Numerical simulation of jet flows in homogeneous and heterogeneous media

A K Shevchenko¹, S V Strijhak² and S N Yakovenko¹

¹Khristianovich Institute of Theoretical and Applied Mechanics, Siberian Branch of Russian Academy of Sciences, Novosibirsk 630090, Russia
²Ivannikov Institute for System Programming, Russian Academy of Sciences, Moscow 109004, Russia

a.shevchenko@live.com, strijhak@yandex.ru, s.yakovenko@mail.ru

Abstract. Submerged jets at different Reynolds numbers (Re) with random perturbations added to the inlet velocity profile are simulated numerically by solving the Navier–Stokes equations. The effects of Re on flow behavior and the length of laminar jet region are studied. A natural gas jet flow emitting form the sea bottom is also simulated using the URANS and VOF methods, and forces acting on an object placed within the multiphase jet are evaluated.

1. Introduction

Study of microjets is relevant [1-5] for energy issues, such as the flow control from various burner nozzles, for transport applications, where jets might be used to reduce engine noise, aircraft drag, fuel consumption. Also, multiphase jets appear [6-10] when underwater oil and gas pipelines leak or break.

The present simulations use the OpenFOAM software with the icoFoam solver for incompressible plane and rectangular jets, and the interFoam solver for a gaseous jet from the water reservoir bottom.

2. Set up of the 2D computations

To simulate a homogenous submerged jet, the unsteady continuity and Navier–Stokes equations are solved in Cartesian coordinates \((x, y)\) at \(Re = u_0 h/v = 80, 160, 320\) where \(u_0\) is the mean inlet velocity, \(h\) is the jet nozzle height, \(v\) is the molecular viscosity. Cases of the top-hat, \(u(y) = u_0\), and parabolic, \(u(y) = 1.5 u_0 (1 - (2y/h)^2)\), inlet velocity profiles are considered. For both cases, a random noise with amplitude \(A_{\text{noise}} = 0.001\) is added to the flow by means of the inlet profile distortion. Figure 1a shows the computational domain. Boundary conditions have been given in [1] whereas the initial conditions are: \(u(x, y, t = 0) = v(x, y, t = 0) = 0\), \(p(x, y, t = 0) = 0\).

To get the solution independent of numerical factors, the effects of mesh density, time step, domain size, numerical schemes have been studied at the first stage. A way to achieve such a solution is as follows: for any selected parameter, results obtained by two runs are compared using the criterion:

\[
\left|(u^{(1)} + u^{(2)})/(u^{(2)}_{\text{max}} - u^{(2)}_{\text{min}})\right| \leq \chi,
\]

where \(u^{(1)}\) represents “less accurate” simulation and \(u^{(2)}\) is “more accurate” simulation (found, e.g. in a domain twice the size or with twice denser mesh), \(u^{(2)}_{\text{max}}\) and \(u^{(2)}_{\text{min}}\) are, respectively, the maximum and minimum of \(u^{(2)}(x, y)\), and the threshold \(\chi = 5\%\) is selected. Figure 1b where areas with \(\chi \geq 5\%\) are marked by white shows an example of such a comparison. After achieving the difference with \(\chi < 5\%\), solution is considered to be accurate enough and an examined parameter is fixed. Although this method seems to be straightforward, on practice each simulation requires it’s numerical parameters to be tuned individually.
3. Results of 2D simulations of submerged incompressible jets

The computation results illustrate three jet regions. The first part near the jet nozzle is laminar; in the second region the growing asymmetric sinusoidal instability arises, which leads to the jet breakdown into an irregular flow in the third region. These results (figure 2a) agree with experiments [2, 3].

![Figure 1](image1.png)

**Figure 1.** Computational domain scheme (a), the normalized difference example for two numerical simulations at Re = 160 where the domain size effect is studied (b).

![Figure 2](image2.png)

**Figure 2.** Instantaneous flow visualized by contours of $u(x, y)$ (a), contours of $(u(x, y))/u_{\text{max}}(x, y = 0)$ obtained after time averaging (b) in the present studies at Re = 160 for parabolic inlet velocity profile.

The length of laminar jet region is a useful characteristic, often mentioned in the literature. In the measurements [4], the jet appeared to explode at a certain “breakdown” point. In [2], the jet was considered to be turbulent when significant disturbances of the flow started to appear. In the present work, to define the transition point, the root-mean square of horizontal velocity ($U_{\text{rms}}$) is calculated along the jet centerline ($y = 0$), i.e. $(U_{\text{rms}})^2 = \langle u' \rangle^2(x, y)$, where $\langle u' \rangle = u - \langle u \rangle$, and $\langle u \rangle$ is time averaged horizontal velocity (figure 2b). Time averaging is performed using the above-mentioned normalized difference criterion with $\chi = 5\%$. As seen in figure 3a, each simulation demonstrates the same behavior: the magnitude of velocity fluctuations grows, reaching the peak value (at Re = 80, there are two peaks), then decays slightly. This is in agreement with three regions, where instabilities appear in the laminar jet part, then grow until the jet breaks down into turbulence. Thus, position $x = L$ where
\(U_{rms}\) reaches its maximum value can be considered as the point of laminar-turbulent transition. Dashed curves represent jets with parabolic inlet velocity profile; solid ones are for top-hat jets. As one can see, top-hat jets exhibit a longer initial laminar region and smaller instability levels.

![Figure 3](image)

**Figure 3.** Root-mean-square velocity values along the jet centerline (a), the length of laminar jet region in [2, 4, 5] and in the present study (b).

The computed values of \(L\) are compared with the data of measurements (figure 3b). In simulations at \(Re = 160, 320\) for both parabolic and top-hat inlet velocity profiles, \(L\) is considerably smaller than in the laboratory experiments. A possible reason of this discrepancy is that the noise levels assigned in simulation \((A_{\text{noise}} = 0.001u_0)\) are higher than in experimental jets. It can also indicate to 3D effects that exist in real jets but are not covered in two-dimensional calculations. The result where top-hat inlet jets showed the longer laminar region length than parabolic ones agrees well with the data of [5] (pricked square and rhomboid markers). Figure 4a displays the curves of the inverted maximum velocity, \(u_{\max}(x = y = 0)/u_{\max}(x, y = 0)\), along \(x\). The curves show an initial region with close proximity to horizontal, then growth which for \(Re = 80\) is lengthy enough to demonstrate asymptotic approach to the power law \(u_{\max} \sim x^{-0.5}\) (that corresponds to laminar jets), then in third region of flow all curves follow \(u_{\max} \sim x^{-2/3}\) power law. Similar behavior can be seen in figure 4b, where typical thicknesses \(\delta_{0.5}\) of jets are shown. Curves of \(\delta_{0.5}\) basically demonstrate the same behavior as \(u_{\max}\) curves, and follow \(x^2\) power law in the far jet region. It is known that the turbulent jets should expand linearly, which means that the \(x^2\) law expansion is an invalid result. Thus, 3D simulations are needed to correct this.

4. Results of 3D simulations of submerged incompressible jets

At this stage, the same equations as in 2D runs discussed above are solved numerically. Two cases are computed (figure 5). One is the rectangular jet at \(Re = 1280\), where the nozzle width is equal to 20h. The second case is the plane jet, where the nozzle width is equal to the computational domain width, and conditions at front and back boundaries (planes normal to the spanwise direction \(z\)) are cyclic.

Figure 6 shows the instability development in rectangular jet through a series of the front views on spanwise cross-sections, as well as via the side view of instantaneous velocity and the top view of averaged velocity. These views exhibit primary (sinusoidal) and secondary (periodic along the span) instabilities, and a phenomenon of axes switch in rectangular jet when a jet turns around its centerline.
Figure 4. Inversed maximum horizontal velocity \( u_{\text{max}}(x = y = 0)/u_{\text{max}}(x, y = 0) \) (a), typical thickness \( \delta_{0.5} = 2|y/u_{\text{max}} = 0.5| \) (b) in comparison with power laws.

Figure 5. Computational domain scheme for rectangular jet (a) and plane jet (b).

Figure 6. Rectangular jet: contours of \( u(x, z) \) at different \( x \) (a), contours of \( u(x, y) \) at \( z = 0 \) (b), contours of averaged velocity \( \langle u(x, z) \rangle \) at \( y = 0 \) (c).
Figure 7. Flow 3D structure in the present plane-jet computations.

Figure 7 shows the plane jet structure which looks similar to that in visualizations of the wide rectangular jet in the laboratory experiments [3].

5. Simulations of multiphase jets

In the present study, computation of a gaseous jet emitting from the bottom of water reservoir of 30 m in height and 4.5 m in diameter is performed too. In the middle of domain bottom there was a nozzle of 0.3 m diameter, with the daily (24 h) discharge rate $Q = 3$ or $5 \times 10^6$ m$^3$ of methane through the nozzle. At distance 1 m above the bottom, a hollow cylinder of 3 m in height and 0.6 m in diameter (inner hole diameter is 0.3 m) is placed coaxially with the nozzle axis. The gas-liquid interface is resolved by means of the VOF model, turbulence is modeled using the URANS approach with the $k$-$\omega$ SST model. The volume fraction of gas $\alpha$ is found at times $t \leq 3.8$ s (figure 8), as well as velocity and pressure distributions. Flow velocity reaches about 70 m/s, maximum Reynolds number is $1.5 \times 10^5$.

Figure 8. Volume gas fraction $\alpha = 1$ (in red) in liquid $\alpha = 0$ (in blue) at $t = 0, 1.0, 2.0, 3.0, 3.8$ s.
Acknowledgments
The study is performed within the framework of the Program of Fundamental Scientific Research of the state academies of sciences in 2013-2020 (Project No. AAAA-A17-117030610128-8).

References
[1] Shevchenko A K and Yakovenko S N 2018 Siberian Journal of Physics 13 35
[2] Lemanov V V, Terekhov V I, Sharov K A and Shumeiko A A 2013 Tech. Phys. Lett. 39 421
[3] Kozlov V V, Grek G R, Litvinenko Yu A, Kozlov G V and Litvinenko M V 2010 Byull. Novosibirsk State Univ. Ser. Fiz. 5 28
[4] Gau C, Shen C H and Wang Z B 2009 Phys. Fluids 21 1
[5] Aniskin V M, Lemanov V V, Maslov N A, Mukhin K A, Terekhov V I and Sharov K A 2015 Tech. Phys. Lett. 41 46
[6] Milgram J H 1983 J. Fluid Mech. 133 345
[7] Cloete S, Olsen J E and Skjetne P 2009 Appl. Ocean Res. 31 220
[8] Skjetne P and Olsen J E 2012 Prog. Comput. Fluid Dyn. 12 187
[9] Fraga B, Stoesser T, Lai C C K and Socolofsky S A 2016 Ocean Modelling 97 27
[10] Yang D, Chen B, Socolofsky S A, Chamecki M and Meneveau C 2016 J. Fluid Mech. 794 798