 10% in this study; U_0 = average velocity at
156 the inlet (see Figure 1); l ($= 0.07 R$) = turbulence length scale, and R = the hydraulic
157 radius at the inlet and taken as the jet slot width d . For aerated jet, the initially
158 prescribed air content, thus the flow rate weighting, is specified at the inlet while the
159 tank is filled with pure water. At the free water surface, the atmospheric pressure is
160 applied and adjusted according to the air-water flow rate weighting in the simulation.
161 At the two outlets (see Figure 1), the pressure outlet boundary condition is specified
162 in which a static pressure at the outlet boundary is realized. . On all solid boundaries,
163 including side walls and the bottom of tank, no-slip boundary condition is applied.

164

165 **Model validation**

166 The multiphase model is validated using the laboratory experiments of Guo and Luo
167 (1999). Though the experimental details can be found in Guo and Luo (1999), we
168 present a brief description for convenience and completeness. The experiments were
169 conducted in a tank of 34 cm wide and 180 cm long with changeable water depth.
170 Two water depths of 29 cm and 39 cm were used in the experiments while several
171 water depths were investigated in numerical simulations. Jet velocity at exit was
172 maintained as constant throughout the experiments by constant water head. For
173 aerated jet, air with prescribed flow rate was fed into pressure relief chambers and a
174 box with small holes upstream by an air compressor. Therefore, air had moved a
175 distance and uniformly mixed with jet water prior to entering into tank. To avoid extra
176 air entrained by jet into tank at the water surface, jet was introduced immediately
177 below the water surface so that the influence of air content could be effectively
178 evaluated. The hydrodynamic pressure at tank bottom was measured using Multi-
179 point Pressure Scanner manufactured by Scanivalve of USA with the accuracy of
180 $\pm 0.5\%$ and pressure range of 0.007 m to 10 m (water height). The Scanner was linked
181 with 23 manometer tubes with the inner diameter of 1 mm. The distances between
182 two tubes varied from 5mm to 20mm. The experimental parameters were: air content
183 (defined as the ratio of air volume to air-water volume) was $\beta_0 = 27\% \sim 44\%$; the
184 mean jet velocity U_0 at exit was from 3.4 m/s to 5.7 m/s; the slot width of the pure
185 water jet (d) was 1.0 cm, 1.6 cm, 2 cm, 2.5 cm, 3 cm, 4 cm and 5 cm, respectively.
186 For aerated jet, the equivalent water width at exit is the total width of the air-water
187 mixture times $(1 - \beta_0)$. The corresponding Reynolds number (defined as $Re = U_0 d / \nu$)
188 was from 54400 to 285000 so that jet was completely turbulent flow (Fischer *et al.*
189 1979).

190

191 **Results and discussions**

192 *Spreading of jet*

193 The spreading of a jet determines the extent of scour hole caused by jet. The
194 spreading of a jet can be evaluated by the angle of jet spreading and the velocity
195 profile at a distance from jet exit. Two jet spreading scales are used and investigated.
196 The first is the half jet width at which velocity drops to the half of the jet centreline
197 velocity while another scale is the jet spreading angel determined using the jet
198 boundary whose velocity is 5% of the jet centreline velocity at the same distance from
199 exit. For pure water jet, the simulated averaged half jet width over the distance being
200 greater than 6 times of jet width at exit is about 0.102 of that distance. This value
201 agrees well with the experimental measurements of Kuang *et al.* (2001) (0.1~0.12)
202 and Miozzi *et al.* (2010) (~0.10) for turbulent plane jet. This spreading value is also
203 reasonably compared with other published data for turbulent plane jets (Fischer *et al.*
204 1979; Chu and Lee 1999). The simulated averaged jet spreading angle determined
205 using 5% of centreline velocity for pure water jet is about 9.2 ± 1 degree, which is
206 slightly smaller than the value of laboratory measurements (10~11 degree) (Ervine
207 and Falvey 1987). For aerated jet, the simulated averaged jet spreading angle is
208 12.9 ± 1 degree, which is in good agreement with the measured value 13~14 degree of
209 Ervine and Falvey (1987).

210

211 Simulations were also run for a range of jet slot width, Reynolds number, tank water
212 depth and air content to examine their effects on jet spreading. The results indicate
213 that no significant effect of such parameters on the spreading of jet.

214

215 *Velocity profile*

216 It is well known that the velocity profile at the jet cross section being more than six
217 times of jet diameter downstream has a self similar form and Gaussian distribution for
218 a pure water jet (Fischer *et al.* 1979). For the problem under investigation, the water
219 depth in tank is relatively shallow and jet diffusion in water is restricted. Vortices are
220 generated near the tank bottom at both sides of jet, affecting velocity field. For aerated
221 jet, air content may also play a role in jet velocity profile. To examine if the velocity
222 profile for aerated jet fits a Gaussian distribution, numerical simulations have been
223 performed covering a range of jet Reynolds numbers, air contents at exit and water
224 depths in tank. Figure 2 is a typical example of the simulated and measured velocity
225 profiles for aerated jet: $\beta_0=27\%$, $Re=58,400$, $H/d=26.7$. In Figure 2, velocity is
226 normalized using the local centreline (maximum) velocity while horizontal distance is
227 normalized by local vertical distance from exit. To examine the self –similar Gaussian
228 distribution, simulated velocity profiles at three downstream distances ($z/d=3.7$; 6.5
229 and 16) are plotted in Figure 2. It is seen that when jet is in the near region from slot
230 ($z/d=3.7$); normalized velocity distribution shows a top hat velocity profile,
231 demonstrating that jet is still in the zone of flow establishment (Fischer *et al.* 1979).
232 The velocity profiles at the downstream distance being larger than six slot width,
233 however, demonstrate perfect self –similar Gaussian distribution. Numerical runs
234 performed for various air contents (up to 50%), Reynolds numbers and tank water
235 depths reveal similar results to Figure 2, indicating that air content has little effect on
236 the self-similarity of jet velocity profile for the flow conditions simulated. In all
237 numerical simulations performed no air bubbles within jet move upwards and escape
238 from tank as their downward velocity is larger than the critical velocity of 0.26m/s
239 (Mckeogh and Ervine 1980). Good agreement between the measured and simulated

240 velocity profiles demonstrates that the model is capable to calculate the spreading of
241 aerated jet in a tank with finite water depth.

242

243 Numerical simulations carried out for non-aerated jet for a range of Reynolds
244 numbers and tank water depths also reveal similar results. Figure 3 is a typical
245 example of the simulated velocity profile at various water depths for $Re=80,000$ and
246 water depth in tank $H=39$ cm ($H/d=19.5$). The solid line is the averaged velocity
247 profiles at four positions whose downstream distance (z/d) is greater than 5 slot
248 widths and smaller than 15 slot widths. The results show that the self-similar Gaussian
249 profile is valid for z/d being greater than 5. A top hat velocity profile is also found for
250 $z/d < 4$ where flow is in the zone of flow establishment. When velocity profile is taken
251 at $z/d=17$, which is close to the tank bottom (at tank bottom $z/d= H/d =19.5$), the
252 boundary edge of jet is influenced by the vortices formed there. In general, the
253 numerically simulated velocity profile is in good agreement with the experimental
254 measurements.

255

256 *Velocity decay*

257 Local maximum velocity, usually occurring at the jet centreline, is a key parameter
258 which primarily determines the plunge pool scour depth. As jet water flows
259 downstream, jet expands due to the ambient water entrained into it and its velocity
260 decreases. It is of engineering importance to investigate how the jet parameters
261 influence the decay of the jet centreline velocity. Figure 4 shows the variation of the
262 simulated and measured centreline jet velocity U_m normalized by the velocity at exit
263 U_0 with the dimensionless distance from the jet exit. Some experimental data by other
264 investigators are also plotted in Figure 4 for comparison. For aerated jet, the jet width

265 used in the figure is the equivalent pure water width. It is seen that the simulated jet
266 centreline velocity decay with downstream distance from exit agrees well with the
267 laboratory experiments of Guo and Luo (1999). This may not be surprised as the flow
268 parameters and geometry used in the numerical model are identical to those in
269 experiments. The results also show that the air content has insignificant effect on the
270 jet centreline velocity decay.

271

272 Comparison of this numerical simulation with the experimental results of Miozzi *et*
273 *al.* (2010) and Kuang *et al.* (2001) reveals that relatively large discrepancy between
274 measurements and simulation exists. In both comparison cases, the numerical model
275 underestimates the decay of centreline velocity with downstream distance. In
276 particular, experiments by Kuang *et al.* (2001) shows a rapid decay of the centreline
277 velocity with distance while numerical simulation demonstrates gradual decrease of
278 the centreline velocity. This discrepancy between numerical simulation and
279 experiments may be ascribed to the different boundary conditions as well as different
280 geometry (Kim and Choi 2009). When jet is bounded in all directions (the situation of
281 the present work), a slower decrease of velocity is expected (Miozzi *et al.* 2010).

282

283 Though there exists relatively large deviation between the numerical simulation and
284 the experimental measurements of Kuang *et al.* (2001), the slope of the centreline
285 velocity decay with distance is similar. This can be revealed by expressing the
286 variation of the jet centreline velocity with downstream distance for a plane turbulent
287 jet as:

$$288 \quad \frac{U_m}{U_0} = k \left(\frac{d}{z} \right)^{1/2} \quad (11)$$

289 where coefficient k determines the speed of the jet centreline velocity decay. The
290 value of k reported by Guo and Luo (1999) is 2.75 for nonaerated jet and 2.87 for
291 aerated jet. For a pure water jet, Beltaos (1976) had a value of 2.72; Davies *et al.*
292 (1975) obtained a value of 2.62; Fischer *et al.* gave a value of 2.41 while Gutmark and
293 Wagnanski's experiments (1976) showed a value of 2.4. The recent study of Miozzi
294 *et al.* (2010) found a value of 2.35. The present numerical study shows that the value
295 is 2.58 for non-aerated jet and 2.62 for aerated jet; which agrees well with that
296 reported by Davies *et al.* (1975); but is smaller than those of Belatos (1976) and Guo
297 and Luo (1999) and slightly greater than others.

298

299 *Maximum pressure at tank bottom*

300 The maximum pressure at the tank bottom is another key parameter determining
301 plunge pool scour hole depth. Figure 5 shows the variation of the maximum pressure
302 at the tank bottom with the flow Reynolds number for tank water depth of 29 cm (5a)
303 and 39 cm (5b). For aerated jet, two velocities are used to calculate the Reynolds
304 number, namely the water velocity U_0 and air-water mixture velocity $U_{aw}=U_0/(1-\beta_0)$
305 (Canepa and Hager 2003). In both cases, the equivalent slot water width is used in
306 calculating the Reynolds number. This means that for the same U_0 and width of air-
307 water mixture, the Reynolds number will decrease with increase of air content.
308 Experimental results of Guo and Luo (1999) for nonaerated jet are also plotted for
309 comparison. The data is a bit scatter. In general, the maximum pressure at the tank
310 bottom increases with the increase of the flow Reynolds number for both the aerated
311 and non-aerated jet. The simulation is reasonably compared with the experimental
312 measurements of Guo and Luo (1999). Figure 5 demonstrates that when only pure
313 water jet velocity is used in aerated jet, the maximum pressure for aerated jet is larger

314 than that of nonaerated jet for the same Reynolds number. This is because the
315 equivalent slot water width for aerated jet is smaller than that of nonaerated jet, thus
316 leading to the decrease of the Reynolds number. Numerical runs also reveal that the
317 maximum pressure increases with the increase of air content provided that the water
318 flow rate and the slot width remains unchanged. In this situation the added air will
319 increase jet velocity from U_0 to $U_0/(1-\beta_0)$. However, when the aerated jet velocity is
320 the same as the pure water jet velocity, the maximum pressure of aerated jet is smaller
321 than that of nonaerated jet due to the decrease of the density of aerated jet. This
322 conclusion is consistent with that of Canepa and Hager (2003).

323

324 *Effect of air content on pressure distribution at the tank bottom*

325 Figure 6 shows the effect of air content on the pressure distribution at the tank bottom
326 for jet slot width of 1.6 cm, $U_0=3.4\text{m/s}$ and $\beta=0, 27\%, 36\%$ and 44% , respectively. In
327 numerical simulation, jet velocity and jet slot width at exit remains unchanged. As
328 such, the increase of air content means the decrease of water fraction in jet, namely
329 the jet density decreases. This causes the decrease of pressure, as shown in Figure 6.
330 Numerical runs were performed for a range of jet velocity and width at exit, water
331 depth in tank and air content at exit, obtaining the similar results shown as in Figure 6.

332

333 **Conclusion**

334 Numerical simulations are performed to investigate the spreading of aerated and
335 nonaerated jet in a tank with finite water depth. Simulations cover a range of jet
336 parameters, such as jet velocity and jet slot width at exit, initial air content at exit and
337 water depth in tank. The results show that the self-similar Gaussian distribution of jet
338 cross sectional velocity profiles exists for the downstream distance which is larger

339 than five jet slot width for both aerated and nonaerated jets. Air content has little
340 influence on velocity profile. The decay of the jet centreline velocity with
341 downstream distance is simulated for a range of flow conditions. Good agreement
342 between the simulation and laboratory measurements with the identical flow
343 conditions and geometry is obtained. Comparison of the numerical simulation with
344 the experimental results of Miozzi *et al.* (2010) and Kuang *et al.* (2001) reveals that
345 the numerical model underestimates the jet centreline velocity decay. This
346 discrepancy between numerical simulation and experiments may be ascribed to the
347 different boundary conditions and geometry (Kim and Choi 2009). In present study,
348 the jet is bounded in all directions, thus, slower velocity decay is expected (Miozzi *et*
349 *al.* 2010).

350

351 The effect of air content on pressure distribution and the maximum pressure at the
352 tank bottom is simulated for various flow conditions. Caution needs to be taken for
353 choosing jet velocity when evaluating the effect of air content on pressure. When the
354 aerated jet velocity and width at exit remains unchanged, increasing air content means
355 the decrease of water fraction in aerated jet. Consequently, density of aerated jet
356 decreases, leading to the decrease of pressure. In practical situation, when air is
357 entrained into jet, jet cross section and the total air-water flow rate will increase.
358 Consequently, the scour hole downstream will become larger, shallower and flatter
359 (Mason 1989a, b).

360

361 *Acknowledgements:* This work is partly supported by the Open Funding from the
362 State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan
363 University (SKHL1302). The author is grateful for Mairi Ng and Megan Johnston

364 who conducted some simulation work as part of their MEng projects at University of
365 Aberdeen. The constructive comments made by three anonymous reviewers have
366 significantly improved the quality of the paper.

367

368 References

369 ANSYS FLUENT (2009). "Theory Guide and User's Guide, version 12.0." ANSYS,
370 Inc. Southpointe, Canonsburg, U.S.

371 Beltaos, S. (1976). "Oblique impingement of plane turbulent jets." *J. Hydr.*
372 *Div.*, 102(9), 1177-1192.

373 Bollaert, E. and Schleiss, A. (2003a). "Scour of rock due to the impact of plunging
374 high velocity jets Part I: a state-of-art review." *J. Hydraul. Res.*, 41(4), 451-
375 464.

376 Bollaert, E. and Schleiss, A. (2003b). "Scour of rock due to the impact of plunging
377 high velocity jets Part II: experimental results of dynamic pressures at pool
378 bottoms and in one- and two-dimensional closed end rock joints." *J. Hydraul.*
379 *Res.*, 41(4), 465-480.

380 Bormann, E. and Julien, P.Y. (1991). "Scour downstream of grade-control structures."
381 *J. Hydraul. Eng.*, 117(5), 579-594.

382 Canepa, S., and Hager, W.H., (2003). "Effect of jet air content on plunge pool scour."
383 *J. Hydraul. Eng.*, 129: 358-365.

384 Chu, P.C.K. and Lee, J.H.W. (1999). An inclined wall jet: mean flow characteristics
385 and effects of acoustic excitation. *J. Hydraul. Eng.*, 125, 193-205.

386 Davies, A.E., Keffer, J.F. and Baines, W.D. (1975). "Spread of a heated plane
387 turbulent jet." *Physics of Fluids*, 18(7), 770-775.

388 Elevatorski, E.A., (1959). Hydraulic energy dissipaters, McGraw-Hill,

389 Ervine,D.A. (1976). “The entrainment of air in water.” *Int. Water Power and Dam*
390 *Constr.*, 28, 27-30.

391 Ervine,D.A., McKeogh, E and Elsayy, E.M., (1980). “Effect of turbulence intensity
392 on the rate of air entrainment by plunging water jet”, *Proc., Inst. Civ. Engrs.*,
393 *Part 2- Research and Theory*, 69, 425-445.

394 Ervine, D.A. and Falvey, H.T., (1987). “Behaviour of turbulent water jets in the
395 atmosphere and in plunge pools.” *Proc., Inst. Civ. Engrs., Part 2- Research*
396 *and Theory*, 83, 295-314.

397 Fischer, H.B., et al., (1979), *Mixing in Inland and Coastal Waters*, New York,
398 Academic Press.

399 Guo, Y.K. and Luo, M. (1999). “Experimental study on the effects of air entrainment
400 on a plane jet diffusion in a plunge pool.” *J. Hydrodynamics*, 11(2), 32-37.

401 Guo, Y.K., Wang, P. and Zhou, H. (2007). “Modelling study of the flow past
402 irregularities in a pressure conduit.” *J. Hydraul. Eng.*, 133(6), 698-702.

403 Guo, Y.K., Zhang, L.X., Shen, Y.M. and Zhang, J.S., 2008. Modelling study of free
404 overfall in a rectangle channel with strip roughness, *J. Hydraul. Eng.*, 134(5),
405 664-668.

406 Guo, Y.K., Wu, X.G., Pan, C.H. and Zhang, J. (2012). “Numerical simulation of the
407 tidal flow and suspended sediment transport in the Qiantang Estuary.” *Journal*
408 *of Waterway, Port, Coastal and Ocean Engineering*, 138: 192-203.

409 Gutmark, E. and Wagnanski, J. (1976). “The planar turbulent jet.” *J. Fluid Mech.*, 73,
410 465-495.

411 Hager, W.H., (2007). “Scour in hydraulic engineering.” *ICE Water Management*,
412 160:159-168.

413 Hoffmans, G.J.C.M. (1998). "Jet scour in equilibrium phase." *J. Hydraul. Eng.*,
414 124(4), 430-437.

415 Jia, Y.F., Kitamura, T., and Wang, S.S.Y., (2001). "Simulation of scour process in
416 plunging pool of loose bed-material." *J. Hydraul. Eng.*, 127, 219-229.

417 Jing, H, Guo, Y.K., Li, C and Zhang, J., (2009). "Three-dimensional numerical
418 simulation of compound meandering open channel flow by Reynolds stress
419 model." *International Journal for Numerical Methods in Fluids*, 59:927-943.

420 Kim, J. and Choi, H. (2009). "Large eddy simulation of a circular jet: effect of inflow
421 conditions on the near field." *J Fluid Mech.* 620: 383-411.

422 Kuang, J., Hsu, C.T. and Qiu, H. (2001). "Experiments on vertical turbulent plane jets
423 in water of finite depth." *J. Eng. Mech.* 127(1), 18-25.

424 Martins, R.B.F. (1975). "Scouring of rocky riverbeds by free-jet spillways." *Water*
425 *Power Dam Constr.*, 27(4), 152-153.

426 Mason, P.J. (1984) "Erosion of plunge pools downstream of dams due to the action of
427 free-trajectory jets." *Proc., Inst. Civ. Eng.*, Part 1, 76, 523-537.

428 Mason, P.J. and Arumugam, K. (1985) "Free jet scour below dams and flip buckets."
429 *J. Hydraul. Eng.*, 111, 220-235.

430 Mason,P.J., (1989a) "Effects of air entrainment on plunge pool scour." *J. Hydraul.*
431 *Eng.*, 115, 385-399.

432 Mason,P.J., (1989b) "Discussion of 'Effects of air entrainment on plunge pool
433 scour.'" *J. Hydraul. Eng.*, 115, 385-399.

434 McKeogh, E.J. and Elsayy, E.M. 1980. "Air entrained in pool by plunging water jet."
435 *J. Hydr. Div., ASCE*, 106: 1577-1593.

436 Miozzi, M., Lalli, F. and Romano, G.P. (2010). "Experimental investigation of a free-
437 surface turbulent jet with Coanda effect." *Exp. Fluids*, 49, 341-353.

438 Pagliara, S., Hager, W.H., and Minor, H.E. (2006). “Hydraulics of plane plunge pool
439 scour.” *J. Hydraul. Eng.*, 132(5), 450-461.

440 Pagliara, S., Massimiliano, A. and Hager, W.H. (2008). “Hydraulics of 3D plunge
441 pool scour” *J. Hydraul. Eng.*, 134(9), 1275-1284.

442 Puertas, J. and Dolz, J. (2005). “Plunge pool pressure due to a falling rectangular jet.”
443 *J. Hydraul. Eng.*, 131(5), 404-407.

444 Rajaratnam, N. and Beltaos, S. (1977) “Erosion by impinging circular turbulent jets.”
445 *J. Hydr. Div.*,103(10), 1191-1205.

446 Rodi, W. (1993). *Turbulence models and their application in hydraulics: a state-of-*
447 *the-art review*. 3rd edition, A.A.Balkema, Rotterdam, Netherlands.

448 Salehi Neyshabouri, A.A., Ferreira Da Silva, A.M., and Barron, R.(2003). “Numerical
449 simulation of scour by a free falling jet.” *J. Hydraul. Res.*, 41, 533-539.

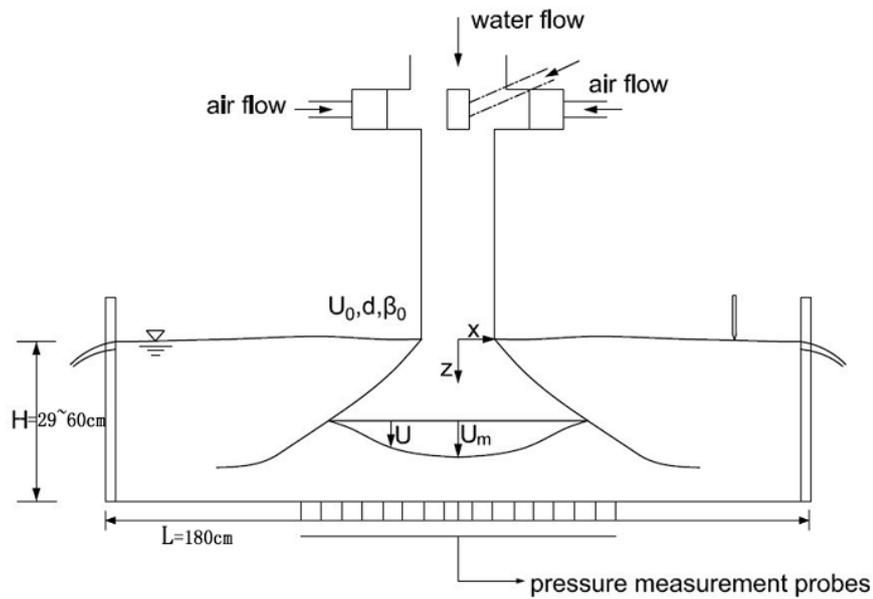
450 Vasquez, S.A., and Ivanov, V.A. (2000). “A phase coupled method for solving
451 multiphase problems on unstructured meshes.” *Proc. ASME FEDSM’00:*
452 *ASME 2000 Fluids Engineering Division Summer Meeting*, ASME, New
453 York.

454 Xu, W., Deng, J., Qu, J., Liu, S. and Wang, W. (2004). “Experimental investigation
455 on influence of aeration on plane jet scour.” *J. Hydraul. Eng.*,130(2), 160-164.

456

457

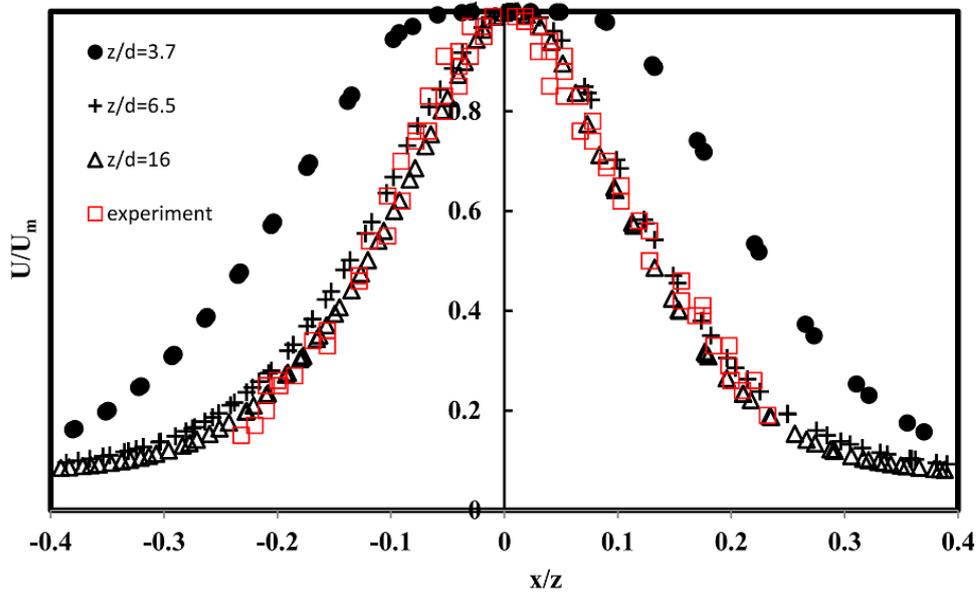
458



459

460 Figure 1. The sketch of the computational domain and experimental set-up of Guo and

461 Luo (1999).



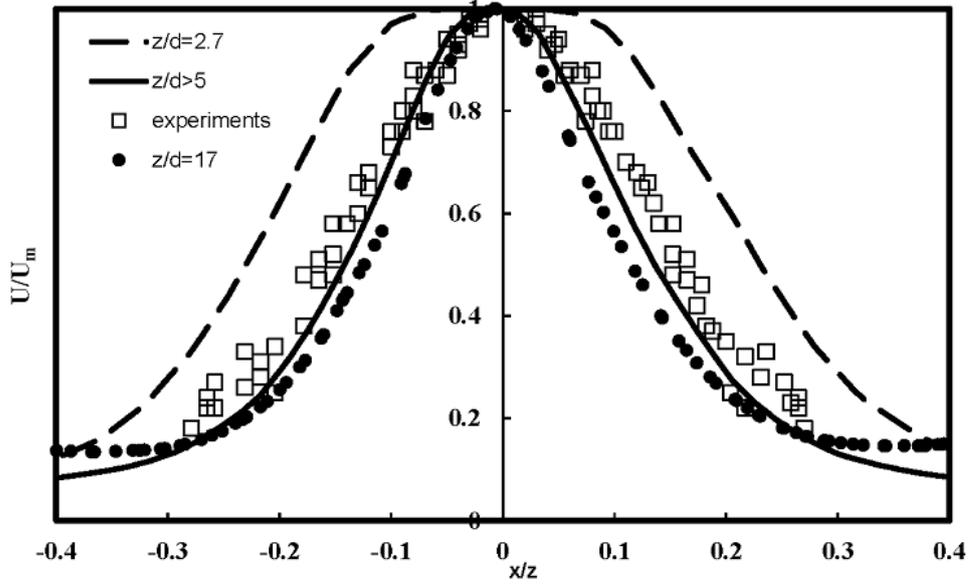
462

463 Figure 2. Comparison of simulated and measured velocity profiles for aerated jet at

464 various distances downstream from exit: $\beta_0=27\%$, $Re=58,400$, $H/d=26.7$.

465 Velocity is normalized by the local centreline velocity while horizontal

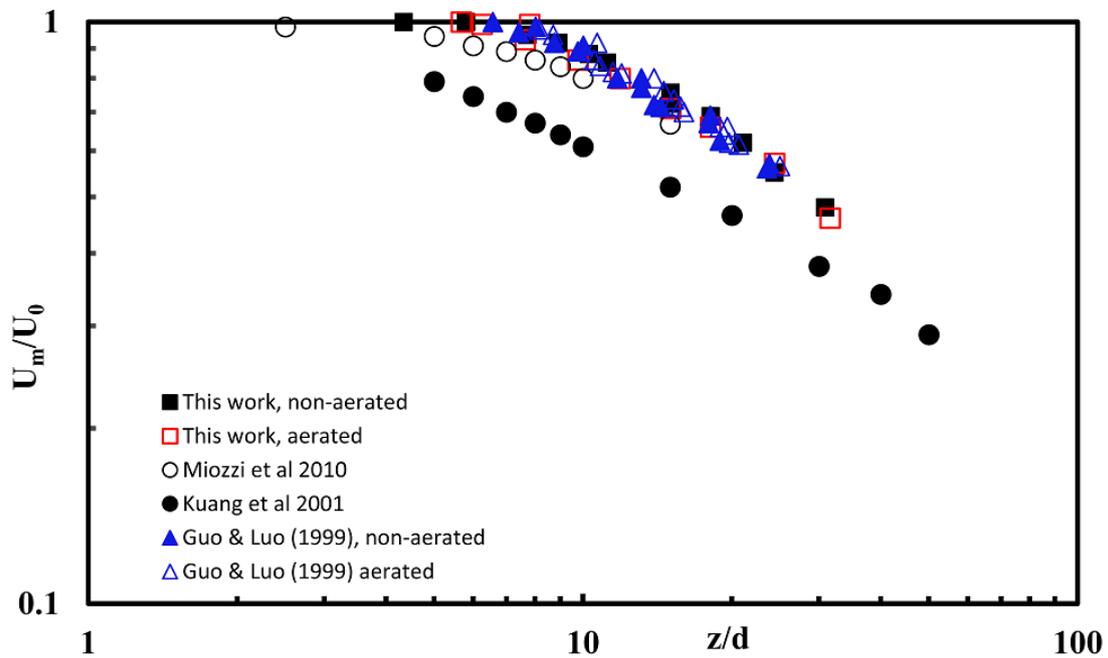
466 distance is normalized by vertical distance from exit.



467

468 Figure 3. Comparison of simulated and measured velocity profiles for nonaerated jet
 469 at various distances from exit: $Re=80,000$ and $H/d=19.5$. Velocity and
 470 horizontal distance are normalized in the same way as in Figure 2.

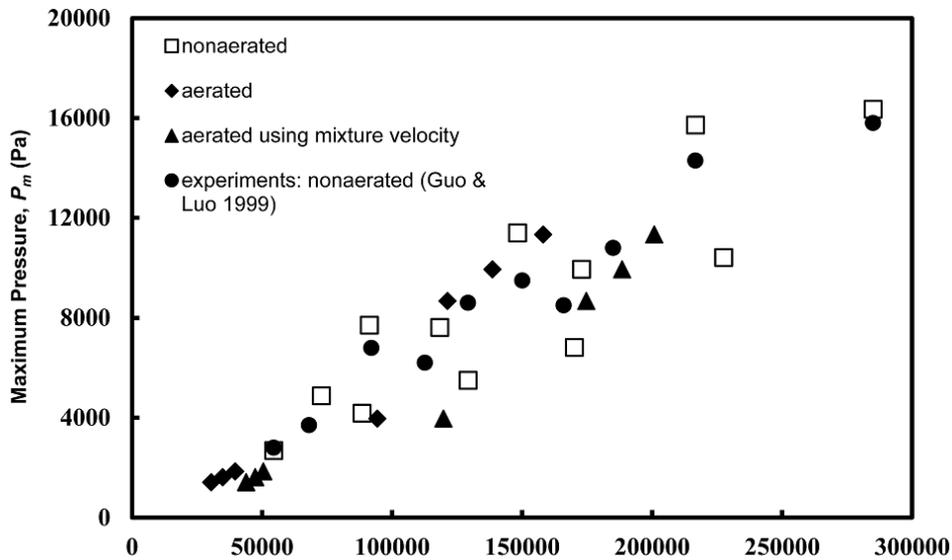
471



472

473 Figure 4. Variation of normalised jet centreline velocity decay with distance
 474 downstream from exit for various flow conditions. Experiments by Guo and

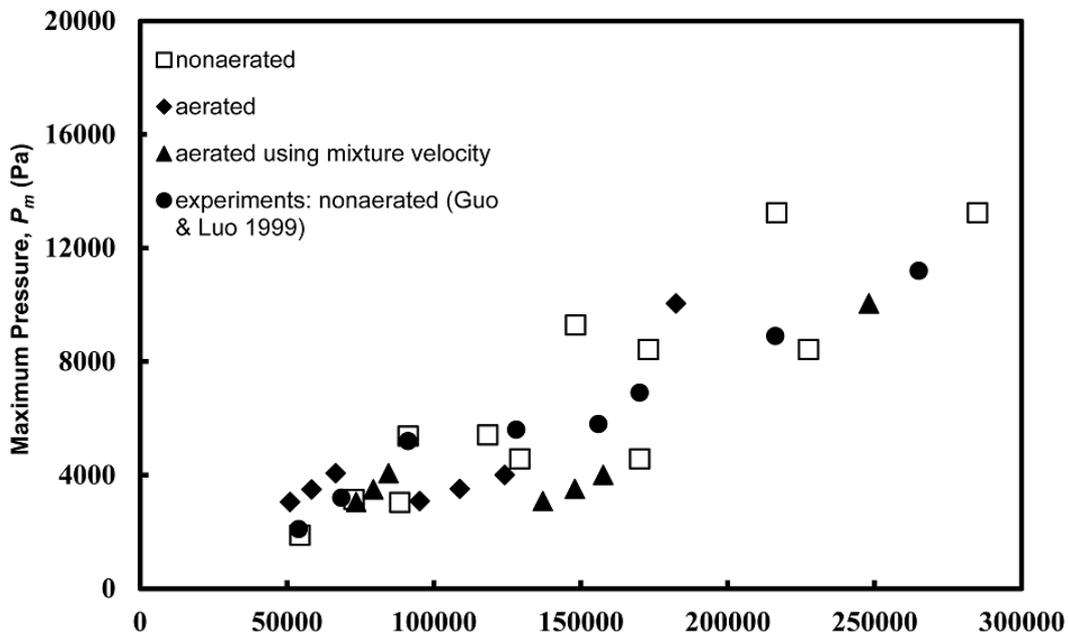
475 Luo(1999), Kuang et al. (2001) and Miozzi et al. (2010) are included for
 476 comparison.



477

478 Figure 5(a)

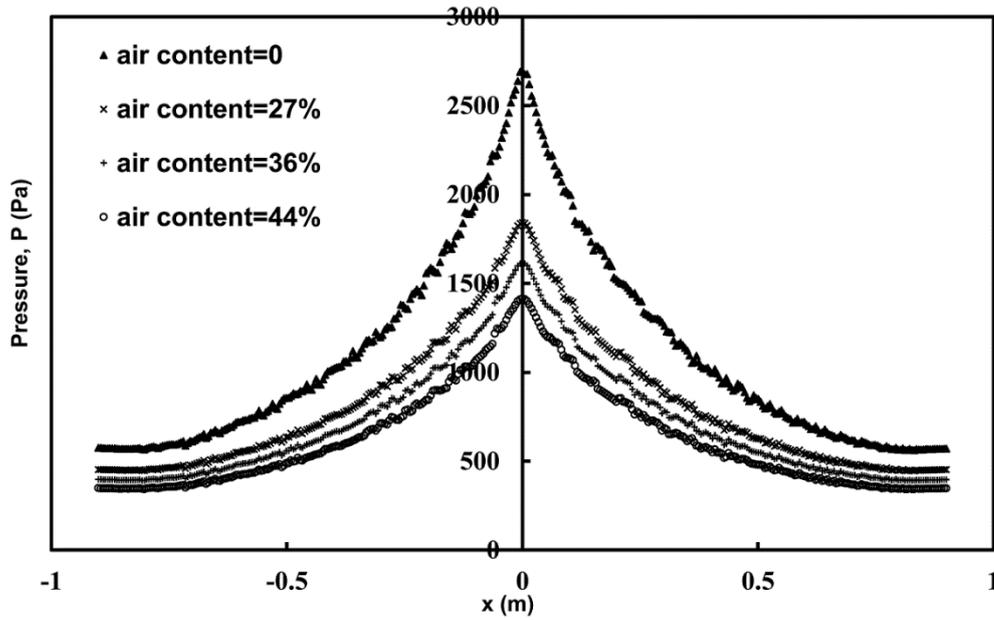
479



480

481 Figure 5(b)

482 Figure 5. The maximum pressure at the tank bottom for both aerated and nonaerated
 483 jets for various flow conditions, (a) water depth at tank H=29 cm; (b) H=39
 484 cm



485

486 Figure 6. Effect of air content on pressure distribution at tank bottom for $d=1.6$ cm,

487 $U_0 = 3.4$ m/s, $H=29$ cm and $\beta=0, 27\%, 36\%$ and 44% , respectively.

488