VISJET

Version 3.0
Copyright 2007 The University of Hong Kong

Documentations

Jointly developed by
Department of Civil Engineering, HKU
Department of Computer Science, HKU

Supported by Industrial Support Fund ITF ----

Table of Contents

Introduction

From Visjet 2 to Visjet 3

The VISJET2 model (2002) developed previously was a stand-alone near field model that assumes arbitrary steady environmental conditions, and predicts the 3D geometry and concentration of an arbitrarily inclined buoyant plume in stratified cross flow - with advanced virtual reality graphics features. There was no modeling of any realistic background terrain feature; the far field concentration and impact was not modeled. Since then, significant progress to advance the hydro-environmental technology has been made as follows:

1.   Based on the Distributed Entrainment Sink Approach (DESA), a three-dimensional environmental modeling system has been developed. The model predicts the mixing and transport from the near to far field in a seamless manner – by dynamic coupling of the near field VISJET model and the far field shallow water circulation model at grid cell level. The method is validated for a variety of complex flow scenarios including the interaction of a confined rising plume with ambient stratification and mixing of rosette jet group in a cross flow. Numerical predictions are in excellent agreement with basic data of other investigators and experiments performed at the Croucher Laboratory of Environmental Hydraulics.

2.   The predictions of the 3D modeling engine are fully integrated with the GIS information and the 3D visualization. Using Tolo Harbour and Port Shelter as the pilot systems, the new technology enables true 3D visualization of the environmental impact from very close to a pollution source (near field, of the order of the water depth) to the sensitive receiver (e.g. beaches in the far field, perhaps several kilometers away). This enables effective visualization of the environmental impact of a pollution source (e.g. an upgraded sewage treatment plant with increased population loading) in a way that allows data interrogation and public engagement. Specifically, the pilot system has the following unique features:

   a) The Hong Kong terrain elevation and bathymetry has been incorporated into the system. To increase visual realism, digital aerial photos for Tolo Harbour and Port Shelter are used to map textures to the 3D terrain.
   b) Standard threshold concentration contours are built into the system to indicate areas affected to different degrees. This feature is particularly useful to examine the impact of pollution discharges on nearby beaches (e.g. experience has shown that

virus in sewage discharge is not likely to cause any health effects after a dilution of around 500.)

c) The vectors representing the flow field and the color-coded concentration results are directly superimposed on the water surface to produce a realistic 3D view of the environment.

d) Fish farms and beaches are two common sensitive receivers for marine pollution in Hong Kong. Past key water quality data and photographic features at a fish culture zone are integrated with the system. Hence at a sensitive receiver future predictions can be contrasted with past observations at the point of interest. The predicted time history of concentration can also be shown.

e) 3D navigation and data interrogation are also completed. The users can freely navigate in 3D environment and obtain an idea of: (i) the extent and severity of the pollution and the affected beaches and coastal areas; (ii) a 3D view of the coastal water from any perspective angle; (iii) time history of the predicted concentration for any point on the water surface. All the above features would encourage and assist with public engagement on environmental projects.

The Main Interface

The main interface consists of two main components: the input panel and the 3D view panel.

The Input Panel

The input panel consists of two smaller panels. The upper one is used to define the ambient flow conditions used for near field simulation. The lower one is used to construct the outfall configuration.

3D Navigation

The 3D view panel is used to display the actual location of the outfall, the outfall configurations and the visualization of near- field simulation. GIS information such as 3D landform is integrated into the view to give the user a better feeling of the actual scale and possible impact of the design.

The 3D view requires a bit of getting used to in order to navigate through the 3D space freely. A mouse with scroll wheel is required. The general idea of the 3D navigation revolves around a viewing centre. The centre is always on the water surface and is always mapped to the centre point of the 3D view panel. To move around the 3D space, either rotate about the centre (changing view direction), move near or away from the centre (zoom in/out) or simple shift the centre to a new position (move).

Rotation

Click and drag the left mouse button to rotate the view about the centre point on the surface. Notice the centre point does not move as it is the centre of rotation.

Zooming

Rotate the mouse wheel to zoom in and out. When zooming in, the viewpoint is essentially moving towards the centre point along the view direction. Therefore, again the centre is not moving. When zooming out, the viewpoint is moving away from the centre.

Moving

Moving is accomplished by selecting a new viewing centre. The new centre can be selected by left clicking, without dragging, on any point on the 3D view panel. The clicked water surface point will be the new viewing centre and it will be moved to the centre of the panel.

General Workflow

The VISJET 3 system consists of 4 major components. The main input interface, the near field JETLAG simulator, the near and far field DESA simulator and a visualizer for visualizing the results generated by the DESA simulator.

The general workflow is shown in the following figure. The .vj3 file is the project file. The .vjx file is an intermediate file used by the visualizer only. The input interface is used to generate the .vj3 file and other simulator related files like UDF (used for near field simulation) and all the *.inp files (used for DESA simulation). All files related to the project must be put in the same project folder. The input interface will create the project folder and the associated files in the folder.

The Visualizer Interface

Ambient Current Flow

Near field ambient input panel can be used to define a stratified flow used for near field simulation. Each layer in the stratified flow can have its own temperature, salinity and current flow speed. To add a layer of flow, click the add button. Usually the first row has a depth of 0 meter that defines the conditions at the water surface. Change the values of salinity, temperature and current flow speed by clicking the corresponding cells in the ambient table and edit their values directly. Click add to add new layers of flows. By default, the depth will be increased by 20 meters for each new layer. Change the depth of the added layer and add subsequently added layers will have depth increment equal to the previous one. i.e. if the first layer's depth is 0 meter and the second layer's depth is changed to 5 meter, the third added layer will have depth of 10 meter automatically. Other values are duplicated from the previously added layer.

Outfall Construction

Outfall, riser and jet

With the modelling engines fully integrated with the GIS information, we can design the outfalls within the supported regions in Hong Kong (Tolo Harbour and Port Shelter). After selected the region, we can define the location of the outfall when a new outfall is created. It is also possible to relocate an existing outfall. The location can be defined either picking the position interactively or entering the co-ordinate directly. The outfall diffuser can then be rotated to the desired orientation. Then, the outfall geometry and effluent properties are to be specified.

Same as VISJET2, the outfall can be with and without riser. For the case with riser, multiple jets can be defined to form the rosette type of discharge. Without the riser, jets can be arranged along the diffuser pipe at user-defined positions.
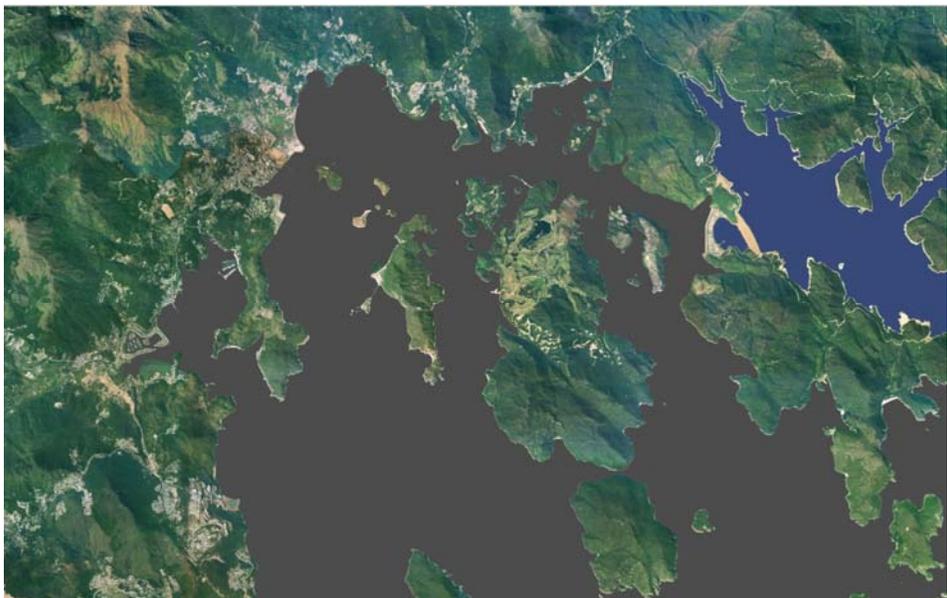
Scenarios

Supported scenarios

To adapt to the two scenarios (Tolo Harbour and Inner Port Shelter, Sai Kung), the basic information such as static GIS data like landforms and sea bed levels are stored in the system.

Tolo Harbour

Inner Port Shelter, Sai Kung

Simulations

Near-field Simulation

The near field simulation engine is JETLAG, the same one for VISJET2. JETLAG is a well-proven robust jet model that predicts the mixing of an arbitrarily-inclined round buoyant jet in a stratified crossflow, with a three-dimensional trajectory. It tracks the evolution of the average properties of a plume element by conservation of horizontal and vertical momentum, conservation of mass accounting for shear and vortex entrainment, and conservation of solute or tracer mass/heat. For a given set of ambient conditions (vertical profile of horizontal velocity, density, and tracer concentration), the jet trajectory, jet velocity and radius, and dilution (entrainment) can be predicted. The turbulent entrainment along the jet trajectory can then be obtained from the discharge point up to a terminal level (free surface, bottom, or trap level in the presence of ambient density stratification).

Near-and-far Field Simulation

From the viewpoint of the surrounding water, the important near field flows are the bulk sink flows ("loss") due to the turbulent jet entrainment and the bulk source flows ("gain") due to the diluted discharges. In order to have a true two-way coupling, it is proposed to represent the discharge by introducing in the far field model: (i) a series of entrainment sinks along the predicted plume trajectory; and (ii) the diluted flow and pollution loading at the predicted terminal level of plume rise. The sources and sinks are determined from a near field model embedded within the far field model. With the new Distributed Entrainment Sink Approach (DESA), the near field mixing is fully resolved by the sub-grid plume model; the bulk fluid and tracer mass transport induced by the effluent discharge as well as the influence of background concentrations or recirculation on the near field can be fully modelled in a dynamic manner. This interaction of the plume and the surrounding flow induces additional mixing beyond the near field. The effect of unsteady evolution of the ambient flow on the plume trajectory and dilution, as well as the changes in plume mixing on the ambient flow are fully accounted for. The near field mixing is modelled by JETLAG, while the far field transport is simulated by the Environmental Fluid Dynamics Code (EFDC). EFDC is a finite difference model that solves the shallow water equations. The model uses a staggered grid for discretization and a sigma-grid co-ordinate in vertical direction. The modal splitting technique is used to solve the discretized equations. The near field mixing can be satisfactorily represented in the far field model by embedding the near field predictions at "grid cell" level. The boundary conditions for the

near field model depend on the far field flow, which can be obtained from a 3D circulation model.

Visualization

Invoked from the Main Interface

Run from Start Menu

VISJET3 Tutorials

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 modelled 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.

In VISJET3, jet discharges are simulated against a real 3D background of the Hong Kong topology, in which the discharge and the topology can be viewed at any orientation. In this way the environmental impact to nearby sensitive receivers can easily be visualized and assessed. In this series of tutorials, we will illustrate this can be done step by step, starting from the simple case of a single buoyant plume discharge, to a more complicated discharge of a rosette jet group, and finally end with a real outfall located in Sai Kung.

Tutorial 1 – Buoyant jet in unstratified stagnant fluid/crossflow.

Tutorial 2 – Buoyant jet in stratified stagnant fluid/crossflow with a riser.

Tutorial 3 – An example outfall in the Port Shelter.

Tutorial 4 - Discharge of rosette jet groups from multiple risers : the HATS Outfall.

Tutorial 5 - Round buoyant jet in a confined region - dynamic coupling of jet and surrounding fluid.

Tutorial 6 - Environmental impact assessment for Sai Kung Sewage Treatment Works (SKSTW) Outfall.

Tutorial 1 – Buoyant jet in unstratified stagnant fluid/crossflow (file Tut_1.vj3)

Problem definition

An outfall located in Port Shelter discharges effluent at a rate of 0.03 m³/s with a concentration of 100 units through a diffuser into an otherwise stagnant ambient. The effluent receives primary treatment; the remaining BOD_5 in the effluent is 200 mg/L; the suspended solids concentration is 90 mg/l; the total coliform count is 5×10^6 counts/100 mL. The temperature and salinity of the ambient are 15°C and 34.5 ppt respectively; and these for the discharge are 20°C and 1 ppt respectively. You are asked to study the mixing characteristics of the outfall, such as the dilution and trajectory under different design conditions, assuming zero background concentration in the ambient. Tides are present so the tidal current effect should also be taken into account.

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:

Basic Data:

Outfall Parameters:
Depth (z)	20 m
Flow	0.03 m³/s
Temperature	20 °C
Salinity	1 ppt
Concentration	1 relative units
Length	5 m
Diameter	2 m
Distance	2.5 m
Orientation	0°

Ambient Parameters:
Depth	20 m
Temperature	15 °C
Salinity	34.5 ppt
Current velocity	0.0 m/s (Current angle 90°)

Jet Parameters:

Diameter	0.1 m
Distance	2.5 m
Vertical angle	90º
Horizontal angle	0º

General Notes:

1. *Creating a scenario*
(i) Click **File** at the menu.
(ii) To create a new scenario, select **New**.
(iii) Give a name to the scenario, type in Tut_1 at the Name blank.
(iv) Select a folder path to save the scenario.
(v) Select a Hong Kong location for the simulation. In this tutorial, we will use Port Shelter as an example. Click on Port Shelter.
(vi) The "type of simulation" does not have any effect on this near-field simulation; you can just select any type. In later tutorials when the far field simulation is performed, the appropriate type needs to be specified

A folder with the name Tut_1 will automatically be created; it is where all the grid information and the VISJET file are contained; check whether it exists in your specified path.

2. *Navigate around the selected topology*
(i) Left click any point on the screen, the clicked point will automatically become the centre point of the screen.
(ii) Scrolling down and up will allow you to zoom in and out with the selected point in step (i) as centre
(iii) left click and hold the mouse button, and freely move the mouse to view the topology at different orientations
(iv) Select "View" at the menu; you will be able to select what geographical information to be displayed. "Landform", "Seabed", and "Outfall" have been pre-selected. Now click on "Water surface" to view the water surface as well.

3. *Select an outfall location*
(i) Click "**create an outfall without Riser**" to create an outfall.
(ii) Left click a point on the screen close to the desired outfall location.
(iii) Click the "Pick" button on the outfall parameters panel.
(iv) Move the mouse cursor to the desired outfall location, and left click.
(v) The outfall will then appear at the position of the cursor.
(vi) The coordinates of the outfall will automatically be calculated and are shown as the X and Y coordinates in the outfall position panel.

4. *Input ambient parameters*
(i) Click "**New**" once just below the "**Ambient parameters window**".
(ii) Input the ambient parameters as given above:

Depth	Salinity	Temperature	Current
0	34.5	15.0	0.0
20	34.5	15.0	0.0

(iii) Keep the "**current angle"** as 90º for this scenario.

(a) Vertical Jet in stagnant fluid

We begin with the simplest case of vertical jet in stagnant fluid.

(i) Highlight **Outfall #01**
(ii) Input the outfall parameters as given above
(iii) Click "**New Jet"**
(vi) Highlight **Jet #01**
(v) Input the jet parameters as given above
You are now ready to run the simulation

5. Click "**Start Near Field VISJET simulation**" ▶ at the toolbar. You can see the evolution and spread of the jet. 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. (Note that you may need to zoom in quite a bit to see the jet and outfall as their size is small relative to the whole Port Shelter.)

6. Study how to get the *jet characteristics* of each Lagrangian element: the computed average velocity, maximum concentration, and average dilution at that height.

(i) Zoom in to the jet and click on any position of the jet you want to get the information
(ii) Now you can see the 'disk' position being display by a red line on the surface of the jet
(iii) The pop up disk properties windows give various jet information of that particular disk. For example, the centre of Disk #605 is located at (0.00, 0.00.10.74), visual radius = 1.2438 m, thickness = 0.0026 m, vertical angle = 90º, horizontal angle = 0º, average velocity = 0.197 m/s, maximum concentration = 0.0267, average dilution S=63.74 and average concentration = 0.0157.

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.

7. Zoom in closely to the jet. Try to obtain the disk properties close to the jet nozzle. For example, disk #63 is located at (0.00,0.00,0.16), Visual radius = 0.0545 m, thickness = 0.0325 m, vertical angle = 90°, horizontal angle = 0°, average velocity = 2.48 m/s, maximum concentration = 1.000, average dilution S=1.54 and average concentration = 0.6476. At a location so close to the source, not much dilution has taken place yet.

8. Try to obtain the disk properties near the water surface. For example, disk #665 is located at (0.00,0.00,14.18), Visual radius = 1.6073 m, thickness = 0.0023 m, vertical angle = 90°, horizontal angle = 0°, average velocity = 0.1780 m/s, maximum concentration = 0.0176, average dilution S=96.53 and average concentration = 0.0104. By the time the plume has reaches the water surface, a considerable amount of dilution has taken place as water is continuously being entrained into the plume.

9. *Save file*: you can save the current scenario by selecting Save in the main menu File (file format .vj3) for later use.

(b) Horizontal Jet in stagnant fluid – effect of varying jet angle

10. Now we are going to study the effect of varying the jet angle to the jet mixing characteristics.
(i) Keep all discharge parameters unchanged as in part (a) except changing the vertical jet angle from 90° to 0°. We now have a horizontal jet instead of a vertical jet discharge.
(ii) Run the simulation again by clicking "**Start Near Field VISJET simulation**"   at the toolbar.
(iii) Observe the jet evolution and spread from the animation. This time instead of seeing a plume going straight upwards, you would see a jet initially going horizontally, and gradually bends upwards due to the buoyancy force and finally reaches the water surface.

11. Zoom in closely to the jet. Try to obtain the disk properties close to the jet nozzle. For example, disk #65 is located at (0.17,0.00,0.00), Visual radius = 0.0554 m, thickness = 0.0319 m, vertical angle = 0.1469°, horizontal angle = 0°, average velocity = 2.44 m/s, maximum concentration = 1.000, average dilution S=1.57 and average concentration = 0.6387. At this location very close to the source, similar to the vertical jet, not much dilution has taken place.

12. Now try to obtain the disk properties near the water surface. For example disk #707 is located at (9.66,0.00,13.24), Visual radius = 1.9457 m, thickness = 0.0021 m, vertical angle = 79.7968°, horizontal angle = 0°, average velocity = 0.1620 m/s, maximum concentration = 0.0132,

average dilution S=129.07 and average concentration = 0.0077. Compare the dilution at water surface of the horizontal jet (S=129.07) with the vertical jet (S=96.53), the horizontal jet has a greater dilution capability!

A horizontal jet entrains more ambient fluid into itself than a vertical jet, resulting in a higher dilution since the jet path of a horizontal jet is longer than that of a vertical jet!

(c) Vertical Jet in a crossflow – Effect of a current

The marine environment is not stagnant for most of time. With even just a slight tidal current the mixing characteristics of a jet can be significantly different. VISJET can illustrate this easily.

13. Returning back to the case of vertical jet by changing the vertical angle of the jet from 0° to 90°. Run the simulation again by clicking **"Start Near-field VISJET** ▶ **simulation"**.

14. Study the use of **"SUSPEND FILE"**
To obtain the computed result, instead of clicking each disk directly, we may open the SUSPEND file. To open the SUSPEND file, click **"View"** at the main menu, and select **"view SUSPEND file"**. The SUSPEND file shows key input parameters and length scales that govern the mechanics of buoyant jet mixing. For example, Total Q = 0.03 m³/s, jet densimetric Froude number F_d = 23.96, buoyancy flux B_j = 0.00762 m⁴/s², jet momentum length scale l_M = 2.2556, 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 the surface layer level and the surface dilution. For this case, the surface layer level is 16.40 m above the discharge port, and the average dilution is S=120.19. Note that the partial blocking by the free surface is taken into account in this calculation (dilution only up to the base of the surface layer).

15. Study the effect of a current.
(i) Keep all the discharge parameters unchanged except changing the ambient current speed from 0.0 m/s to 0.1 m/s for both depths 0 m and 20 m in the ambient parameters window.
(ii) Re the simulation again by clicking **"Start Near-field VISJET simulation".** ▶
(iii) Observe again the jet evolution and spread. Notice any differences. Note that for this time the jet no longer goes straight upwards due to the effect of the current. Instead, the jet is bent over by the current soon after discharge (current direction is indicated by the red arrow near the outfall).
(iv) View the **"SUSPEND FILE"** again, and compare different mixing characteristics with the case with a current.
(v) The dilution for this case is found to be S=398.12 at z=13.42 m above the port. Compare the dilution with the case without a current (S=96.53).

The dilution is increased for nearly 4 times for just a small current!

16. Try out the effect of current of different magnitude. Study their effect on the jet mixing characteristics.

(d) Multiple vertical jets in a crossflow – Merit of multiport diffuser design

17. Consider the vertical buoyant jet in Tutorial 1 again, but this time instead of using one jet, divide the flow of 0.03 m³/s in to 4 jets, each discharge from a different port. So now we have FOUR vertical buoyant jets in a uniform stagnant fluid, but each port discharges ¼ of the original flow. The outfall and jet parameters of the multiport diffuser are as follows:

Outfall Parameters:

Depth (z)	20 m
Flow	0.03 m³/s
Temperature	20°C
Salinity	1 ppt
Concentration	1 (relative unit)
Length	60 m
Diameter	2 m
Distance	30 m
Orientation	0º

Jet Parameters:

Flow	0.0075 m³/s
Diameter	0.1 m
Vertical angle	90º
Horizontal angle	0º
Distance:	0 m (jet 1)
	20 m (jet 2)
	40 m (jet 3)
	60 m (jet 4)

18. You can change the orientation of the diffuser with respect to the current by entering the diffuser angle at the **"Angle"** blank.

19. Change the outfall dimensions and parameters as given above (location of the outfall is unchanged).

20. Create the first jet by clicking the **"New Jet"** button. Enter the jet parameters as given above (Note that you don't need to input the Flow

for the jet, it will automatically be calculated as Outfall Flow/number of jets). Create the remaining jet by clicking the **"New Jet"** button; pay attention especially to the distance of each jet. (Instead of entering the distance of each jet manually, a short cut is to first create an outfall with length 60 m, then click the **"Distribute jets evenly"** button, jets will then be automatically separated at a distance (s) equal to L/(n-1) where L is the diffuser length and n is the number of jets.

21. Re-run the **"Near-field VISJET ꙮ simulation"** and see the evolution of the jets. At this jet separation distance, do neighbouring jets interact with each other (do they overlap)?

22. You can see more clearly on the jet interaction by cutting a vertical plane on the jet group. To view the cross-sectional plane of the jet group, choose vertical plane and click **"toggle clip plane"** ꙮ at the menu bar. You would then see a clip plane is created on the main screen. The green circle on the clip plane is used to translate the position of the clip plane. Left click and hold the green circle to move back and fro; you would see the cross-sectional view of the jet group. The yellow triangle of the clip plane is used to rotate the angle of the clip plane, try left click and hold the yellow triangle to move left and right; you can see the angle of the clip plane being rotated. In this way the extent of merging of the rosette jet group can clearly be visualized.

23. Study the mixing characteristics of the multiple jet discharge.
(i) Open the **"SUSPEND FILE"**.
(ii) The mixing characteristics of each jet is tabulated. For example in Jet #01, Total Q = 0.0075 m³/s, jet densimetric Froude number F_d = 5.99, buoyancy flux B_j = 0.00191 m⁴/s², jet momentum length scale l_M = 0.5639, and so on. Note the difference between these and the case with only a single port discharge.
(iii) Study the dilution of the jet group. The average dilution of this case is S=2304 at 12.13 m above the port, much higher than that of a single port discharge!

Multiport diffuser give significantly higher dilution than that of a single port discharge!

24. *Multiple jet interaction*.
(i) Now change the outfall length from 60 m to 10 m. Follow the steps in 20. to distribute the jets evenly along the diffuser (jet spacing should be 3.3 m).
(ii) Run the simulation again. Can you see the interaction (overlapping) of the jets? Try to toggle the clip plane and cut a cross-section at different position of the jet group and see the extent of overlapping.

(iii) For closely spaced jets, they would interact and merge with each other, causing a reduction in dilution. This merging effect has been taken into account by VISJET through computation of the overlapped area. The resulting composite dilution is available in the VISJET 2 software.

Calculations of concentration of water quality parameters
It is given in the problem that the BOD_5 in the effluent is 200 mg/l; the suspended solids in the effluent is 90 mg/L; and the coliform count is $5x10^6$ counts/100 mL. With the near field dilution computed by the VISJET model, we now can calculate the concentration of these quantities at the water surface at the end of the initial dilution zone, assuming zero concentration background in the ambient.

Example calculation:
For the case of a vertical jet in stagnant water, average dilution obtained S= 96.53.
Thus the concentration of BOD_5 for this case is 200/96.53 = 2.07 mg/L
Concentration of the suspended solid = 90/96.53 = 0.93 mg/L
Coliform count = $5x10^6$ / 96.53 = 5.2 x 10^4 counts/100mL

The water quality parameters at the end of the initial dilution zone are summarised as follows:

Quantities	Vertical jet in stagnant water	Horizontal Jet in stagnant water	Vertical jet in a crossflow	Multiple vertical jets in a crossflow
Dilution	96.53	129.07	398.12	2304
BOD_5 (mg/L)	2.07	1.55	0.50	0.087
Suspended solid (mg/L)	0.93	0.70	0.23	0.04
Coliform (counts/100 mL)	5.2 x 10^4	3.9 x 10^4	1.26 x 10^4	2.2 x 10

25. Close this file and start the next tutorial.

Tutorial 2 – Buoyant jet in stratified stagnant fluid/crossflow with a riser *(file Tut_2.vj3)*

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/crossflow.

Quite often the outfall is constructed deep under the seabed; in this case a riser is needed to bring the effluent from the underground outfall back to the sea. In this example we will also illustrate the use of a riser.

Basic Data:

Ambient Parameters:

Depth below surface (m)	Temperature (^{o}C)	Salinity (ppt)
0	20	12.9
1	20	13.4
4	20	16.8
7	20	18.4
10	20	19.8
13	20	22.5
16	20	24.1
19	20	24.2
20	20	24.2

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

Jet Parameters:
Diameter   0.125 m
Port height   30 m
Vertical angle   0º
Horizontal angle   0º

Outfall Parameters:
Depth (z)   50 m
Flow   0.0147 m³/s
Temperature   20 ºC
Salinity   1 ppt
Concentration   1 relative units
Length   5 m
Width   2 m

Distance	2.5 m
Orientation	0°

Riser Parameters:

Distance	2.5 m
T. Radius	0.3 m
B. Radius	0.3 m
Height	30 m

(a) Vertical jet in stagnant ambient

General Notes:

1. Start up a new scenario in Port Shelter as instructed in Tutorial 1.

2. Enter ambient parameters: To add more layers of ambient parameters, click the button **"New"** just below the ambient parameters window. To delete extra ambient layer, click the button "**Delete**". Input the ambient parameters as given above.

3. Choose a location for the outfall as in Tutorial 1. Enter outfall parameters: Select **"Create an Outfall with Risers"**. Enter outfall parameters as given above.

4. Enter riser parameters: Enter the riser parameters as given above. Note that the riser height should be tall enough such that it can penetrate the seabed. Mathematically, Outfall depth – riser height ≤ Ambient depth, for example in this case outfall depth (50 m) – riser height (30 m) ≤ Ambient depth (20 m).

5. Enter jet parameters as given above by clicking **"New Jet"**. Note that port height again needs to satisfy outfall depth – port height ≤ Ambient depth. In addition the port height cannot be greater than the riser height.

6. Run the **"Start Near-field VISJET simulation"**.
The jet evolution can be seen. Note the major difference this time from the previous tutorial is that the plume is trapped under the water surface due to the effect of ambient stratification.

7. To obtain the computed result, instead of clicking each disk directly, we may open the SUSPEND file. To open the SUSPEND file, click **"View"** at the main menu, and select **"view SUSPEND file"**. The SUSPEND file shows key input parameters and length scales that govern the mechanics of buoyant jet mixing. For example, Total Q = 0.0147 (m³/s), jet densimetric Froude number Fd =8.22, Buoyancy flux Bj = 0.0026 m4/s², jet momentum length scale lM = 0.9482, 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 = 5.19 m, with a corresponding average dilution S=37.7 and visual radius = 1.61 m; jet centre maximum rise height = 7.34 m, corresponding average dilution S= 40.5 and visual radius = 2.55 m.

8. In the 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.

(b) Horizontal jet in stagnant ambient

9. Jet angle does play a part in determining the dilution of the discharge. In this example, keep all the input parameters unchanged except change the vertical jet angle from 0° (horizontal) to 90° (vertical), and re-run the simulation.

10. Open the SUSPEND file and check the dilution. At the neutral buoyancy level (5.88 m), the dilution is S=27.4. At the maximum rise level (8.30 m), the dilution is S=31.2. Both dilution are less than that of a horizontal jet, this is due to the shorter entrainment path of the vertical jet.

(c) Horizontal jet in a stratified crossflow

11. We can similarly study the effect of a current as in Tutorial 1 in a stratified ambient condition.
(i) Keep the discharge parameters unchanged except changing the current speed of each ambient layer to 0.1 m/s as instructed in tutorial 1.
(ii) Run the simulation again by clicking **"Start Near-Field VISJET simulation"**.
(iii) Observe the evolution and spread of the jet. It can be seen that the jet oscillates and is trapped under the water surface due to the effect of the ambient stratification.
(iv) Open the **"SUSPEND FILE"** to study different mixing characteristics of the jet.
(v) It can be seen that the neutral buoyancy level is 5.1815 m above the discharge port with a dilution of S=125.42. Visual radius is 2.38 m at this level. The maximum rise of the jet is 6.04 m above the discharge port and the average dilution is S=226.14 at this level.

Tutorial 3 – An example outfall in the Port Shelter

After going through Tutorial 1 and 2, it is assumed users have already gained familiarity to using the VISJET 3 interface, and how to access the information provided. In this tutorial, we will illustrate how the VISJET 3 can be used in assessing the impact of an outfall, which is often useful for choosing an outfall location and detailed outfall design.

Problem definition

In order to meet the projected flow and loads generated from future development and planned extension of the sewerage collection system of Sai Kung, it is proposed that a new outfall should be constructed. The projected Average Dry Weather Flow (ADWF) is 22,000 m^3/d, and the peaking factor is 3.1, giving 68,200 m^3/d as the design flow of the outfall. The wet weather flow is about 30% larger and is given by 28,600 m^3/d. The proposed outfall has a length of 90 m at a water depth of 15 m. 18 ports are proposed to be installed at the diffuser, with port diameter of 0.15 m.

The proposed standards of the partially treated wastewater from the treatment plant are:

	95 percentile	Maximum
BOD_5 (mg/l)	20	40
Total suspended solid (mg/l)	30	60
Unionized ammonia (mg/l)	-	0.9 (assumed)
E. coli (counts/100 ml)	1500	-

The measure ambient marine water quality and the target water quality objectives are given as follows:

Parameter	Measured ambient level	Water quality objectives
Temperature (^{o}C)	23.6 (16.3 – 28.4)	Not cause daily temperature range to change more than 2°C
Salinity (psu)	31.4 (28.3 – 33.1)	Not cause natural salinity level to change more than 10 %
SS(mg/l)	2.1 (0.7 – 4.2)	Not to cause natural ambient level to be raised by 30%
BOD_5 (mg/l)	0.8 (0.3 – 1.9)	Not to exceed 5 mg/l for inland waters
Unionized Ammonia (mg/l)	0.001 (<0.001- 0.010)	Not more than annual average of 0.021 mg/l

E. coli (count/100ml)	1 (1-3)	Not exceed 610/100ml at fish culture subzones Not exceed 180/100 ml at bathing beach subzones

The ambient stratification of the dry weather and the wet weather is given as follows:

Dry Weather:

Water depth (m)	Temperature (^{o}C)	Salinity (ppt)
0	23.5	33.2
7.5	23.2	33.2
15	23.1	33.2

Wet Weather:

Water depth (m)	Temperature (^{o}C)	Salinity (ppt)
0	30.9	28.6
7.5	28.1	29.6
15	26.4	31.6

You are asked to decide a suitable outfall location to ensure the effluent discharge will meet the water quality standards. As a quick estimation, you should ensure the near field mixing zone will not affect the nearby sensitive receivers (Fish culture zones and beaches) to a standard as stated in the above tables.

Examples for both the dry weather and wet weather are done for you, which are roughly the designed outfall parameters.

Scenario 1 – Dry weather

Dry weather flow = 22,000 m^{3}/d = 0.255 m^{3}/s

Outfall location:
EASTING (X) : 846523.9
NORTHING (Y): 825846.5
Outfall depth (Z): 15 m

Outfall parameters:
Total Flow: 0.255 m^{3}/s
Temperature: 26 ^{o}C
Salinity: 1 ppt
Concentration: 1 relative unit
Length: 90 m

Diameter: 1 m
Distance: -
Orientation: -60°

Jet parameters:
Number of jets: 18
Diameter: 0.15 m
Vertical angle: 90°
Horizontal angle: 0°
Distance: 5.3 m (evenly distributed)

General Notes:
1. *Create firstly a scenario* in Port Shelter as instructed in tutorial 1, choosing a desired file location path.

2. *Input ambient parameters*: Input the dry weather ambient parameters given above as instructed in the previous tutorials. The current direction is given as 30°. Current speed assumed to be 0.01 m/s.

3. *Choose the outfall location*:
(i) As an example, the coordinates of the outfall have been given to you.
(ii) enter 846523.9 in X and 825846.5 in Y in the position window.
(iii) enter the outfall depth 15 m in Z in the position window.

4. *Input outfall parameters*:
(i) highlight Outfall#01, enter the outfall parameters as given above.
(ii) The horizontal outfall orientation is given as -60°.

5. *Create jets at the diffuser*:
(i) Highlight Outfall#01, click the **"New Jet"** button.
(ii) Enter the given jet parameters.
(iii) Highlight Jet #01, click **"New Jet"** , the new jet (Jet $02) will automatically be created with the parameters entered as Jet#01. Create Jet #03 – Jet #18 in similar way.
(iv) Highlight Outfall#01, click the **"Evenly Distribute Jets"** button, these jets will then be distributed evenly along the diffuser with spacing calculated.

6. Check all the information entered, and run the model by clicking **"Start Near-Field VISJET simulation"** at the   toolbar.

Mixing zone analysis
7. Observe the evolution and spread of the jet group. The mixing zone can be visualized. In this case, the mixing zone is small enough that nearby sensitive receivers are not affected. The remaining work is to ensure the effluent meets the water quality standards stated in the problem definition in the water quality control zone.

Calculation of water quality parameters at end of near field
8. Study the dilution of the jet group
(i) In this case, the jets are not interacting with each other (as can be visualized), the dilution attained can be treated as the dilution attained by each individual jet.
(ii) Open the **"SUSPEND FILE"** and obtain the surface dilution.
(iii) In this case the average dilution at the surface layer is S=99.29 at a level z=12.57 m above the discharge port.

9. Calculate the concentration of each quantity
(i) Note that for this time, different from tutorial 2, there is a different background concentration for each quantity.

10. Example calculations:

(i) BOD_5 –
Background: 0.8 mg/l
Standard: 5 mg/l
Effluent: 40 mg/l

BOD_5 after dilution (BOD_{5d})
$S=(40-0.8)/(BOD_{5d}-0.8) = 99.29$
$BOD_{5d} = 1.19$ mg/l < 5 mg/l (OK)

(ii) Suspended solid (SS) –
Background: 2.1 mg/l
Standard: $2.1(1+0.3) =$ 2.73 mg/l
Effluent: 60 mg/l

SS after dilution (SS_d)
$S=(60 - 2.1)/(SS_d - 2.1) = 99.29$
$SS_d = 2.68$ mg/l < 2.73 mg/l (OK but marginal)

(iii) Unionized Ammonia (UA)
Background: 0.001 mg/l
Standard: 0.021 mg/l
Effluent: 0.9 mg/l

UA after dilution (UA_d)
$S = (0.9 - 0.001)/(UA_d - 0.001) = 99.29$
$UA_d = 0.010$ mg/l < 0.021 mg/l (OK)

(iv) *E. coli* (EC)
Background: 1 counts/100 ml
Standard: 180 counts/100 ml
Effluent: 1500 counts/100ml

EC after dilution (EC_d)
$S = (1500 - 1)/(EC_d - 1) = 99.29$
$EC_d = 16$ counts/ 100 ml < 180 counts/100 ml (OK)

Summary:

	Effluent	Standard	After dilution	OK ?
BOD_5 (mg/l)	40	5	1.19	Yes
Total suspended solid (mg/l)	60	2.73	2.68	Yes
Unionized Ammonia (mg/l)	10	0.021	0.010	Yes
E. coli (counts/100 ml)	1500	180	16	Yes

11. Try out other current speed and outfall designs to study their effect on the mixing zone and dilution

Scenario 2 – Wet weather
12. This example is similar to Scenario 1 except the discharge and the ambient stratification is changed. Change the ambient parameters and outfall flow ($Q = 0.331$ m^3/s) to the wet weather conditions as given above.

13. Simulate the discharge again by clicking **"Start Near-Field VISJET simulation"**.

14. Observe the jet group evolution and spread. Due to ambient stratification, jets are trapped under the water surface. The jets are not interacting at this current speed and spacing.

15. Obtain the dilution by viewing the **"SUSPEND FILE".** This time the dilution is found to be S=44.0 at a level of z=9.71 m above the discharge port.

Mixing zone analysis
16. The mixing zone is again small and should not affect the nearby sensitive receivers.

Calculation of water quality parameters at end of initial dilution zone

17. Similar to the calculation in the dry weather, the near field water quality for wet weather is tabulated:

	Effluent	Standard	After dilution	OK ?
BOD$_5$ (mg/l)	40	5	1.69	Yes
Total suspended solid (mg/l)	60	2.73	3.42	No
Unionized Nitrogen (mg/l)	0.9	0.021	0.021	Marginal
E. coli (counts/100 ml)	1500	180	35	Yes

18. Try out other current speed and outfall designs to study their effect on the mixing zone and dilution.

Tutorial 4 - Discharge of rosette jet groups from multiple risers : the HATS Outfall

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 ocean outfalls for the Hong Kong Harbour Area Treatment Scheme (HATS), as well as the Shanghai Sewage Project Outfall, are examples of outfalls of this type. This tutorial shows how VISJET3 can be used to simulate the HATS discharge in which multiple rosette risers are used.   The input parameters are as follows:

Ambient Parameters:

Depth below surface (m)	Temperature (°C)	Salinity (ppt)
0	27.7	25.5
5.5	27.8	31.2
11	25.0	33.7
16.5	23.6	34.2
23	23.0	34.5

Current velocity   0.05 m/s (Current Angle = 90º)

Outfall Parameters:
Depth   24.0 m
Length   1200 m
Temperature   25 ºC
Salinity   10 ppt
Flow   19 m^3/s
Orientation   0º
Width   4 m
Concentration 1

Riser Parameters:
No. of risers   24
Riser separation   52 m
Top radius   1.3 m
Bottom radius   2 m
Height   1.5 m

	jet 1	Jet 2	jet 3	jet 4	jet 5	jet 6	jet 7	jet 8
Flow (m³/s)	0.09896	0.09896	0.09896	0.09896	0.09896	0.09896	0.09896	0.09896
Diameter	0.25	0.25	0.25	0.25	0.25	0.25	0.25	0.25

(m)								
Port height (m)	1.5	1.5	1.5	1.5	1.5	1.5	1.5	1.5
Vertical angle	0º	0º	0º	0º	0º	0º	0º	0º
Horizontal angle	0º	45º	90º	135º	180º	225º	270º	315º
Salinity (ppt)	10	10	10	10	10	10	10	10

General Notes:
1. The given outfall configuration is close to that of the operating outfall near the Stonecutters Island being used in the HATS. For a more realistic simulation, you may wish to create *a scenario in Stonecutters Island*. The way of creating a scenario in Stonecutters Island are the same as instructed in the tutorial 1 except that the location is changed to Stonecutters Island.
2. *Input ambient parameters*: Input the ambient parameters, including the current velocity given, as instructed in other tutorials.
3. At this stage it is assumed that readers have gained familiarity with various functions of the VISJET 3 system, certain detailed steps will be omitted, instead we will outline a sensible sequence of an outfall construction in the system which users can take it as reference. The sequence by no means need to be strictly followed but it is considered to be an efficient way for constructing a realistic outfall scenario based on our own experiences.
4. The construction sequence can be thought to be a "Top-down approach", in which the outfall parameters are considered first, then to the riser, and finally to the jet parameters.
5. For example, in this scenario the outfall parameters can first be entered (after clicking "**Create Outfall with Risers**"), including the length, depth, and flow, etc.. We do not need to care about the riser or jet parameters yet. Choose a suitable location by entering directly the outfall coordinates or by using the "**Pick**" function.
6. Next, click the "**New Riser**" button, and enter various given riser parameters. Note that at this moment we need to care about ONE riser only as you will see later one (distance can be ignored at present).
7. Next, click the "**New Jet**" button, enter various given jet parameters, such as diameter, port height, and vertical angle (horizontal angle can be ignored as it will be calculated automatically in this case). Repeat clicking the "**New Jet**" button until Jet #08 appears. Highlight Riser #01 and click "**Evenly Distribute Jets**". Jets are then distributed evenly on the riser circumference from angle 0° of Jet #01 to 315° of Jet #08.
8. You may have noticed that there are totally 24 risers installed on the outfall. To create all of these risers, simply hightlight Riser #01 and

click "**New Riser**" for an extra of 23 times until Riser #24 appears, then 24 identical risers will be created.

9. The last step would be entering the suitable riser distance; for evenly distributed risers simply highlight Outfall #01 and click "**Evenly Distribute Risers**". The 24 risers will then automatically be distributed evenly along the riser with a riser distance of 52.2.

Effect of buoyancy
10. Click **"Start Near-Field VISJET simulation"** 🔊 . Observe the evolution of the jet groups. Jets can be seen to be discharged at different angles merging and interacting with each other within the same riser. Not much bending by the current is observed. After reaching the trapping layer the computation continues and stops after several oscillations.
11. Open the suspend file and obtain different jet mixing characteristics. The buoyancy length scale l_b = 143.6 m, the water depth of the port is 22.5 m. Since $l_b/z \gg 1$, the strong buoyancy effect dominates the jet mixing characteristics in this case, thus jet groups are only slightly advected by the ambient crossflow.

Effect of ambient crossflow
12. Now change the ambient current to 0.2 m/s, and re-run the simulation.
13. Observe the jet evolution again, notice the difference between this case and the previous case. Jet groups can be seen to be strongly bent by the ambient current within a short distance from the discharge point. The jet groups are trapped under the water surface. Computations stop after several oscillations of the jets in the trapping layer.
14. Open the suspend file and obtain different jet mixing characteristics. The buoyancy length scale l_b = 2.24 m, the water depth of the port is 22.5 m. Since $l_b/z \ll 1$, ambient current dominates the flow, and the jets are observed to be strongly bent by the ambient.

Jet interactions
15. For the case with ambient current = 0.2 m/s, change the angle of view until the cross-section of the jet group can be seen. Re-run the simulation and observe the interaction (overlapping) among jets. Do the jets between adjacent risers merge? At a riser separation of 52 m (which is quite large), only jet interaction within the same riser can be observed, there is no interaction between jets of neighbouring risers.
16. As can be seen in the simulation, jets from risers which are separate far apart do not merge with each other even after reaching the trapping layer. The *line plume approximation* then, strictly speaking, is not valid in this case.
17. Try reduce the riser spacing by halving the outfall length, and distribute evenly the risers on outfall. Re-run the simulation again. This time do jets from neighbouring risers interact?

18. When risers are places close enough, interaction between jets from neighbouring risers can be observed. When risers are close enough such that jets merge and form a uniform two-dimensional plume layer, the *line plume approximation* may then be a reasonable approximation

Tutorial 5 - Round Buoyant Jet in a Confined Region - Dynamic Coupling of Jet and Surrounding Fluid

The previous tutorials illustrate the ability of the VISJET 3 system in predicting mixing characteristics (e.g. dilution, trajectory, and mixing zone) of any 3D multiple jet groups in arbitrary ambient conditions in the near field. Computations of the near field model cease once the simulated jets reach the trapping level or hit a boundary (water surface or sea bed). In reality, the pollutant cloud would continue to spread due to the remaining density difference with the surrounding ambient water (gravitational spreading); the dilution process would continue but at a much slower rate. This zone is given the name intermediate field. Further away from the intermediate field is the far field, where the mixing and dilution are governed by the ambient current, passive turbulent dispersion and biochemical processes such as bacterial decay. The DESA technology has been developed to dynamically coupling the near and far field models to enable seamless simulation of the dilution processes from the near field to the far field region.

In the following tutorial cases, the functions and uniqueness of the VISJET 3 system employing the DESA technology will be illustrated.
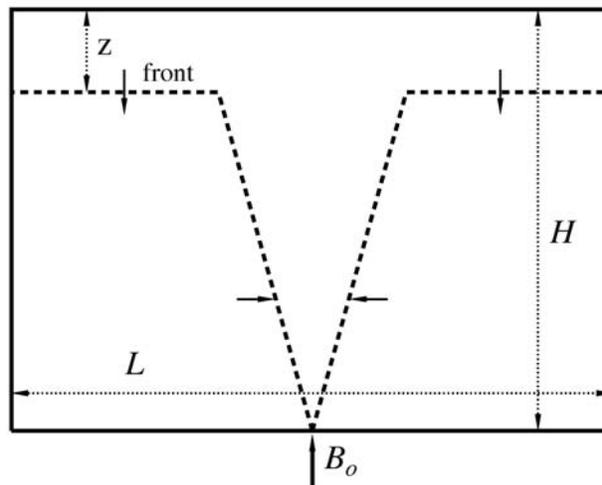
The effect of continuous convection from a source of buoyancy on the properties of the environment in a confined basin (e.g. typhoon shelter) is studied. A round buoyant jet is situated at the centre of a tank 57.2 m x 48.4 m in plane and 30 m deep. Different kinematic buoyancy flux $B_o = \dfrac{Q_o g \Delta \rho_o}{\rho_a(0)}$, where Q_o = discharge volume flux, can be tested. The

buoyant plume mixes with the ambient fluid by turbulent entrainment; when it rises to the water surface the mixed fluid spreads sideways as a surface layer, with negligible vertical mixing (damped by density difference). The thickness of this layer of mixed fluid increases with time, and the front of this surface layer advances with a velocity dz/dt. A 13 x 11 x 10 model grid with horizontal grid size 4.4 m by 4.4 m and 10 uniform vertical layers is used to simulate the round buoyant jet.

The main parameters are as follows:

Basic Data:

Outfall Parameters:
Depth (z)   30 m
Flow   0.046 m³/s
Temperature   20 °C
Salinity   0 ppt
Concentration   100 relative units

Initial Ambient Parameters:
Depth   30 m
Temperature   20 °C
Salinity   40 ppt
Current velocity   0.0 m/s (Current angle   90°)

An initial relative density difference $\dfrac{\Delta \rho_o}{\rho_a(0)} = 0.030$

Jet Parameters:
Diameter   0.3 m
Vertical angle   90°
Horizontal angle   0°

General Notes:

1. *Opening the exiting scenario*
(i) Click **File** at the menu.
(ii) To open an exiting scenario, select **Open**.
(iii) Select the directory roundjet01.
(iv) Select the file roundjet01.vj3.

A view of the basin with the jet at the centre from the top will appear in the right-hand side visualization panel, while the initial ambient conditions and the outfall parameters are shown in the panel in the left-hand side. Click on Outfall #01, the Outfall Parameters will be displayed. Click on Jet #01, the Jet Parameters will be shown.

2. *Navigate around the basin*
(i) Left click any point on the screen, the clicked point will automatically become the centre point of the screen.
(ii) Scrolling down and up will allow you to zoom in and out with the selected point in step (i) as centre
(iii) left click and hold the mouse button, and freely move the mouse to view the topology at different orientations

3. *Near field simulation*
(i) As the previous tutorial cases, the near field simulation will be performed by clicking "**Start Near Field VISJET simulation**" at the toolbar.
(ii) The near field *jet characteristics* of each Lagrangian element can be examined in the same manner as: the first four tutorial cases.

4. *Coupled near and far field simulation*
(i) The coupled near and far field simulation with the DESA technology will be performed by clicking "**Start Near-and-Far Field simulation**" at the toolbar.
(ii) A pop-up window with the warning message "DESA simulation usually takes hours to finish. Confirm to begin simulation now" will appear. Click "Yes" to proceed.
(iii) A progress meter coloured in green and the estimated time to completion will appear to the right of . For the present cases, it will take about 5 minutes to complete the simulation in a machine with 1.86 GHz Xeon processor.

4. *Visualize the near and far field simulation results*
(i) After the completion of the simulation, click "**Far-field visualization**" at the toolbar to launch a separated program Visjet Visualizer for visualizing the far field results.
(ii) Again a view of the basin with the jet from the top will appear initially. A vertical clip plane running across the discharge point is also defined.
(iii) You can navigate around the basin in the same manner as described above.
(iv) Press space bar or select the "Play / Pause" option under the "Control" menu to begin the modelling output display.
(v) A black line is included to indicate the 1:500 dilution contour line.

(vi) The development of the surface layer and the downward movement of the front are clearly visualized.

(vii) Left click at a point in the water surface, a pop-up window will appear showing the time variations of the depth-averaged concentration, water level and current speed.

(viii) The changing in colour in the upper part of the jet can also be observed, which indicates the interaction between the jet and the ambient environment. For the present case, the relative concentration at the top of the jet will increase from about 0.46 to 1.20 (or reduce in averaged dilution from 217 to 84) at the end of the simulation.

5. *Velocity variation at the surface layer*
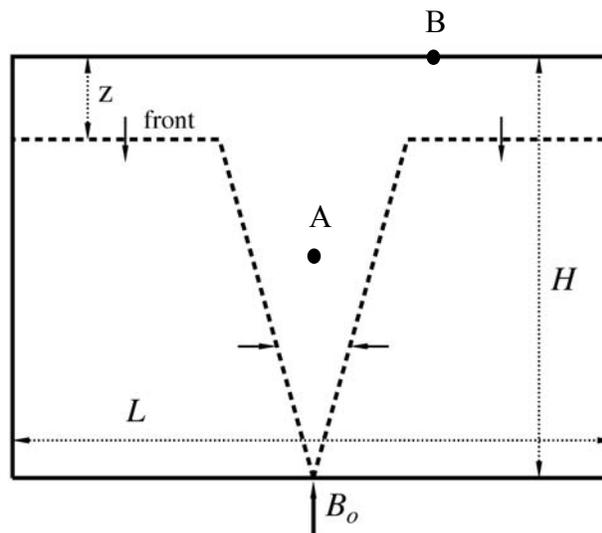
(i) Click a point at the top of the jet (point B in the above figure).

(ii) Play the simulation starting from the beginning, and observe the velocity variation with the time. (In order to provide a better visualization effect, the animation rate is slowed down by scaling the display time; one day showed in the clock represents 30 minutes in the simulation and the simulation is for a duration of 150 minutes instead of 5 days as shown in the digital clock.)

(iii) At the beginning, the velocity at point B is 0.0 m/s as the jet induced velocity is negligible at B at the beginning. At about Day 1, 02:00 a.m., velocity at B is about 0.001 m/s, then increases steadily with time to a maximum of about 0.060 m/s at Day 2, 02:00 p.m., and starts to fall back to about 0.036 m/s which remains approximately constant in the rest of the time. (From the near-field simulation, it can be seen the vertical velocity at point A is about 0.22 m/s.)

(iv) The variation of this velocity shows that there is a continuous variation in the ambient condition with time. This varied ambient condition affects the mixing characteristics (such as concentration, and velocity) of the jet all the time. By using the DESA, this has been fully

taken into account. The change in jet mixing characteristics can be visualized as well by looking closely to the variation of the jet colour (which represents the jet concentration), and the jet movement due to the ambient condition variation.

6. *Concentration variation at the surface layer in the horizontal direction*
(i) Consider again point B in the same figure. This time we will be more focused on its concentration variation.
(ii) Play the visualization again. The concentration at point B can be seen to increase steadily from 0 unit to 0.5 unit at the end of simulation.
(iii) Now shift the monitoring point slightly to a point C further away from the centre as shown in the figure, and play the visualization again.
(iv) Observe the variation of the concentration at point C with time.
(v) Except a slight observed time shift, the concentration variation at point C is similar to that of point B, with concentration 0.5 unit at the end of the simulation. This shows that the horizontal variation in the surface layer is small.
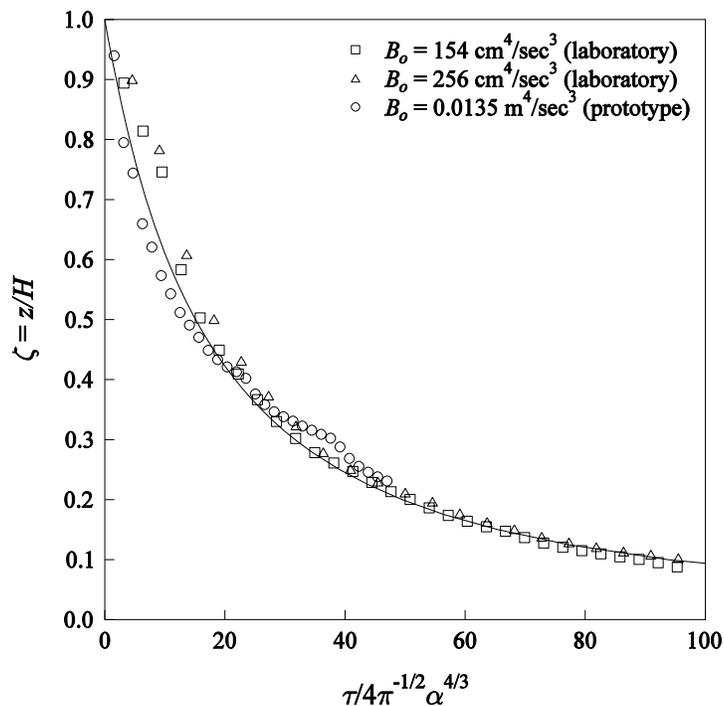
7. *Variation of the interface with time*
(i) It can be seen from the visualization that the interface (defined by a certain concentration contour) of the surface layer keeps moving downwards.
(ii) Baines and Turner (1969) performed laboratory experiments and obtained an analytical solution which describes the variation of the interface defined by the $0.1\Delta\rho_{max}$ contour with time, and is given below:

$$\tau = 5\left(\frac{5}{18}\right)^{1/3}[\zeta^{-2/3} - 1]$$

in which $\zeta = \dfrac{z}{H}$ and $\tau = \dfrac{4}{\pi^{4/3}}\alpha^{4/3}\left(\dfrac{H}{R}\right)^2\dfrac{(B_o/2)^{1/3}}{H^{4/3}}t$ with $\alpha =$ entrainment constant $= 0.1$, and R is the equivalent radius for the rectangular tank.
(iii) Try to obtain the interface position from the visualizer at a few time instants according to the definition given above. Compare with the analytical solution given above. Results are represented by circle in the figure below.
(iv) The figure below also shows the comparison between the result of the calculations simulating the experiments carried out by Baines and Turner with the laboratory scale (using a 19 x 15 x 20 model grid with horizontal grid size 3 cm by 3 cm and 20 uniform vertical layers) and the analytical solution. The predicted interface position by the DESA is in excellent agreement with the analytical solution deduced from the experimental results.

Legend (inside plot):
$B_o = 154$ cm^4/sec^3 (laboratory)
$B_o = 256$ cm^4/sec^3 (laboratory)
$B_o = 0.0135$ m^4/sec^3 (prototype)

Y-axis: $\zeta = z/H$

X-axis: $\tau/4\pi^{-1/2}\alpha^{4/3}$

8. *Jet with greater discharge flow*

(i) To see the effect of greater discharge flow, click on Outfall #01 and change the total flow to 0.092 m^3/s (three times the original flow).

(ii) Click "**Save Project**" 🖫 to save the updated information to the file.

(iii) Perform the simulation and visualize the results in the same way as described about.

(iv) A higher concentrations at the jet and the surface indicate smaller dilutions can be observed. The initial relative concentration at the top of the jet increases from about 0.46 to 0.70, i.e. the averaged dilution decreases from 217 to 143. (The similar results can be obtained from the near field simulations).

9. *Jet with weaker buoyancy*

(i) To see the effect of a discharge with weaker buoyancy, click on Outfall #01 and change the salinity to 35 ppt (an initial relative density difference equals to 0.004).

(ii) Click "**Save Project**" 🖫 to save the updated information to the file again.

(iii) Perform the simulation and visualize the results in the same way as described about.

(iv) It can be observed that an even higher concentrations are resulted indicate a further reduction in dilutions. The initial relative concentration at the top of the jet further increases to about 1.23, i.e. the averaged dilution reduces to 82. (The similar results can be obtained from the near field simulations).

Tutorial 6 - Environmental Impact Assessment for Sai Kung Sewage Treatment Works (SKSTW) Outfall

In many densely populated cities in Asia, such as Hong Kong, jet groups discharged from an outfall is usually located not far away from water sensitive receivers (beaches or fish farms). While relying on the near-field model alone can give one a quick impression of the outfall performance close to the source, a complete computation from near to far field is necessary when conducting the environmental impact assessment.

The purpose of this example is to evaluate the potential environmental impact of the operation of the Sai Kung Sewage Treatment Works (SKSTW) under the design capacity of 22,000 m3/day. The SKSTW dry-weather influent flow pattern observed in mid-December 2001 was adopted to derive the 24-hour effluent flow pattern. The outfall for SKSTW is assumed to be a 90 m long diffuser with 18 vertical jets of diameter 0.1 m. Pollution loads from 12 storm outfalls and Sai Kung Typhoon Shelter are also included. The effluent discharge flow is taken to be fresh water with zero salinity.
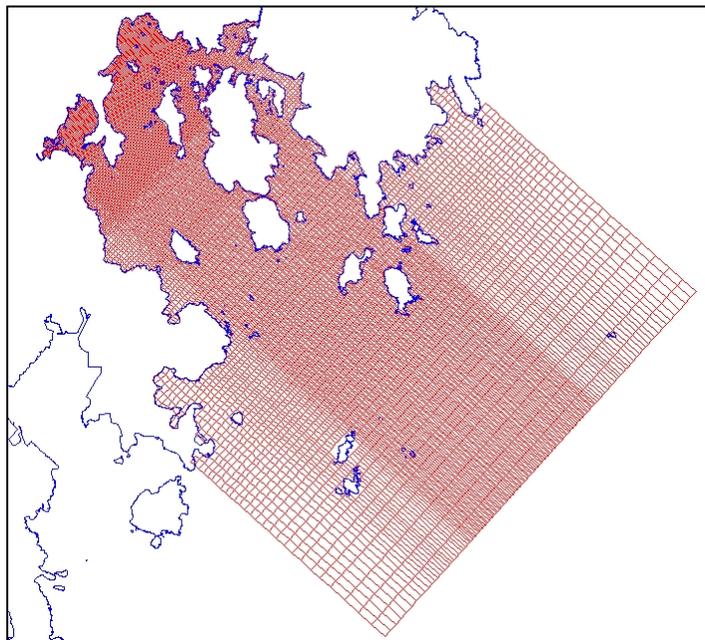
The far field model uses a 153 x 153 x 8 curvilinear grid and has 100192 active cells. The tidal forcing at the open boundary consists of the five

main tidal constituents (M2, S2, O1, P1 and K1) and 15-day spring-neap cycle is simulated. With DESA, the impact of the effluent discharge from the SKSTW outfall in both the near and far field is simulated.

The files for this tutorial are "PS06_scan07b9.vjx", "PS06_scan02b9.vjx" and PS07w_nh301.vjx".

Problem definition

Example sensitive receivers near the Sai Kung Sewage Treatment Works include :

Kiu Tsui beach (1.6 km from SKSTW)
Trio beach (1.9 km from SKSTW)
Ma Nam Wat (1.8 km from SKSTW)

You are asked to investigate whether the pollutant concentration (e.g. *E. coli* and un-ionized ammonia) will exceed the water quality objective at the above sensitive receivers, under the following land-based sewage treatment scenarios:
 (a) No sewage treatment before discharge
 (b) Only primary treatment

Water Quality Objectives for Port Shelter Water Control Zone (WCZ)

Parameters	Objectives	Sub-zone
E. coli	Not to exceed 610 counts/100 ml, calculated as the geometric mean of all samples collected in one calendar year	Secondary Contact Recreation Subzone and Fish Culture Subzones
	Not to exceed 180 counts/100 ml, calculated as the geometric mean of all samples collected in one calendar year	Bathing Beach Subzones
Ammonia	Annual mean not to exceed 0.021 mg/L as unionized form	Whole zone

General Notes:

1. To study the GIS application.
   (i) Load "PS06_scan07b9_set". This scenario simulates the case of no sewage treatment and the effluent *E. coli* concentration is 10^7 counts / 100 mL.
   (ii) Zoom in to Sai Kung District until names of beaches and marine water quality monitoring stations can be seen.
   (iii) Left click a marine water quality monitoring station (e.g. PM3). Past data of water quality will be displayed. Water quality data at surface, middle and bottom layer are available at a monthly/bi-monthly interval.
   (iv) Left click Trio Beach. Water quality data, including water temperature, DO, DO saturation, salinity, pH, turbidity and *E. coli* concentration, are available. Click the '+' button next to the parameter to display the data time series from 1986 to 2007. Data interval ranges from 3 to 14 days.
   (v) By looking at the time series of *E. coli* concentration, you can observe significant *E. coli* reduction since 2000. The high bacteria concentration in 90's is possibly caused by the contribution of polluted fresh water source from the Ho Chung River (west of Trio beach) and Sai Kung.
   (vi) Relationship between *E. coli* concentration and environmental parameters can also be observed. Left click and drag on the *E. coli* time series from 1992 to 1993, the selected area will be enlarged. Observe the sudden increase in *E. coli* concentration with the corresponding salinity reduction. The immediate surge in *E. coli* due to salinity change suggests the impact from fresh water source (i.e. Ho Chung River and stormwater outfalls).
   (vii) Left click Kiu Tsui Beach. Observe the time series of *E. coli* concentration, and note the difference between Trio and Kiu Tsui. Although the two beaches are close to each other, Kiu Tsui is facing more open oceanic water without major freshwater source. The consistently lower *E. coli* concentration at Kiu Tsui indicates the importance of beach orientation, existence of a nearby freshwater source etc., towards beach water quality.

2. To display the model output (non-conservative pollutants, *E. coli*)
   (i) Press space bar to begin the modeling output display.

(ii)   Click 'View' and select 'Current Flow'. Arrows with length and direction representing current speed and current direction respectively will be displayed.

(iii)   Left click any point in the map to display the time series of *E. coli* concentration, tide level and current speed.

(iv)   Observe if the current speed in Port Shelter lies between 0 to 0.05 m/s.

(v)   Click 'View' and select 'Jets'. Sewage outfall with 18 jets will be displayed at grid [846523.9 825846.5].

(vi)   These 18 jets are computed from the Lagrangian model JETLAG, and dynamically coupled with the 3D far field circulation model through the distributed entrainment sink approach (DESA) at grid cell level.

(vii)   At each time step, ambient information from the 3D circulation model is fed back into the near field model. Using the updated ambient data, the near field jet characteristics are then re-computed using JETLAG. The updated jet trajectories and the action of the jet entrainment on the far field are then computed for the next time step. This process is repeated for every time step. When close up the jet trajectories can be seen to move back and fro with the tidal current direction; in this case the action of the jets on the ambient current is small since the current speed is small (see (iv)).

(viii) Observe how the current changes with the tide again. The current is relatively constant over a time scale in the order of an hour. Since the jet mixing process has a time scale of minutes, it is justified to assume a quasi-steady flow for computing near field jet mixing (this is not relevant to the present simulation but is expressed as a point of note).

(ix)   This dynamic coupling of the near and far field is necessary in Hong Kong as it can be seen sensitive receivers are not far away from the source. This is also the uniqueness of the VISJET 3 system.

(x)   The modeling result shows that Kiu Tsui beach is the only sensitive receiver that may be affected by the sewage outfall. Right click Kiu Tsui beach and observe the time series of *E. coli* concentration. The *E. coli* concentration is below 24 count / 100 mL most of the time, reflecting an effective dilution and bacterial decay that reduces the impact to the beach.

(xi)   Click 'View' and select 'clip plane' to observe the vertical profile of *E. coli* concentration.

(xii) Other sensitive receivers, Trio and Ma Nam Wat, are not directly influenced by the SKSTW and thus are not heavily polluted in this scenario.

(xiii) Besides the main source contributed by the sewage outfall, locate the 12 storm outfalls distributed along the shoreline of Sai Kung. Investigate their area of influence.

(xiv) Load "PS06_scan02b9". This scenario models the sewage outfall with primary treatment and the effluent *E. coli* concentration is 1500 counts/ 100 mL. A clear improvement can be observed in this scenario when compared to scenario "PS06_scan07b9". The impact from the sewage outfall nearly fades out before reaching Kiu Tsui beach.

3. To display the modeling output (conservative pollutants, un-ionized ammonia)

(i) Load "PS07w_nh301". This scenario simulates the case when a conservative pollutant, un-ionized ammonia, is considered. It represents the operation of SKSTW under the design capacity of 22,000 m^3/day, and the effluent un-ionized ammonia concentration is 0.9 mg/L.

(ii) Observe the difference between non-conservative and conservative pollutants.

(iii) Due to the conservative nature of un-ionized ammonia, the region affected by the discharge increases with time. However, the concentration at most locations is below the 0.0046 mg/L threshold.

(iv) Given that the source concentration of the nutrient is 0.9 mg/L, ambient current near the outfall can be observed to be about 0.04 m/s – 0.05 m/s, outfall depth is 11.5 m, and linear ambient stratification can be assumed with salinity varies from 33.5 ppt on the top to 34 ppt at the bottom. With the given information try run VISJET 2. What is the initial dilution for this case? How does it compare with the VISJET 3 output?

(v) The near field average dilution for this case is computed to be about S=200–300 at the water surface with a near field model. The VISJET 3 results show that the **minimum dilution** at the surface near the outfall is about 200 (~0.05 mg/L). It can be seen that the two models given comparable results; with some additional mixing in the intermediate field (average dilution is typically 1.7-2 times minimum dilution).