VISJET 2.0 User Manual

1. Introduction

2. JETLAG

3. User Interface

4. Tutorial Examples

1. Introduction

1.1 VISJET-Visualization and Lagrangian Modeling for rosette plumes in an ambient current

For environmental impact assessment and outfall design studies, it is desirable to take into account the effect of an ambient current on the initial mixing of buoyant wastewater discharges. The prediction of the concentration (or dilution) of a pollutant or passive scalar along the unknown jet trajectory of a buoyant effluent discharge is a complicated fluid mechanics problem which is not fully resolved. In particular, there are very few mathematical models which can treat satisfactorily a three-dimensional jet trajectory, such as a horizontal jet into a perpendicular cross-flowing tidal current - a common outfall design configuration. For impact assessment, post-operation monitoring and risk analysis, it is necessary to have a model that is capable of giving predictions for an arbitrarily-inclined buoyant jet in a crossflow - covering the entire range of ambient current velocities and stratification conditions.

VISJET is a Windows-based flow visualization tool to portray clearly the evolution and interaction of multiple buoyant jets discharged at different angles to the ambient current. The modeling engine is a robust Lagrangian model, JETLAG, which has been tested extensively against theory, basic laboratory experimental data, field verification studies and applications. It is aimed to facilitate the environmental impact assessment and outfall design studies. It is able to
1) Predict the initial mixing of buoyant wastewater discharges in a current, and
2) Communicate the predicted impact effectively to the user or stakeholder.

1.2 Summary of graphics features

The system has the following features:

Three-dimensional graphics

3D colour graphics is used to display the spatial layouts of all jet trajectories. The user can adjust the virtual viewpoint or viewing direction. The 3D view is displayed instantly to give the user real-time visual feedback. The system supports zoom-in and zoom-out to allow the user to have a close-up look at features of small scales.

Animation

The evolution of jets and other time-varying properties, such as velocity, can be displayed with special animation effects to enhance the understanding of the data displayed. Users can have a sense on how the wastewater jets evolve.

Realism of ambience

External factors, such as direction of ambient flow currents and reference objects, are displayed to provide a proper context for the data to be visualised.

Colour coding

Colour is assigned to the jet according to the effluent concentration.

Data interrogation

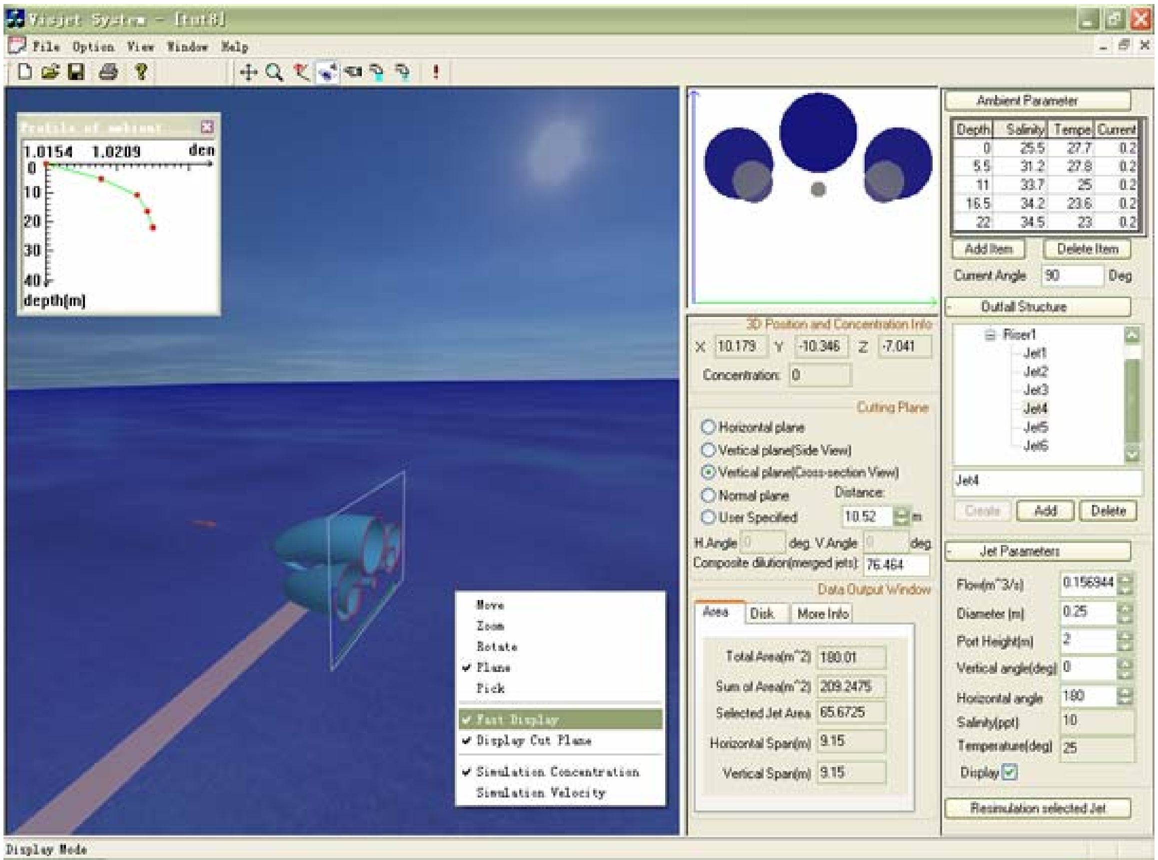
If the user wishes to know about data values defined at a point on a jet, such as velocity or concentration, it is possible to locate the point of interest with a pointing device to interactively retrieve the data required.

Jet inspection by intersection

The user can use a cutting plane at different positions to intersect the jets and observe the resulting sections. This is helpful in understanding how the jets merge and in computing composite dilution (accounting for jet merging).

1.3 VISJET Main window layout

The Main components in the VISJET window:



The right panel is for data input. The left panel is for the 3D graphic output or Outfall View. The top part of the middle panel shows the projection on the cutting plane. The lower part provides the numerical results of the simulation. The toolbox allows the user to manipulate the view of the graphic outputs and the cutting plane.

1.4 Notes on system requirements for using VISJET 2.0

1. 3D visualization is quite resource demanding, so users need to have suitable computer hardware for running VISJET 2.0 with satisfactory performance.

2. The minimum system requirements are:

   - Pentium II 400 MHz
   - 128 MB RAM
   - 100 MB free hard-disk space
   - Windows 98 SE, ME, NT 4.0, 2000 and XP
   - Screen resolution 1024 x 768 supporting 16-bit high colour
   - 3D graphics board with 4 MB memory

3 To ensure a reasonable performance, some of the special virtual reality features, such as the disturbance of the water surface and the more realistic atmosphere representation, are turned off by default if your system only satisfies the minimum requirements.

4. To run VISJET 2.0 with all the special visualization features switched on, it is recommended that your system should have:

   - Pentium III 1 GHz
   - 256 MB RAM
   - 3D graphics board with 16 MB memory

5. Running VISJET 2.0 will take up at least 20 MB memory. More memory will be used up when greater number of jets is modeled (for 20 jets, 39 MB memory will be occupied). Also, displaying the cut plane will require much more memory during the creation process (for 20 jets, 80 MB memory is needed at the peak). Therefore, when working with a large number of jets, users should make sure that sufficient memory is available. In most cases, the user will probably be working with less than 20 jets.

6. The VISJET file with an extension *vj* has a minimum size of about 100 KB. With 20 jets, the file size will increase up to about 3 MB.

7. For Windows 98 users, they should set their colours to 16-bit high colour. Using 24-bit true colour may cause VISJET failure to display the 3D graphics window.

2. JETLAG

2.1 JETLAG - Introduction

JETLAG is a robust **LAG**rangian **JET** model that handles an arbitrarily inclined round buoyant jet in a current, with a three-dimensional trajectory. It uses a Lagrangian Projected Area Entrainment (PAE) concept which assumes that the “forced entrainment” (the vortex entrainment in the bent-over jet/plume) is equal to the ambient flow intercepted by the ‘windward’ face of the plume element. The model has a rigorous theoretical basis, and its connection with Eulerian models can be established; it is consistent with the concept of asymptotic flow regimes (e.g. the advected puff and thermal in the bent-over phase).

JETLAG is unique in that the Lagrangian model does not, strictly speaking, solve the usual Eulerian governing differential equations of fluid motion and mass transport. Instead, the model simulates the key physical processes expressed by the governing equations. The unknown jet trajectory is viewed as a series of non-interfering “plume-elements” which increase in mass due to shear-induced entrainment and vortex-entrainment (forced entrainment) due to the crossflow - while rising by buoyant acceleration. The model tracks the evolution of the average properties of a plume element at each step by conservation of horizontal and vertical momentum, conservation of mass accounting for entrainment, and conservation of mass/heat - all in a fixed reference frame. The vortex entrainment is accurately determined, while pressure drag is ignored. The approach can also be shown to be equivalent to but more robust than the alternative of formulating and solving the Eulerian governing equations in natural co-ordinates.

The model predictions have compared well with basic laboratory experimental data, and it displays the correct asymptotic behaviour. JETLAG reproduces the correct behaviour of i) a round buoyant jet in stagnant or near stagnant fluid, and ii) a line puff/impulse advecting at the ambient velocity, in the bent-over phase, for momentum/buoyancy dominated jets. The current version of the model has been validated against experimental data by different investigators for: straight jets and plumes, vertical buoyant jet and dense plume in crossflow, oblique momentum jet in crossflow, horizontal buoyant jet in coflow; horizontal buoyant jet in crossflow; vertical buoyant jet in stratified crossflow; coflow and counterflowing momentum jets; buoyant plumes in weak current. The detailed derivation of the model can be found in Lee and Cheung (1990); related studies and verification can be found in the references provided at the end of the user guide.

2.2 The origin of JETLAG

JETLAG has its roots in the model UOUTPLM initially developed by the United States Environmental Agency (Frick 1984; Muenlenhoff *et. al* 1985). It uses a Lagrangian Projected Area Entrainment (PAE) concept to treat the buoyant jet in crossflow problem.

The model UOUTPLM has several important limitations: a) it can handle only jets with two-dimensional trajectories (e.g. vertical jet in crossflow); b) the shear entrainment hypothesis is incorrect; c) by virtue of the control volume formulation and the implementation of the entrainment computation, the scheme can be unstable in regions of strong plume curvature; d) its connection with basic jet theory has not been established, and e) the interpretation of the model predictions in terms of the contaminant concentration field in the bent-over jet is not clear.

Hence a great number of practical outfall discharge problems cannot be handled by UOUTPLM; for example, a horizontal buoyant jet in a crossflow, dense plumes, oblique jets, and jet in coflow or counterflow. Many outfalls located in shallow coastal waters around the world (e.g. in SE Asia and in the UK) fall into this category. The prediction of mixing for these outfalls cannot be satisfactorily handled by the USEPA models. Even within the category of jets with two-dimensional trajectories, the stated range of applicability for UOUTPLM is only VANG= -5° to 90°, where VANG is the initial discharge angle from the horizontal. These significant limitations were removed in the newly developed and considerably more powerful JETLAG (Lee and Cheung 1990; Cheung 1991; Cheung and Lee 1996, 1999) which handles an arbitrarily-inclined round buoyant jet in a current, with a three-dimensional trajectory. Since its inception around 1989 (Lee and Cheung 1990), JETLAG has proved to be a robust model that has been applied and verified in many situations (e.g. Cathers and Pierson 1991; Gordon and Fagan 1991; Horton et al 1997; see next section). The model also includes a general formulation for jet mixing in a weak crossflow (the near-far field transition), and has been validated against all available basic laboratory data, including jets issuing into a counterflow (Chan and Lam 1998; Lam and Chan 1997).

Major advances in our understanding of jet/plume in crossflow have been made possible in recent years with the development of non-intrusive laser-induced fluorescence (LIF) and digital image processing techniques, and 2D/3D turbulence models (Chu et al 1996; Chen and Lee 2000; Lee, Kuang and Chen 2002; Lee, Chen and Kuang 2002). These findings have been incorporated in the current version of JETLAG. In particular, a novel treatment of the transition from the jet/plume-dominated to the ambient current-dominated regime is included. The coflowing jet situation has also been entirely re-modeled (Lee et al. 2000). These studies have also greatly facilitated the interpretation of the Lagrangian model predictions in relation to the complex, bifurcated scalar field in the bent-over phase of the buoyant jet.

2.3 List of JETLAG/VISJET Users and Prototype Outfall Applications

The Lagrangian model JETLAG/VISJET has been extensively validated against laboratory experimental data of buoyant jets in a crossflow by many investigators. It has also been verified in field experiments, and applied to a number of actual outfall studies. The users and applications include:

Sydney Deepwater Ocean Outfall post-operation environmental monitoring study
Zarautz Marine Outfall, Spain
Shek O Outfall beach pollution study
Urmston Road Sewage Outfall (Northwest New Territories, Hong Kong) post-operation monitoring study
Hong Kong Strategic Sewage Disposal Scheme (SSDS) Environmental Impact Assessment Project
Shanghai Stage 2 Sewage Outfall
North Point Outfall
Hastings Outfall, Sidmouth outfall, Gosport outfall, Jaywick outfall (United Kingdom)
Stonecutters Island Interim Outfall, Harbour Area Treatment Scheme (HATS)
Bhabha Atomic Research Centre (Isotope Division), Trombay, India

List of Users

Academic Institutions

Alaska Department of Environmental Conservation
British Columbia Government
California State Water Resources Control Board
City University of Hong Kong
Clarkson University
Duke University
Fisheries and Oceans Canada
Goddard Earth Sciences Technology Center/NASA Goddard Space Flight Center
Hong Kong University of Science and Technology
Indian Institute of Technology, Bombay
Iran University of Science and Technology
Korea Ocean Research and Development Institute
Loughborough University
Massachusetts Institute of Technology
McGill University
Monterey Institute of International Studies
National Center for High-Performance Computing, Taiwan
National Institute of Water & Atmospheric, New Zealand
National Taiwan University
National Water Research Institute, Canada
NATO Saclant Undersea Research Centre
OGI School of Science & Engineering
Rutgers University
Seoul National University
South China Sea Institute, China
The Department of Environmental Protection, Australia
The Hong Kong Polytechnic University
The University of Hong Kong
The University of Queensland
Tianjin University
Tongji University
Tsinghua University
Universidade Do Porto
University of Alberta
University of Arizona
University of Canterbury
University of Hawaii
University of Karlruhe
University of Michigan
University of Southern California
University of Toronto
University of Wisconsin-Madison
Univ. Metropolitana, Venezuela

Consulting Engineering Firms

Applied Science Associates, USA
Atkins China Ltd, HK
Battelle, USA
CDM International Inc.
CH2M HILL
Joiner Engineering
Montgomery Watsons HK Ltd.
Mott Connell Ltd., HK
Mouchel Asia Ltd, HK
Patterson Britton & Partners, Australia
Tetra Tech, Inc

Other Individual Users From

Australia
Brazil
Canada
Germany
Greece
India
Iran
Italy
Korea
New Zealand
Portugal
P. R. China
Taiwan, China
Saudi Arabia
United Kingdom
USA
Venezuela

2.4 Output file - SUSPEND

SUSPEND is a standard output file. Discharge parameters and characteristic length scales are also printed in addition to the numerical results. The (x, y, z) co-ordinates of the computed jet trajectory are printed along with the jet half-width, average dilution and velocity, and average density deficit.

2.5 References

Cathers, B. and Peirson, W.L. (1991). “Verification of plume models applied to deepwater outfalls”, in *Proceedings of the International Symposium on Environmental Hydraulics*, Hong Kong, Lee, J.H.W. and Cheung, Y.K. (ed.), December 1991, Vol.1, Balkema, pp.261-266.

Chan, C.H.C. and Lam, K.M. (1998), “Centreline velocity decay of a circular jet in a counterflowing stream”, *Physics of Fluids*, Vol. 10 (3), pp. 637-644.

Chen, G.Q. and Lee, J.H.W. (2000). “Numerical experiment of two-dimensional line thermal”, *Journal of Hydrodynamics*, Ser.A, Vol.15, No.4, pp.411-423.

Cheung, V. (1991) Mixing of a round buoyant jet in a current. *Ph.D. thesis, Dept. of Civil & Structural Engineering*, University of Hong Kong, Hong Kong.

Cheung, V. and Lee, J.H.W. (1996) Discussion of “Improved prediction of bending plumes”. *Journal of Hydraulic Research*, Vol.34, No.2, pp.260-262.

Cheung, V. and Lee, J.H.W. (1999) Discussion of “Simulation of oil spills from underwater accidents I: model development”, *Journal of Hydraulic Research*, Vol.37, pp.425-429.

Chu, P.C.K. and Lee, J.H.W. (1996). “Mixing of a bent-over jet in crossflow”, Proc. 11th ASCE Engineering Mechanics Specialty Conference, Fort Lauderdale, Florida, May 1996, pp. 910-913.

Chu, P.C.K., Lee, J.H.W., and Chu, V.H. (1999). “Spreading of a turbulent round jet in coflow”, *Journal of Hydraulic Engineering, ASCE*, Vol.125, No.2, pp.193-204.

Chu, V.H. and Goldberg, M.B. (1974) “Buoyant forced plumes in crossflow”, *J. Hydraulics Div., ASCE*, Vol.100, HY9, pp.1203-1214.

Chu, V.H. (1977). “A line impulse model for buoyant jets in a crossflow”, in *Heat Transfer and Turbulent Buoyant Convection* (Spalding, D.B. ed.), Hemisphere, Washington D.C., Vol.1, pp.625-636.

Chu, V. H. (1994) “Lagrangian scalings of jets and plumes with dominant eddies,” in *Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes*, NATO ASI Series E: Applied Sciences, Vol. 255, (Davies, P.A. and Valente Neves, M.J., eds.), Kluwer Academic Publishers, Dordrecht, pp. 45-72.

Chu, V.H. and Lee, J.H.W. (1996). “A general integral formulation of turbulent buoyant jets in crossflow”, *Journal of Hydraulic Engineering, ASCE,* Vol. 122, No.1, pp. 27-34.

Fischer, H. B., etal (1979) *Mixing in Inland and Coastal Waters*, Academic Press, San Diego, California.

Frick, W.E. (1984) Non-empirical closure of the plume equations. *Atmospheric Environment*, Vol.18, No.4, pp. 653-662.

Gordon, A.D. and Fagan, P.W. (1991). “Ocean outfall performance monitoring”, in *Proceedings of the International Symposium on Environmental Hydraulics*, Hong Kong, Lee, J.H.W. and Cheung, Y.K. (ed.), December 1991, Vol.1, Balkema, pp.243-248.

Horton, P.R., Lee, J.H.W. and Wilson J.R. (1997). “Near-field JETLAG modelling of the Northwest New Territories Sewage Outfall, Urmston Road, Hong Kong”, *Proc. 13th Australasian Coastal and Ocean Engineering Conference (Pacific Coasts and Ports '97), Christchurch, New Zealand*, Sept. 97, Vol.2, pp. 561-566.

Lam, K.M. and Chan, H.C. (1997), “Round jet in ambient counterflowing stream”, *Journal of Hydraulic Engineering*, ASCE, Vol. 123 (10), pp.895-903.

Lee, J.H.W. and Neville-Jones, P. (1987). “Initial dilution of horizontal jet in crossflow”, *Journal of Hydraulic Engineering, ASCE*, Vol.113, HY5, pp.615-629.

Lee, J.H.W. and Neville-Jones, P. (1987). “Design of sea outfalls - Prediction of initial dilution and plume geometry”, *Proceedings of the Institution of Civil Engineers, Part 1, [Design and Construction]*, Vol.82, pp. 981-994.

Lee, J.H.W. and Cheung, V. (1990) Generalized Lagrangian model for buoyant jets in current. *Journal of Environmental Engineering, ASCE*, **116(6)**, 1085-1105.

Lee, J. H. W. and Chu, P. C. K. (1995) “Application of video image processing in the study of environmental flows”, *Proc. 10th ASCE Engineering Mechanics Conference*, University of Colorado, Boulder, Vol.2, pp. 1014-1017.

Lee, J.H.W. Rodi, W. & Wong, C.F. (1996). "Turbulent line momentum puffs." *Journal of Engineering Mechanics, ASCE*, 122, 19-29.

Lee, J.H.W. and Chu, P.C.K., ``On the added mass of a turbulent jet in crossflow'', Proc. 27th IAHR Congress, August 1997, San Francisco, Vol.1 ({\it Environmental and coastal hydraulics}), pp. 269-274.

Lee, J.H.W. and G.Q. Chen, “The jet in crossflow and the puff analogy'', Proc. 12th ASCE Engineering Mechanics Conference, La Jolla, California, May 17-20, 1998 (H.Murakami and J.E.Luco Ed.), pp. 1792-1795.

Lee, J.H.W., Li, L., and Cheung, V. (1999). “A semi-analytical self-similar solution of a bent-over jet in crossflow”, *Journal of Engineering Mechanics, ASCE*, Vol.125, pp.733-746.

Lee, J.H.W., Cheung, V., Wang, W.P., and Cheung, S.K.B., ``Lagrangian modeling and visualization of rosette outfall plumes'', Proc. Hydroinformatics 2000, Iowa, July 23-27, 2000 (CDROM)

Lee, J.H.W., Kuang, C.P., and Chen, G.Q. (2002). “The structure of a turbulent jet in crossflow - effect of jet-to-crossflow velocity”, *China Ocean Engineering*, Vol.16, No.1, pp.1-20.

Lee, J.H.W., Chen, G.Q., and Kuang, C.P. (2002). “Mixing of a turbulent jet in crossflow - the advected line puff”, in *Environmental Fluid Mechanics: theories and applications* (Ed. H. Shen et al), American Society of Civil Engineers (in press).

Muellenhoff, W.P. et al. (1985) Initial mixing characteristics of municipal ocean discharges. *Report EPA-600/3-85-073*, USEPA, Newport, Oregon.

Schatzmann, M. (1981) “Mathematical modeling of submerged discharges into coastal waters," *Proc. 19th IAHR Congress, New Delhi*, Vol. 3, pp. 239-246.

Wood, I. R. (1993) “Asymptotic solutions and behaviour of outfall plumes,” *J. Hydr. Eng., ASCE*, Vol. 119, pp. 555-580.

Wright, S.J. (1977) "Effects of ambient crossflow and density stratification on the characteristic behavior of round turbulent buoyant jets," *Report No. KH-R-36*, W.M. Keck Lab. of Hydr. and Water Resour., California Inst. of Tech., Pasadena, Calif.

3. User Interface

3.1 Input parameters

3.1.1 Ambient parameters

Specify the vertical structure of the ambient water:

Name	***Meaning***	***Acceptable range***
Depth	Depth below surface (m).	
Salinity /Density	Ambient salinity (ppt or psu) if temperature not equal to 0; Ambient density (g/ml) if temperature = 0.	Salinity: 0~100 ppt (psu) Density: 0.5~1.5g/ml
Temperature	Ambient temperature (°C).	0~100°C
Current	Horizontal current speed (m/s) (assuming in the x-direction).	

Notes:
Only stable ambient stratification is allowed, i.e. $\rho_a(d_i) \le \rho_a(d_j)$ if $d_i \le d_j$. (d=depth below free surface).

3.1.2 Outfall Parameters

Specify the properties of the outfall:

Outfall parameters with riser:

Name	***Meaning***	***Acceptable range***
Depth	Depth below surface (m).	
Salinity/Density	Effluent salinity (ppt or psu) if temperature not equal to 0; Effluent density (g/ml) if temperature = 0.	Salinity: 0~100 ppt (psu) Density: 0.5~1.5g/ml
Temperature	Effluent temperature (°C).	0~100°C
Length	Length of the outfall (m).	No less than 0.1m
Width/ diameter	Width/ diameter of the outfall (m).	

Outfall parameters without riser:

Name	***Meaning***	***Acceptable range***
Depth	Depth below surface (m).	
Temperature	Effluent temperature (°C).	0~100°C
Salinity/ Density	Effluent salinity (ppt or psu) if temperature not equal to 0; Effluent density (g/ml) if temperature = 0.	Salinity: 0~100 ppt (psu) Density: 0.5~1.5g/ml
Length	Length of the outfall (m).	No less than 0.1m
Radius	Radius of the outfall. Outfall is defined as a pipe in this mode (m).	
Space between Jets	Space between jets mounted on the outfall (m).	No less than 0.001m

Notes:

If the input variables are outside the specified range, then the upper/lower limit will be assumed.

3.1.3 Riser parameters

Specify the properties of the riser:

Name	***Meaning***	***Acceptable range***
Flow	Sum of the effluent flow of all the ports mounted on the riser (m^3/s).	No less than 1.0E-8m^3/s
Distance	Distance from the offshore end of the outfall (m).	No greater than outfall length - 2* bottom radius of riser-0.5*width (diameter) of outfall
Bottom radius	Radius at the bottom of the riser (m).	0.05 m ~ 0.5*width (diameter) of outfall
Top radius	Radius at the top of the riser (m).	0.01 m ~ bottom radius of riser
Height	The height of the riser (m).	0.1 m ~ outfall depth

Notes:

If the input variables are outside the specified range, then the upper/lower limit will be assumed.

3.1.4 Jet parameters

Specify the properties of the jet:

Name	***Meaning***	***Acceptable range***
Flow	Effluent flow from the port (m^3/s).	No less than 1.0E-8m^3/s
Diameter	Port diameter (m).	No less than 1.0E-4m
Port Height	Port height (m).	No greater than riser height
Vertical angle	Vertical jet discharge angle relative to Horizontal plane (90º for a vertical port).	
Horizontal angle	Horizontal angle of current direction with respect to jet discharge (90º for a perpendicular crossflow).	
Salinity/ Density	Effluent salinity (ppt) if temperature not equal to 0; Effluent density (g/ml) if temperature = 0. *(Read only)*	Salinity: 0~100 ppt Density: 0.5~1.5g/ml
Temperature	Effluent temperature (ºC). *(Read only)*	0~100ºC

Notes:

If the input variables are outside the specified range, then the upper/lower limit will be assumed.

3.1.5 Cutting plane parameters

A cutting plane in VISJET is defined by its normal vector. The orientation of the normal vector is defined by the vertical and horizontal angle (as in the JETLAG model); refer to ``How To'' under ``Startup Tips''. The control parameters for the cutting plane:

Vertical angle	The angle of the normal vector of the cutting plane relative to horizontal plane (90º for a horizontal plane; 0º for vertical plane)
Horizontal angle	The angle between the projection of the normal vector on the horizontal plane and the flow direction (x-axis) (90º for a vertical section - side view, and 0º for a vertical section, cross-section view)
Distance	Distance from the discharge point (No greater than 25 times of water depth).

3.2 Output parameters

3.2.1 Key parameters and length scales

The following are the key parameters and length scales for each jet:

Depth.	Port depth (m)
Diameter	Port diameter D (m)
U_j	Initial jet velocity, U_j (m/s)
U_a	Ambient current velocity, U_a (m/s)
$D\rho/\rho_a$	Dimensionless initial density difference, $(\rho_a(0)-\rho_j)/\rho_a(0)$
ρ_j	Initial effluent density (g/ml)
ρ_a	Ambient density at source level, $\rho_a(0)$ (g/ml)
Ver. Ang	Vertical discharge angle, Φ_j (degree)
Hor. Ang	Horizontal discharge angle; θ_j (degree)
Fd	Jet Densimetric Froude no., $F = Vj/(gD(\rho_a(0)-\rho_j)/\rho_a(0))^{1/2}$
Q_j	Volume flux / discharge flow, $Q = Vj\ (\pi D^2/4)$ (m^3/s)
M_j	Momentum flux, $M = Vj^2(\pi D^2/4)$ (m^4/s^2)
B_j	Buoyancy flux = $Qg(\rho_a(0)-\rho_j)/\rho_a(0)$ (m^4/s^3)
l_Q	Discharge (volume, geometric) length scale, $l_Q = Q/M^{1/2}$ (m)
l_m	Cross momentum length scale, $l_m = M^{1/2}/U_a$ (m)
l_b	Cross buoyancy length scale, $l_b = B/U_a^3$ (m)
l_M	Jet/plume length scale, $l_M = M^{3/4}/B^{1/2}$ (m)
S_m	Characteristic dilution for momentum dominated far field (mdff), $S_m = U_a\, l_m^2/Q$
S_b	Characteristic dilution for buoyancy dominated far field (bdff) , $S_b = U_a\, l_b^2/Q$

3.2.2 Disk information

The following information about the computed disk will be displayed:

Disk no.	The sequence no. of the selected computed disks for the selected jet
Centre position	The (x, y, z) co-ordinates of the center of the selected disk, which is the computed jet trajectory
Radius	Jet half width of the selected disk
Thickness	Thickness of the selected disk
Angle	Vertical angle is the angle between the jet axis and the horizontal plane; horizontal angle is the angle between the x-axis and the projection of the jet axis on the horizontal plane
Velocity	Jet velocity of the selected disk
Concentration	Maximum/average concentration of the selected disk

3.2.3 Cross section concentration

The following information related to the cross section concentration will be displayed:

Position	The (x, y, z) co-ordinates at the position of the point selected by the mouse or pointing device
Concentration	Average concentration at the above position

3.2.4 Cross section area information

The following information related to the projected area of the jets will be displayed:

Total area	The total projected area of the jets on the cutting plane; the overlapped areas are not double counted
Sum of areas	The sum of all the projected areas of the individual jets
Selected jet area	The projected area of the selected jet
Horizontal span	The horizontal span of the projected region for the selected jet (defined by bounding box)
Vertical span	The vertical span of the projected region for the selected jet

4. Tutorial Examples

4.1 Example 1

Vertical buoyant jet in stagnant fluid

4.2 Laboratory Example

Wah Fu Outfall Discharge

4.3 Example 2

Horizontal buoyant jet in stagnant stratified fluid

4.4 Example 3

Multiple buoyant jets in stagnant fluid

4.5 Example 4

Vertical buoyant jet in stratified crossflow

4.6 Example 5

Vertical dense jet in uniform crossflow

4.7 Example 6

Horizontal buoyant jet in uniform crossflow - Zarautz Marine Outfall, Spain

4.8 Example 7

Horizontal buoyant jet in stratified crossflow - Zarautz Marine Outfall, Spain

4.9 Example 8

Buoyant jets from a rosette-shaped ocean outfall riser in natural flow –Hong Kong Strategic Sewage Disposal Scheme (SSDS)

4.1 Example 1 Vertical buoyant jet in stagnant fluid

The file is **tut1.vj**.

VISJET simulates the mixing of single or multiple buoyant jets discharged from one or more risers mounted on an ocean outfall. In a particular application, the input parameters for the ambient condition, the outfall, riser, and jet characteristics are needed. For example, a single buoyant jet can be simulated by specifying a single jet on a single riser. Multiple jets can be simulated by specifying a single jet on each of a number of risers. Rosette jet groups on multiple risers can be simulated by specifying the multiple jet characteristics on each of the risers. We start with several examples on how to use VISJET for simulating a single buoyant plume, which is of interest in many applications, followed by more complicated situations.

The first example is for a single vertical buoyant jet discharge into an otherwise stagnant fluid. **For a single jet the riser flow is the same as the jet flow**. The main parameters are as follows:

Ambient Parameters:

Depth	20 m
Density	1.0256 g/ml
Current velocity	0.0 m/s (Current angle 90º)

Outfall Parameters:

Depth	22 m
Temperature	0ºC
Density	1.0 g/ml

(The density of the effluent is entered as input. There are two possible formats:
- i) if density is input directly, a zero value, 0.0, must be entered for temperature;
- ii) alternatively, both the temperature and salinity of the effluent can be entered as input and the effluent density will then be computed by the model.

Riser Parameters:

Distance	0 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

(For single jet the riser distance is immaterial; set to zero)

Jet Parameters:

Flow	0.03 m³/s
Diameter	0.1 m
Port height	2 m
Vertical angle	90º
Horizontal angle	0º

General Notes:

1. Study how to input parameters.
   - i) Click **New** in the startup tips window.
   - ii) Click **Add a level** twice, input ambient parameters as shown above.

iii) Click **Next**, select **Create a scenario with riser**. Then click **Create an outfall**, and click **Outfall_1**, and input the above outfall parameters.
iv) Click **Riser1**, input the above riser parameters.
v) Click **Jet1**, input the above jet parameters.
vi) Click **Finish**

If you have problems with the input parameters, you can open the file **tut1.vj** in the folder *tutorial files* to examine the correct input parameters (Click Outfall_1, Riser_1, and Jet_1 to observe the input parameters).

2. Use the animation function to see the evolution and spread of the jet. Click toolbar to see the rise and growth of the jet from the discharge port to water surface. In the Lagrangian model, the jet path is made up of a series of plume elements (`disks') which vary in position, width, and velocity as they mix with the surrounding fluid.

3. Study how to get the jet characteristics of each Lagrangian element: information about the computed average velocity, maximum concentration, and average dilution at that height.
   i) Select toolbar or press the right mouse button and select **Pick**.
   ii) Put cursor at any point on the jet you want to get the information in the *data output window.*
   iii) Move *scroll bar* up or down at the right of the data output window to choose the disk number. For example, the center of Disk 46# is located at (0, 0, 0.11), visual radius = 0.048 m, thickness = 0.037 m, vertical angle = 90º, horizontal angle = 0º, average velocity = 2.8 m/s, maximum concentration = 1.0, average dilution = 1.4 and average concentration = 0.7332.

   The dilution is a measure of the degree of mixing achieved by the jet; the inverse of dilution is the relative concentration of any pollutant contained in the discharge.

4. Save file: you can select **Save as** in main menu **File** to save the file (file format *****.vj**) for later reuse or modification.

5. Compute a **horizontal buoyant jet** in stagnant fluid based on this case. You just need to change the vertical angle 90º to 0º in the input jet parameters, resimulate the model (Click the toolbar then save this example. If you have problems, you can open the file **tut1a.vj** in the folder *tutorial files*). Notice the dilution is increased for this jet as the length of the jet path is greater than that of the vertical jet at the same vertical position. For example, Disk 296#, the disk center is located at (2.03, 0, 0.11), visual radius = 0.27 m, thickness = 0.0066 m, vertical angle = 8.4º, horizontal angle = 0, average velocity = 0.50 m/s, maximum concentration = 0.226 and average dilution = 7.5.

6. **You can also predict the jet from an actual outfall and compare it with observations in a laboratory experiment!**

7. You can also try the "**Create a scenario without riser**" at 1.iii)

8. Close this file to start next tutorial.

4.2 Laboratory Example Wah Fu Outfall Discharge

The file is **WahFu.vj**.

Consider the Wah Fu Outfall which discharges domestic wastewater from a housing estate in the form of a number of submerged buoyant jets, at a depth of about 7-12 m. The jets are sufficiently spaced apart so that they may be considered independent of each other. In this development, salt water is used for flushing, so that the jet discharge is brackish water (i.e. a mixture of sea water and freshwater). Compute the mixing for this discharge which is inclined at 20 degrees to the horizontal, and compare your computed results with the corresponding observed jet in a laboratory experiment. The main parameters are as follows:

Ambient parameters:

Depth below surface (m)	Density (Sigma-t)	or	Density (g/ml)
0	18.0		1.018
7	18.0		1.018

Current velocity 0.0 m/s (Current Angle=90º)

(N.B. density (g/ml) = 1. + 0.001 Sigma-t)

Outfall parameters:

Depth	9.0m
Density	1.004 g/ml (Sigma-t = 4 unit)

Riser Parameters:

Distance	0m
T. Radius	0.05m
B. Radius	0.05m
Height	2 m

Jet Parameters:

Flow	0.00626 m³/s
Diameter	0.1 m
Port height	2 m
Vertical angle	20º
Horizontal angle	0º

General Notes:

1. Animate the jet evolution and compare the computed results with the corresponding observed jet in a laboratory experiment. Note the irregular edge of the real turbulent jet; the model computes only the average turbulent-mean properties.

2. Click **View suspend file** (**More info** in data output window) to see the printout of the computed results in a **SUSPEND** file.

3. The **SUSPEND** file shows key input parameters and length scales that govern the mechanics of buoyant jet mixing. For example, Total Q = 0.00626 (m³/s), jet velocity = 0.8 m/s, Jet densimetric Froude number F_d =6.86.

4. The following is the observed dyed jet of the Wah Fu outfall discharge; in this experiment the jet is a 1:11 scale model of the actual outfall, discharged at the same jet densimetric Froude number of 6.8.

4.3 Example 2 Horizontal buoyant jet in stagnant stratified fluid

The file is **tut2.vj**.

The ambient receiving water often has a vertical variation of salinity and/or temperature, leading to density stratification. The jet may cause so much mixing that the mixed effluent stays trapped below the free surface. In this example we compute the mixing for a horizontal buoyant jet in a stratified fluid. The main parameters are as follows:

Ambient Parameters:

Depth below surface (m)	Density (Sigma-t)	Density (g/ml)
0	8.0	1.008
1	8.4	1.0084
4	11.0	1.011
7	12.2	1.0122
10	13.2	1.0132
13	15.3	1.0153
16	16.5	1.0165
19	16.6	1.0166
20	16.6	1.0166

(N.B. Density (g/ml) = 1 + 0.001 Sigma-t)

Current velocity 0.0 m/s (Current Angle =90º)

Outfall Parameters:

Depth	22 m
Temperature	0ºC
Density	0.999 g/ml (Sigma-t = -1 unit)

Riser Parameters:

Distance	0 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

Jet Parameters:

Flow	0.0147 m^3/s
Diameter	0.125 m
Port height	2 m
Vertical angle	0º
Horizontal angle	0º

General Notes:

1. Click **Open** in the startup tips window, then select and open file **tut2.vj** in the folder *tutorial files*

2. The density variation of the ambient fluid can be specified in either of the following two ways: i) the salinity and temperature at each depth are entered, from which the density will be computed by the model by the equation of state; ii) the density is given directly in units of ρ (g/ml) – for this case the density is entered under the Salinity column in ambient parameter

window, and a zero (0.0) value must be entered for temperature. For this case the natural density stratification is represented by values given at 8 depths.

3. Animate the jet evolution and observe how the jet is trapped beneath the water surface. Click **View suspend file** (**More info** in data output window) to see the printout of the computed results in a **SUSPEND** file.

4. The **SUSPEND** file shows key input parameters and length scales that govern the mechanics of buoyant jet mixing. For example, Total Q = 0.0147 (m^3/s), Densimetric Froude number F_d =8.22, Buoyancy flux B_j = 0.0025 m^4/s^2, jet momentum length scale l_M = 0.97, and so on. The computed jet characteristics (co-ordinates of the jet trajectory, plume visual radius, velocity, concentration, dilution etc.) are tabulated. You will find some information about trap level. For this case, the **neutral buoyancy level = 5.1 m**, with a corresponding **average dilution = 38.1** and visual radius = 1.14 m; the buoyant jet center **maximum rise height = 7.1 m**, corresponding average dilution = 41.1 and visual radius = 2.58 m.

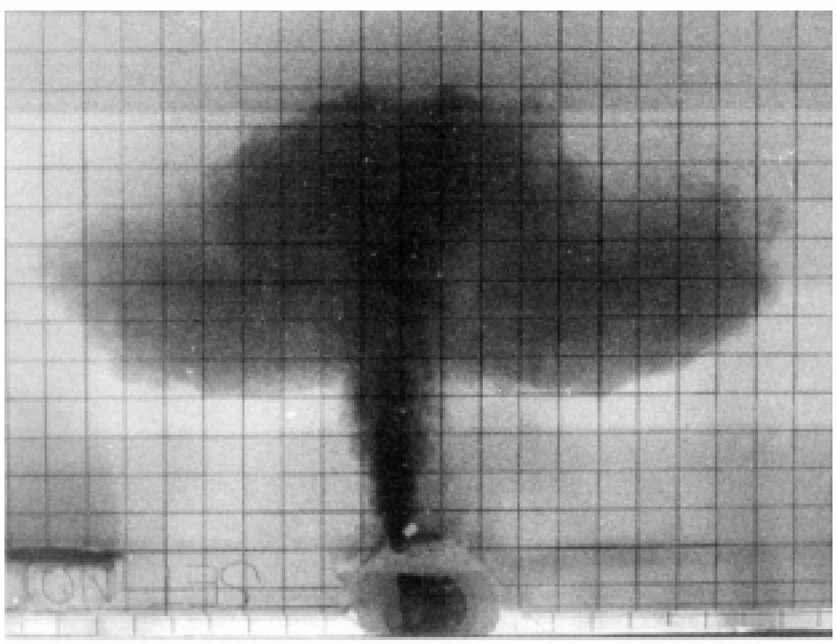
5. In the summer wet season, the receiving water is often stratified, and the sewage field may not reach the surface; the submergence of the sewage may be desirable for protection of nearby beaches.

   Notice that the computation can continue after the first trap level; the computation will be stopped after the first oscillation in VISJET, at trapped level = 5.41 m, corresponding average dilution = 53.9 and visual radius = 1.61 m.

6. Close this file to start next tutorial, or change input parameters to make your own run.

The following shows two examples of real life examples of a plume or buoyant jet in stagnant fluid: i) the trapped smoke plume from the Lamma Island power station in the early morning, when there was a temperature inversion. ii) laboratory experiments of a plane (two-dimensional) vertical buoyant jet in linearly stratified fluid.

Smoke plume from Lamma Island Power Station trapped in atmospheric inversion layer



Vertical plane buoyant jet in linearly stratified fluid

4.4 Example 3 Multiple buoyant jets in stagnant fluid

The file is **tut3.vj**.

Consider the horizontal buoyant jet example in Tutorial 1 again, but this time instead of using one jet, divide the flow of 0.03 m^3/s into 4 jets, each discharged from a different riser. So now we have FOUR horizontal buoyant jets in a uniform stagnant fluid, but each port discharges ¼ of the original flow. The main parameters are as follows:

Ambient Parameters:

Depth	20 m
Density	1.0256 g/ml
Current velocity	0.0 m/s (Current Angle=90º)

Outfall Parameters:

Depth	22 m
Temperature	0ºC
Density	1.0 g/ml

Riser Parameters:

Distance	0 m (riser1)
	20 m (riser2)
	40 m (riser3)
	60 m (riser4)
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

(The distance from the most offshore end riser is indicated above; the spacing between two adjacent risers is 20 m).

Jet Parameters:

Flow	0.0075 m^3/s
Diameter	0.1 m
Port height	2 m
Vertical angle	0º
Horizontal angle	0º

General Notes:

1. You can also change the orientation of the diffuser axis with respect to the current by changing the default current angle 90^o in the ambient parameter window.

2. Click **Open** in the startup tips window, then select and open file **tut3a.vj** in the folder *tutorial files*

3. Do the following to add three new risers with one jet on each riser:
   i) Highlight Riser1, click Add, riser2 and the jet from this riser will be created.
   ii) Highlight the new riser and its jet1, input the corresponding parameters, then specification of the riser and the associated jet will be completed.

iii) Repeat the procedure to create other two risers and their associated jets. Resimulate the model (Click the toolbar, then save this file). Use the animation function (Click the toolbar) to see the evolution and spread of the jets.

If you have any problems, you can consult **tut3.vj** to look up the correct input parameters.

4. Open the SUSPEND file (Click **View suspend file** in menu **More info** in Data output window) to see the results and determine the dilution for this case of multiple jets. You will find that each jet achieves an **average dilution** at water surface = **397**; the water quality is significantly improved by this design compared with the case in tutorial 1 (average dilution = 213).

5. When will the plumes from adjacent risers merge? Decrease the distance between risers to find the pattern of plume interaction.

6. Close this file to start the next tutorial or change parameter values to create your own run.

4.5 Example 4 Vertical buoyant jet in stratified crossflow

The file is **tut4.vj**.

Consider a single vertical discharge into a horizontal crossflow of u_a=0.1 m/s, at a depth of 14 m below the free surface. The receiving water is linearly stratified. The main parameters are as follows:

Ambient Parameters:

Depth below surface (m)	Density (Sigma-t)
0.0	22.9
14.0	25.0

Current velocity 0.1 m/s (Current Angle=90º)

Outfall Parameters:

Depth	16 m
Temperature	0ºC
Density	1 g/ml

Riser Parameters:

Distance	20 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

Jet Parameters:

Flow	0.0147 m³/s
Diameter	0.1 m
Port height	2 m
Vertical angle	90º
Horizontal angle	0º

General Notes:

1. Click **Open** in the startup tips window, then select and open file **tut4.vj** in the folder *tutorial files*

2. Animate the jet (Click the toolbar ) evolution and observe how the jet is bent over into a trapped submerged layer. Click **View suspend file** (in menu **More info** in Data output window) to see the printout of the computed results in a **SUSPEND** file.

3. In the **SUSPEND** file, you will find some information about trap level. For this case, the **neutral buoyancy level = 4.85 m**, with a corresponding **average dilution = 123.3**; the buoyant jet center **maximum rise height = 5.86 m**, corresponding average dilution = 225.4. The computation will be stopped at **trapped level = 5.16 m**, with corresponding **average dilution = 274**.

4. Click **Key parameters and length scales** (in menu **More Info** in Data output window) to see key input parameters and length scales that govern the mechanics of buoyant jet mixing. For example, discharge length scale l_Q = 0.089 m, momentum length scale l_m = 1.66 m, Buoyancy

length scale l_b = 3.52 m, jet/plume length scale l_M = 1.14 m, characteristic dilution for momentum dominated far field S_m = 18.7 and for buoyancy dominated far field S_b = 84.1.

5. Observe what happens when the ambient current is increased to 0.3 m/s. Input 0.3 under the Current column in ambient window, resimulate the model (Click the toolbar, then save this file. If you have problems, you can open the file **tut4a.vj** in the folder *tutorial files*). You will find the sewage will be more trapped as shown in screen figure or from **SUSPEND** file. In this case, **trapped level = 2.93 m**, with corresponding **average dilution = 390.7**.

6. Compute a vertical buoyant jet in **uniform crossflow** based on this case. You just need to change density to 1.025 at depth = 0 m in the input jet parameters, resimulate the model (Click the toolbar, then save this file. If you have problems, you can open the file **tut4b.vj** in the folder *tutorial files*). For this case, you will find that the sewage plume will not be trapped and reach the free surface.

8. Close this file to start the next tutorial, or change the parameter values to create your own run.

4.6 Example 5 Vertical dense jet in uniform cross flow

The file is **tut5.vj**.

Consider a single vertical **dense** jet discharged into a uniform crossflow. In some applications the jet or plume discharge is negatively buoyant – i.e. the effluent discharge is heavier than the ambient fluid (e.g. concentrated brine discharge from desalination plants). The main parameters are as follows:

Ambient Parameters:

Depth	40 m
Density	0.9756 g/ml
Current velocity	0.2 m/s (Current Angle=90º)

Outfall Parameters:

Depth	22 m
Temperature	0ºC
Density	1 g/ml

Riser Parameters:

Distance	0 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

Jet Parameters:

Flow	0.03 m^3/s
Diameter	0.1 m
Port height	2 m
Vertical angle	90º
Horizontal angle	0º

General Notes

1. Study how to modify input parameter based on file **tut1.vj**.
   i) At the startup window, select **Open**, open **tut1.vj** in the folder *tutorial files.*
   ii) Modify depth to 40 m and ambient density to 0.9756 g/ml, current to 0.2 m/s in ambient parameter window and flow to 0.03 m^3/s in jet parameters window.
   iii) Resimulate the model (Click the toolbar ! , save this file**.** If you have problems, you can open the file **tut5.vj** in the folder *tutorial files*).

   For this case, the ambient density ρ_a = 0.9756 g/ml is less than the jet density ρ_j = 1g/ml, it is a negatively buoyant discharge. On the screen, you will see the dense plume first moves upwards due to its initial momentum, reaches a maximum height, and then bends downwards to reach the seabed.

2. Open the **SUSPEND** file (Click **View suspend file** in menu **More info** in Data output window) to see the computed results: the buoyant jet center **maximum rise height = 2.69 m** and corresponding **average dilution = 43.9**. Plume hits the seabed (z = -20.0 m) with average dilution = 4865.

3. Click **Disk** in Data output window, the information on the Lagrangian elements (disks) is shown. For example, the disk 703# located at (8.3557, 0, 1.288) has the following properties: visual radius = 1.50 m, thickness = 0.0027 m, vertical angle = -19.6º, horizontal angle = 0, average velocity = 0.21 m/s, maximum concentration = 0.022, average dilution = 98.6 and average concentration=0.0101.

4. Close this file to start the next tutorial.

4.7 Example 6 Horizontal buoyant jet in uniform crossflow -Zarautz Marine Outfall, Spain

The file is **tut6.vj**

Consider a single horizontal buoyant jet discharged into a perpendicular uniform crossflow. Since the direction of jet momentum is different from the direction of buoyancy in the presence of the crossflow, the jet has a three-dimensional trajectory. The main parameters are as follows:

Ambient Parameters:

Depth	32.75 m
Density	1.0256 g/ml
Current velocity	0.2 m/s (Current Angle=90º)

Outfall Parameters:

Depth	34.75 m
Temperature	0ºC
Density	1.0 g/ml

Riser Parameters:

Distance	0 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

Jet Parameters:

Flow	0.025221 m^3/s
Diameter	0.12 m
Port height	2 m
Vertical angle	0º
Horizontal angle	90º

General Notes:

1. Study how to modify input parameter based on file **tut1.vj**.
   i) At the startup window, Select **Open**, open **tut1.vj** in the folder *tutorial files.*
   ii) Input depth to 32.75 m and current to 0.2 m/s in ambient parameter window.
   iii) Click **Outfall structure** in input parameter window, then click **Outfall 1** and modify depth to 34.75 m.
   iv) Click **jet1** and change flow to 0.025221 m^3/s, diameter to 0.12 m, vertical angle to 0º and horizontal angle to 90º.
   v) Resimulate the model (Click the toolbar, then save this file. If you have problems, you can open the file **tut6.vj** in the folder *tutorial files*).

2. Navigate through and view the jet from different angles.
   i) Click toolbar or press the right mouse button and select **Rotate**.
   ii) When the *cursor* is moved in the vertical direction of the screen, the rotation is about a horizontal axis.
   iii) If the *cursor* is moved in the horizontal direction of the screen, rotation is about a vertical axis.

3. Zoom and move screen figure.
   i) Click the toolbar or press the right mouse button and select **Zoom**. By moving the cursor upward, you can zoom out from the current setting. By moving the cursor downward, you can zoom in towards the center of current setting.
   ii) Click the toolbar or press the right mouse button and select **Move**. Press the mouse left button, you can move the viewing window up, down, left and right.

4. Use the animation particle function to see the evolution and spread of the jet as well as how the velocity changes as the plume comes up to the surface. Click the toolbar to see the evolution and spread of plume from the jet port to water surface.

5. Close this file to start the next tutorial.

4.8 Example 7 Horizontal buoyant jet in stratified crossflow -Zarautz Marine Outfall, Spain

The file is **tut7.vj**.

This is about the same single horizontal buoyant jet considered in Tutorial 6, but the jet is discharged into a stratified crossflow. Since the direction of jet momentum is different from the direction of buoyancy in the presence of the crossflow, the jet has a three-dimensional trajectory. The main parameters are as follows:

Ambient Parameters:

Depth (m)	Salinity (ppt)	Temperature (ºC)
0	15	30
15	22	25
32.75	34	20

Current velocity 0.2 m/s (Current Angle=90º)
(N.B. The salinity is given in parts per thousand, ppt)

Outfall Parameters:

Depth	34.75 m
Temperature	20ºC
Salinity	0.0 ppt

(N.B. The effluent salinity and temperature are specified rather than density)

Riser Parameters:

Distance	0 m
T. Radius	0.05 m
B. Radius	0.05 m
Height	2 m

Jet Parameters:

Flow	0.025221 m^3/s
Diameter	0.12 m
Port height	2 m
Vertical angle	0º
Horizontal angle	90º

General Notes:

1. At the startup window, select **Open**, open **tut6.vj** in the folder *tutorial files*

2. Modify ambient parameters and outfall parameters to the values shown above in the **input parameter** window. The density ρ will be automatically computed from the supplied salinity and temperature using the equation of state. Save this file. If you have problems, you can open the file **tut7.vj** in the folder *tutorial files*

3. Use the cutting plane function to view the horizontal section or vertical section of the jet, and examine some characteristics of jet cross-section.

i) Click toolbar or press the right mouse button and select the type of plane to display the cutting plane in **Cutting plane** parameter window.

a) Horizontal cutting plane

For example, by selecting the **Horizontal plane** and inputting the desired distance = 12 m, you will get a cutting horizontal plane located at z=12 m. You can also locate the cutting plane by directly clicking at any position at the jet section you like.

b) Vertical cutting plane

You may obtain the cutting vertical plane in a similar manner by selecting the **Vertical plane**; for example, by selecting the **Vertical plane (Side View)** and inputting desired distance = 3 m, a cutting vertical plane located at y=3 m will be obtained; for the **Vertical plane (Cross-section View)** selection and inputting the distance = 5 m, a cutting vertical plane located at x=5 m is obtained.

c) Normal and arbitrary cutting plane

You can obtain the cutting plane normal to the jet trajectory by selecting the **Normal plane** and inputting **Disk No**. The user can also define a cutting plane by specifying the horizontal and vertical angle of the plane in the **H. Angle** and **V. Angle**. The cross section of the jet cut by this plane is also shown in the small window on the screen, which is called **Cross section window**.

ii) Put the cursor in any point in the gray **Cross section**, you will get the position of this point and the concentration in **Concentration Info** window. The **area of the jet cross section** is shown in the **Area** window.

iii) Click any point in **Cross section** window, then you can use toolbar to zoom the **Cross section** or toolbar to move the **Cross section**.

4. Use **Continuous Mode** to continue computation of the buoyant jet spread after the jet is trapped by stratification.
   i) Select **Continuous Mode** in main menu **Option**
   ii) Resimulate the model (Click the toolbar)
   iii) Show the spread of jet (Click the toolbar).

5. Close this file to start next tutorial.

4.9 Example 8 Buoyant jets from a rosette-shaped ocean outfall riser in natural flow– Hong Kong Strategic Sewage Disposal Scheme (SSDS)

The file is **tut8.vj**.

In modern ocean outfalls, sewage effluent is often discharged through a number of adequately spaced outfall risers; the effluent is discharged as a jet group from each of the risers in a `rosette' like pattern. The planned ocean outfalls for the Hong Kong Strategic Sewage Disposal Scheme (SSDS), as well as the Shanghai Sewage Project Outfall, are examples of outfalls of this type. The main parameters for the SSDS outfall with a six-jet group are as follows:

Ambient Parameters:

Depth (m)	Salinity (ppt)	Temperature (ºC)
0	25.5	27.7
5.5	31.2	27.8
11	33.7	25.0
16.5	34.2	23.6
22	34.5	23.0

Current velocity 0.2 m/s (Current Angle=90º)

(N.B. The salinity is expressed in parts per thousand, ppt)

Outfall Parameters:

Depth	24.0 m
Temperature	25 ºC
Salinity	10 ppt

Riser Parameters:

Flow	0.94164 m^3/s
Distance	0 m
T. Radius	1.5 m
B. Radius	2 m
Height	3 m

Jet Parameters:

	jet 1	jet 2	jet 3	jet 4	jet 5	jet6
Flow (m^3/s)	0.15694	0.15694	0.15694	0.15694	0.15694	0.15694
Diameter (m)	0.25	0.25	0.25	0.25	0.25	0.25
Port height (m)	2	2	2	2	2	2
Vertical angle	0º	0º	0º	0º	0º	0º
Horizontal angle	0º	60º	120º	180º	240º	300º
Salinity (ppt)	10	10	10	10	10	10
Temperature (ºC)	25	25	25	25	25	25

General Notes:

1. In the **Cutting plane** window, the user can obtain a **Composite dilution for merged bent-over jets** when the **Vertical plane (cross-section view)** is selected.

2. At the startup window, select **<u>Open</u>**, open **tut8.vj** in the folder tutorial files

3. In this case, the measured salinity and temperature at different depths are used as input, linear variation is assumed between the consecutive adjacent levels. The sum of the six jet discharge flows is equal to the flow of the riser. All jets discharge horizontally (vertical angle = 0º); only the horizontal jet discharge angle (relative to the current) is different (0º = coflow; 90º = perpendicular crossflow; 180º = counterflow).

4. Use the animation function to see the computed rosette jet group pattern, and how different jets merge with each other. Click the toolbar to see active process of evolution and spread of the jet from source to trapped level. Use the **Rotate**, **Zoom**, and **Move** functions to view the jet cross-section from different angles. Observe how the plumes can merge with each other (even kinematically). The merging of the multiple plumes is related to the definition of **near field dilution** and mixing zone in environmental impact assessment.

5. View the jet from a horizontal plane and in a vertical section. i) Click toolbar and select **Horizontal plane** or **Vertical plane** with desired distance and input data in **Cutting Plane** window. For example, by selecting **Vertical plane (cross-section view)** and setting distance = 16 m, you will get a cutting vertical plane and this plane is also shown in small window in the screen---**Cross Section**. ii) Click any point in **Cross Section** window, then you can use toolbar to zoom the plane or use toolbar to move the plane in **Cross Section** window.

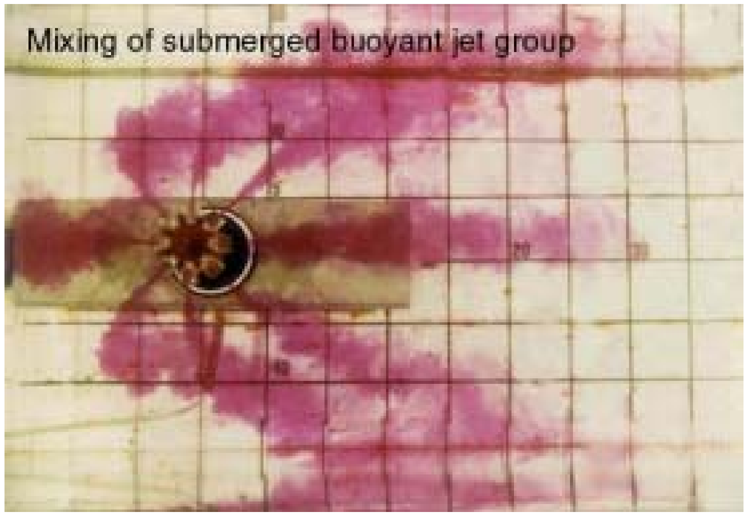
6. View some characteristics of jet cross-section.
   i) Put the cursor on any blue point in **Cross Section** window, you will get the position of this point and concentration in **<u>Concentration Info</u>** window. You will see that the concentration at a point where adjacent plume elements overlap is larger than that at a non-overlap point, because of the plume merging.
   ii) Click the jet number shown in **input parameter** window to get the jet cross-sectional area (area of this Lagrangian element) for the selected jet, the total plume area (with overlapped area subtracted) and sum of areas of the jets (summation of the projected area of each individual jet) in this cutting plane. For example, in this case with selecting **Vertical plane (cross-section view)** and setting distance = 16 m, the area of jet1, jet2, jet3, jet4, jet5 and jet6 is 7.9, 32.2, 66.9, 71.5, 66.9 and 32.2m^2 respectively, the sum of the areas of the six jets is 277.6 m^2, and total jet area in the plane is 214.7 m^2 (excluding overlap). The ratio of the total jet area to the sum of areas is a measure of reduced dilution due to plume merging. You can also obtain the composite dilution in the **Cutting plane** window. **(Note: the computed composite dilution is only valid up to the point where the plume surfaces or settles to an equilibrium level)**

7. Create three identical six-jet risers and observe the merging between adjacent jets, and also between plumes from adjacent risers, in both uniform flow and under stratified ambient conditions.

8. Create different orientations of the diffuser axis with respect to the current by changing the current angle in the **ambient parameter** window as well as multiple diffusers.

9. Try to explore all the functions on the preceding tutorials 1-7.

The following photo shows the observed jet mixing of an 8-jet rosette-jet group in a laboratory experiment.



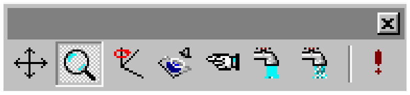
Congratulations!

You have successfully finished all the tutorials.

5. Advanced graphics features (for experienced users)

5.1 Main components

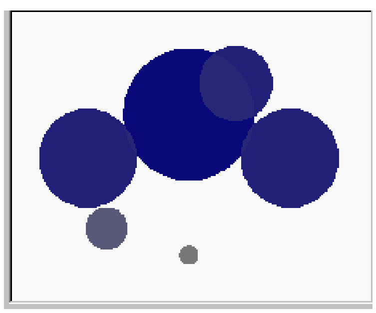
5.1.1 Toolbar



The toolbar can be moved around the whole screen and provides quick access to various actions for manipulating the images in the 3D outfall View and the Cross Section View.

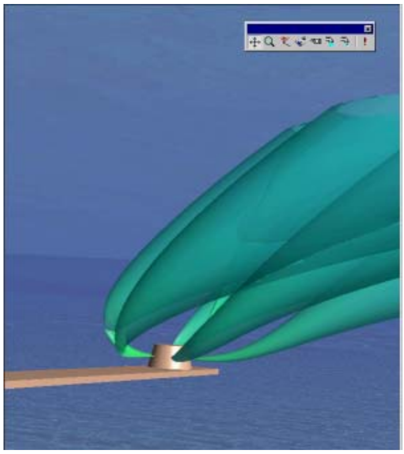
Click	To
	move
	zoom
	rotate
	cutting plane
	pick
	solid animation
	particle tracing
	refresh

5.1.2 Cross section view



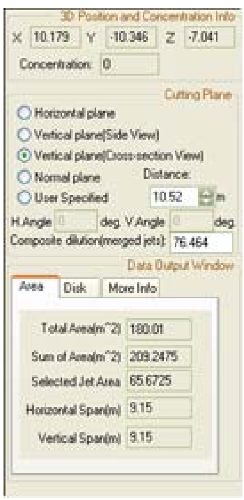
The Cross Section (cutting plane projection) View shows the cross sections of the buoyant jets projected on the cutting plane. The jet sections are coloured. Moving the mouse or pointing device inside the panel and click, this view will be selected. Moving the mouse with the left button being pressed down, the cross-section will be zoomed in or out. Moving the mouse with the left button and the key CTRL being pressed down, you can drag the cross-section into any position in this view.

5.1.3 3D outfall view



This view provides the users with the 3D animation of the jets simulated. Users can obtain different look of the jets from different angles.

5.1.4 Result data view



This panel displays the information resulted from the simulation. These include: the disk information, the cross section concentration and area. Information about the cutting plane is provided.

5.2 Graphics manipulation

5.2.1 Actions

5.2.1.1 Zoom

Select  or press the right mouse button and select “Zoom”.

By moving the cursor upward, you can zoom out from the current setting.
By moving the cursor downward, you can zoom in towards the centre of current setting.

5.2.1.2 Move

Select  or press the right mouse button and select “Move”.

Simply using the mouse or pointing device will allow you to move the viewing window to where you want.

5.2.1.3 Rotate

Select  or press the right mouse button and select “Rotate”.

By moving it in the vertical direction, rotation can be made about the horizontal axis running along the screen. If the cursor is moved in the horizontal direction, rotation will be made about the vertical axis running along the screen.

5.2.1.4 Cutting plane

Select  or press the right mouse button and select “Plane”.

Five options are provided when defining a cut plane:

(1) Horizontal plane--- the plane parallel to the surface of the sea.
(2) Front vertical plane--- the plane parallel to the current.
(3) Side vertical plane--- the plane normal to the current.
(4) Normal plane---the cross-section plane normal to the jet trajectory.
(5) User-specified plane---two angles need to be specified by the user. The meanings of the angles are shown in the cutting plane parameters.

5.2.1.5 Pick

Select  or press the right mouse button and select “Pick”.

Using the cursor, the user can pick or select any point on the jets and display the computed properties of the disk (including the centre’s position, radius, thickness, orientation, concentration and velocities) containing that point.

5.2.1.6 Solid animation

Select 

The animation of the jet evolution will be re-run.

5.2.1.7 Particle tracing

Select 

Tracer pattern following the fluid gives a feeling for the velocity at different elevations.

5.2.1.8 Refresh

Select 

The evolution process of the jets will be re-run in the 3D Outfall View.

5.2.2 Option

5.2.2.1 Fast display mode

Press the right mouse button and select “Fast Display”.

The fast display mode has less demand upon the graphics capability of the computer and allows a faster response in changing the displayed graphics. When displaying multiple jets, the normal display mode has the transparent effect. The user could tell which jet is far from him and which is near to him. The fast display mode does not have the transparent effect, the user would not see the jet which is hindered by the front one. Also, with the fast mode, the user could not see the change of concentration on the jet.

5.2.2.2 Display cutting plane

Press the right mouse button and select “Display Cutplane”.
The cutting plan can be displayed or hidden.

5.2.2.3 Show concentration change

Press the right mouse button and select “Show Concentration Change”.
Display the change in concentration by gradual change in colour.

5.2.2.4 Show velocity change

Press the right mouse button and select “Show Velocity Change”.
Display the change in velocity by gradual change in colour.

5.3 Option menu command

The Option Menu offers the following commands:

Animation speed	Select the animation speed among “fast”, “medium” and “slow”
Continuous mode	Require the simulation to continue up to the maximum simulation step
Max number of steps	Allow the user to specify the maximum number of simulation steps (default no. is 1500)
Disturbance of the water surface	Show or stop the animated disturbance of the water surface
Water surface property	Allow the user to change the display properties of the water surface