Windows Based Drainage Design Software

M C Zhou  (cmczhou@ntu.edu.sg)
Tommy S W Wong (cswwong@ntu.edu.sg)

Introduction

As Singapore is a tropical country with abundant rainfall, in order to minimize the risks of flooding and the associated health hazards, it is important that an effective drainage system be put in place. To this end, the drainage systems in Singapore are required to be designed in accordance with the “Code of Practice on Surface Water Drainage” (Public Utilities Board, 2000). In this Code, the drainage systems are required to satisfy the Rational formula and the Manning equation. As such, designers are often required to carry out a tedious and time consuming iteration procedure. After satisfying the Rational formula and the Manning equation, the design may still require revision as it is also obliged to satisfy other requirements as specified in the Code. With the intention of saving designers much time and effort, a Windows-based drainage design software has been developed. The software is known as the “Drainage Design Software” or “DDSoft”. This article describes the features in the DDSoft.

Features of DDSoft

The DDSoft is written in Visual Basic language, and operates on the Microsoft Windows platform. It comprises three main interfaces: one principal and two supplementary.

In the DDSoft, the input data and the output results are displayed on the same principal interface, namely “Drainage Design Software” as shown in Figure 1. This interface enables the designers to carry out a design easily. At the top left hand corner, the designer can enter a project title. Then he is required to select a design option. Two options are available: (i) channel, (ii) sewer. For each design option, there are three input modes. For the “channel” option, the input modes are: (i) channel bed slope and channel width, (ii) channel bed slope and flow velocity, and (iii) channel width and flow velocity. For the “sewer” option, the input modes are: (i) sewer bed slope, (ii) sewer diameter, and (iii) sewer flow velocity. The designer then selects the rainfall location, either “Singapore” or “others”. If he selects “Singapore”, he can then select a return period from a drop down menu. Alternatively, he can enter the return period directly. On the other hand, if he selects “others”, the designer is required to input the rainfall intensity-duration equation of that location.

After entering the information on the rainfall, the designer is required to enter the information on the catchment. From a drop down menu, the designer may select a single development type. For this case, the value of the runoff coefficient according to the Code will appear automatically. Alternatively, the designer may select the composite development type. For this case, as shown in Figure 2, a supplementary interface known as the “Composite Development Catchment” will appear. The designer may select the appropriate development type in terms of area in ha, or in %. When the designer clicks the button “Apply”, DDSoft will work out the weighted runoff coefficient. If the designer is satisfied with his input, he can click “OK” and DDSoft will return to the principal interface. If the percentage for any development type or the summation of the percentages exceeds 100%, DDSoft will give a warning message. The designer is then required to amend his input. In the “Composite Development Catchment” interface, if the designer enters the area in terms of ha, the total area will be calculated and automatically transferred to the

Figure 1. Principal interface for data input and output results

Figure 2. Supplementary interface for calculating weighted runoff coefficient
principal interface as the catchment area. Otherwise, on the principal interface, the designer is required to enter the catchment area. The designer can specify his own runoff coefficient value if it is different from those described in the Code. Other information on the catchment includes the overland flow time and the upstream channel (or sewer) flow time. Their values can be specified directly by the designer, or computed using the equations provided.

Next, the designer is required to enter the information on the channel or sewer. For the “channel” option, the designer is required to select a channel shape. Four channel shapes are available. They are vertical curb, triangular, rectangular and trapezoidal. For the “sewer” option, one shape is available and it is circular under full flow condition. After selecting the channel shape, the designer is required to select a surface type. From a drop down menu, the designer may select a single surface type. For this case, the value of the Manning’s roughness coefficient according to the Code will appear automatically. Alternatively, the designer may select a composite surface type. For this case, as shown in Figure 3, a supplementary interface known as the “Composite Surface Channel/Sewer” will appear. The designer may select the appropriate surface type in terms of wetted perimeter in m, or in %. When the designer clicks the button “Apply”, DDSoft will work out the equivalent Manning’s roughness coefficient. If the designer is satisfied with his input, he can click “OK” and DDSoft will return to the principal interface. If the percentage for any surface type or the summation of the percentages exceed 100%, DDSoft will give a warning message. The designer is then required to amend his input. If the channel (or sewer) surface type is different from those described in the Code, the designer can specify his own value directly. The remaining input data on the channel (or sewer) depends on the selected design option and input mode, and the channel shape. The requested data will be a combination of channel length, channel width (or sewer diameter), channel bed slope (or sewer bed slope), flow velocity, and channel side slope. DDSoft catered for channels with unequal side slopes.

With the input data, DDSoft will perform the design calculations as soon as the designer clicks the button “Run”. The output results are displayed on the principal interface. The output results include time of concentration, design rainfall intensity, design discharge. The remaining output results depend on the selected design option and input mode. They can be the channel flow time, flow velocity (or sewer flow velocity), Froude number, channel width (or sewer diameter), or channel bed slope (or sewer bed slope). For the output results, DDSoft will perform an automatic check to see if they satisfy the other requirements in the Code (e.g. minimum and maximum design velocities, and maximum Froude number). If there is any result that does not satisfy the requirements, DDSoft will display a warning message, and give a suggestion on how to revise the design.

For certain sets of input data, there are no possible solutions for flow depth or channel width. For these cases, DDSoft will also display a warning message and a suggestion on how to revise the design. For certain sets of input data, DDSoft will work out the equivalent Manning’s roughness coefficient according to the Code. DDSoft will also display a warning message and a suggestion on how to revise the design. The input data and the output results can be printed out as reports or saved as ASCII files. The latter output option allows earlier designs to be retrieved easily.

**Conclusion**

The “Drainage Design Software” or “DDSoft” is a Windows-based software. It is able to perform the drainage design calculations according to the “Code of Practice on Surface Water Drainage”. In the DDSoft, the input data and the output results are displayed on the same principal interface. This enables the designers to carry out a design easily. Further, for certain sets of input data, if the output results do not satisfy the requirements in the Code, DDSoft will display a warning message with suggestions on how to revise the design. The input data and the output results can also be printed out as reports or saved as ASCII files. The latter output option allows earlier designs to be retrieved easily. DDSoft will save the designers a lot of time and effort in carrying out a drainage design. DDSoft is available on the web site: http://www.ntu.edu.sg/home/cswwong/software.htm.

**Reference**

Influence of Tides on a Coastal Aquifer in Singapore

Mzila N (PZE348703@ntu.edu.sg)
Shuy E B (cshuyeb@ntu.edu.sg)

Introduction

The periodic rise and fall of tides in the open sea produce sinusoidal fluctuations in the groundwater level in an adjacent hydraulically connected coastal aquifer. In an unconfined aquifer, the fluctuations occur as pressure waves generated by changes in storage due to dewatering and re-saturation of pores propagate inland. The characteristics of these pressure waves, i.e. their velocity, amplitude, wavelength and attenuation, are a function of the tidal period and amplitude, the aquifer’s transmissivity and storage coefficient, and the distance inland.

The present study was conducted in the shaded areas (A4 and A2) of the Changi reclaimed land as shown in Figure 1 in local co-ordinates. The fluctuations in groundwater level at the site caused by tidal influence were measured using a TROLL 8000 pressure transducer at observation wells A2S10, A4S17, A4S08, A4S20, A2S09 and A2S08, shown in Figure 2. An isometric section of the study area is shown in Figure 3. The aquifer is composed of artificial sand fill, with mean grain size 0.75 mm, porosity n = 40, d_{10} = 0.12 to 0.37 mm, shell content = 6 to 9%, uniformity coefficient = 2.5 to 4, and extends from the ground surface to a depth of between 12 and 16m.

Relation between tide level and piezometric level

The observed groundwater level fluctuations can be divided into two components: (1) tidally induced fluctuations, and (2) fluctuations caused by other factors, and can be described by the following equation (British Geological Survey, 2001):

\[
\frac{dh'(t)}{dt} = \frac{dh(t)}{dt} - E_{side} \frac{dH(t-t_{lag})}{dt}
\]  

(1)

where:
- \( h' \) = groundwater elevation without tidal influence (L)
- \( h \) = observed groundwater elevation (L)
- \( H \) = tidal elevation on surface water body (L)
- \( E_{side} \) = tidal efficiency (dimensionless)
- \( T \) = the time when groundwater elevation was measured (T)
- \( t_{lag} \) = time lag between tidal effects in surface water body and corresponding effects at groundwater observation points (T).

The relationship between tidally induced piezometric levels and tidal fluctuations is determined by two parameters: tidal efficiency (\( E_{side} \)), and time lag (\( t_{lag} \)). The tidal efficiency factor is the ratio of amplitudes of the piezometer readings and tidal readings. The time lag is the time difference between the tidal levels and the corresponding groundwater levels.

In a confined, homogenous and isotropic aquifer, sinusoidal oscillations in pressure will propagate along the aquifer according to the following equation (Erskine, 1991):

\[
h = h_i \exp(-x \sqrt{\pi \tau / t_i}) \sin(2\pi / t_i - x \sqrt{\pi \tau / t_i} T)
\]  

(2)

where \( h \) = groundwater head relative to mean sea level (m);
- \( x \) = distance from the sea (m);
- \( t_i \) = time (d);
- \( t_o \) = period of tidal oscillation (d);
- \( h_i \) = amplitude of tidal oscillation (m);
- \( T \) = transmissivity of aquifer (m²/d); and \( S \) = storage of aquifer (dimensionless).
Both the tidal efficiency factor and time lag are determined by a number of factors including aquifer hydraulic conductivity and storativity (or diffusivity), aquifer thickness, and distance of the observation piezometer from the sea. The solution of equation 2 shows that the tidal oscillations remain sinusoidal with a time lag and decrease exponentially in amplitude with distance from the sea. The magnitudes of these parameters are as follows:

\[ E_{\text{tide}} = e^{- \frac{\pi S}{t_s K b}} \]  
and

\[ t_{\text{lag}} = \frac{t_s S}{4 \pi K b} \]

where \( K \) = aquifer hydraulic conductivity (L/T) and \( b \) = aquifer thickness (L).

Equations 3 and 4 developed for confined conditions can also be used for unconfined conditions if the range of fluctuation of groundwater levels is small compared to the saturated depth of the aquifer.

**Calculation of tidal efficiency and time lag**

For calculating the tidal efficiency factor and time lag from the observed tidal and tidally induced groundwater piezometer data, an observation period is selected during which other factors affecting groundwater levels such as rainfall are negligible, i.e. \( \frac{dh'}{dt} = 0 \). The groundwater level fluctuations are then solely a function of tidal fluctuations, and equation 1 reduces to:

\[ \frac{dh(t)}{dt} = E_{\text{tide}} \frac{dH(t-t_{\text{lag}})}{dt} \]  

For a time period from \( t_0 \) to \( t_1 \) in the groundwater observation record, the solution of equation 5 can be obtained by integration as follows:

\[ \int_{t_0}^{t_1} \frac{dh(t)}{dt} dt = \int_{t_0}^{t_1} E_{\text{tide}} \frac{dH(t-t_{\text{lag}})}{dt} dt \]

The above integral can be expressed as follows:

\[ h(t_1) - h(t_0) = E_{\text{tide}} \left[ H(t_1 - t_{\text{lag}}) - H(t_0 - t_{\text{lag}}) \right] \]

From equation 7, the tidal efficiency for the period from \( t_0 \) to \( t_1 \) can be calculated as follows:

\[ E_{\text{tide}} = \frac{h(t_1) - h(t_0)}{H(t_1 - t_{\text{lag}}) - H(t_0 - t_{\text{lag}})} \]

High and low tides are selected from tidal records and the corresponding groundwater high and low levels are identified. The time lag is calculated from:

\[ t_{\text{lag}} = t_{\text{tide}} - t_{\text{gw}} \]

where

\[ t_{\text{tide}} \] = Time for the \( i^{th} \) high (or low) tide (T)

\[ t_{\text{gw}} \] = Elevation time for the \( i^{th} \) high (or low) groundwater elevation corresponding to the \( i^{th} \) high (or low) tide (T).

and the tidal efficiency is calculated from:

\[ E_{\text{tide}} = \frac{h_i - h_{i-1}}{H_i - H_{i-1}} \]  

where

\[ H_i \] = \( i^{th} \) high (or low) tidal elevation (L)

\[ h_i \] = The \( i^{th} \) high (or low) groundwater elevation corresponding to the \( i^{th} \) high (or low) tide (T).

The time lag and tidal efficiency factor were calculated for 6 groundwater piezometers at the study site. The time lag was calculated by a least-squares fit method. In order to perform this analysis, the time series of piezometric readings were compared to tidal elevations for the same period and the piezometric series were shifted until the sinusoidal function was synchronous with tidal elevation data. The shift represents the time lag between observed groundwater piezometric data and tidal data. The piezometric readings were then shifted in elevation to have the same mean as the tidal records and then amplified by the tidal efficiency factor for each standpipe. Mathematically, this shift and amplification may be written as:

\[ h'(t) = \bar{H} + \left[ h(t) - \bar{h} \right] E \]

where

\[ h(t) \] = piezometric reading at time \( t \) (m);

\[ h'(t) \] = shifted piezometric level at time \( t \) (m);

\[ \bar{H} \] = mean piezometric level (m);

\[ \bar{T} \] = mean tidal level (m);

\[ E \] = tidal efficiency factor (dimensionless).

The answers were checked visually by overlaying the shifted, amplified and correctly lagged plot with the tidal record, as shown Figure 4.

**Results and discussion**

The time lags and log tidal efficiency factors were plotted against distance from the coastline for the groundwater piezometers used in the study as shown in Figures 5 and 6. From these graphs it can be observed that time lags vary...
Three Dimensional Flow Prediction using POM for Waters off Singapore

Arthur Lim (ctblim@ntu.edu.sg)
Edmond Lo (cymlo@ntu.edu.sg)
Tan Soon Keat (ctansk@ntu.edu.sg)

Introduction

The numerical Princeton Ocean Model (POM) was applied to a 3D flow prediction for waters off Singapore. This was done as part of an ongoing 3-year joint MPA-NTU project to develop and validate POM into a Computational Ocean Modelling System (COMS) for tidal and circulation flow predictions for the South China Sea. The present work reported here forms the initial part of the effort with subsequent work focusing on developing a South China Sea model and a graphical user interface (GUI) platform for COMS.

The POM prediction for waters off Singapore was performed on a small-scale domain stretching from 103.4°E to 104.3°E in longitude and 1°N to 1.4°N in latitude. Excellent predictions were obtained, specifically, the variation of the predicted tidal current magnitude and direction at 3 different locations for 2 different time periods compared very well with MPA field data taken at these locations. Vertical profiles of current characteristics with depth variation on these locations also showed the ability of the POM to model well the physical vertical structures of downwelling and direction reversal induced by bathymetry change.

Brief description of POM

POM is a three-dimensional, time-dependent primitive equation model, first created by Blumberg and Mellor [1] around 1977, and has now been extended and used by over 450 research groups worldwide for applications ranging from estuarine, to coastal and to oceanic bodies.

The principal features of the model incorporates the following:

- A bottom-following sigma coordinate scaling for the vertical dimension to model significant bottom topographical variability.

References


An imbedded second moment turbulence closure model of Mellor and Yamada [2] to provide a realistic parameterization of the vertical mixing process.

A curvilinear orthogonal coordinate and the staggered “Arakawa-C” differencing on the horizontal grid. The horizontal diffusivity coefficients are based on Smagorinsky [3] parameterisation.

A free sea-surface coupled with a split-mode time step. In this treatment, an external 2D depth averaged mode is used, solving first for the sea-surface elevation and depth-averaged variables, and uses a smaller time step based on CFL requirement and the external surface wave speed. The internal mode is 3D and solves for the vertical and horizontal velocities, temperature and salinity through the layers, and uses a longer time step due to implicit time stepping and a CFL condition based on the internal wave speed.

Governing equations

The equation set of POM, consists of continuity, momentum transport, energy or temperature conservation and salinity conservation. These are written, in a physical coordinate system with x increasing eastward, y increasing northward and z increasing vertically upward, and with corresponding velocities (U,V,W), as:

Continuity: \[ \nabla \cdot \vec{V} = 0 \] \quad V = \text{vector} (U, V) \quad (1)

Momentum:

\[ U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} + W \frac{\partial U}{\partial z} = -\frac{1}{\rho_0} \frac{\partial P}{\partial x} + (K_M \frac{\partial U}{\partial z}) + F_x \]

\[ V \frac{\partial U}{\partial x} + U \frac{\partial U}{\partial y} + W \frac{\partial U}{\partial z} = -\frac{1}{\rho_0} \frac{\partial P}{\partial y} + (K_M \frac{\partial U}{\partial z}) + F_y \]

W term (hydrostatic and Boussinesq approx.):

\[ \rho g = \frac{\partial \rho}{\partial z} \] \quad (2a,b,c)

Here \( \rho_0 \) is the reference density, \( \rho \) the in-situ density, \( g \) the gravitational acceleration, \( P \) the pressure, \( K_M \) the vertical eddy diffusivity of momentum mixing, and \( f \) the Coriolis parameter, given in terms of Earth’s latitude by the \( \beta \)-plane approximation, \( 2 \Omega \sin(\phi) \). \( F_x \) and \( F_y \) are terms describing the horizontal mixing processes, and in analogy to molecular diffusion, are written as:

\[ (F_x, F_y) = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}) \left[ 2A_M \left( \frac{\partial U}{\partial x}, \frac{\partial V}{\partial x} \right) + (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}) \left[ A_M \left( \frac{\partial U}{\partial y}, \frac{\partial V}{\partial y} \right) \right] \right] \]

(3)

\( A_M \) is the horizontal diffusivity computed via Smagorinsky’s parameterization based on the local grid scales and deformation field. In present computations, \( A_M \) is generally initialized to be between 2 to 20 m²/s.

Given that the free surface is expressed by \( \eta(x,y,t) \), as depicted below, the pressure at a given depth \( z \) can be obtained by integrating Eqn(2c) over \( z \) to give

\[ P(x, y, z, t) = P_{sm} + \rho \rho_0 \eta + \int_{0}^{z} \rho(x, y, z', t) dz' \]

(4)

where atmospheric pressure \( P_{sm} \) is assumed to be constant.

The temperature (\( \theta \)) and salinity (S) conservation equations are written in similar forms as the \( U, V \) momentum equations. For our present case of Singapore waters, whereby both fields are generally uniform due to the small water depth (averaging approximately 35m), these are not solved, even though they are part of the POM model. In deeper waters with large geographical coverage such as the South China Sea, these terms would influence the pressure and density fields, through the baroclinic approximation where the density is given as \( \rho = \rho(0, S, p) \). This would in turn affect the velocity fields in Eqn (1) and (2) through the density term and pressure gradient terms.

In Equation (2), the vertical mixing coefficient \( K_M \) is based on the second-order turbulence closure model of Mellor and Yamada [2]. It characterizes the turbulence by 2 equations for turbulence kinetic energy, \( q/2 \), and a turbulence macroscale, \( l \). Further details can be found in references [1] and [2]. In particular \( K_M \) is expressed as a function of \( l \) and \( q \).

The formulation in physical \( x,y,z \) coordinates has disadvantages of numerical implementation and accuracy in vicinities of large bathymetric variations. In POM, this is overcome through the use of the sigma coordinate that transforms the sea surface and the bottom into coordinate surfaces as

\[ \sigma = \frac{z - \eta}{H + \eta}, \]

(5)

where \( H(x,y) \) is the bottom bathymetry and \( \eta(x,y,t) \) is the surface elevation. Thus the sea-surface at \( z=\eta \) is mapped to \( \sigma=0 \) and the sea-bed at \( z=-H \) is mapped to \( \sigma=-1 \). After conversion, the velocity component normal to sigma surfaces \( \omega \) is related the Cartesian vertical velocity, \( W \) as:

\[ W = \omega \left[ (\sigma \frac{\partial}{\partial \sigma}) \left( \frac{\partial \eta}{\partial \sigma} \right) + (\sigma \frac{\partial}{\partial \sigma}) \left( \frac{\partial \eta}{\partial \sigma} \right) + (\sigma \frac{\partial}{\partial \sigma}) \left( \frac{\partial \eta}{\partial \sigma} \right) \right] \]

(6)

The POM equations have further been modified to include the equilibrium astronomical tide level at any geographical position and time. These relationships are based on Newtonian theory of tide generation from the gravitational and centrifugal forces of Earth-Moon and Earth-Sun System. This astronomical tide component is built into our code as an additional source of forcing function to account for large astronomical gravitational forces over large geographic surface.

Vertical boundary conditions

The boundary conditions at surface and bed are:

\[ \omega(0) = \omega(-1) = 0 \] \quad (7a,b)

The surface boundary conditions for (3) and (4) are

\[ K_M \frac{\partial U}{\partial \sigma} \frac{\partial V}{\partial \sigma} = - (\langle \omega W \rangle >, \langle \omega W \rangle >), \sigma \to 0 \]

(8a,b)
where the right hand side of Eqn (8a,b) are the input values of the surface turbulence momentum flux or the surface wind stress vector. In our application, the simulation was done without surface wind forcing. The corresponding bottom boundary conditions are

\[ \frac{K_M}{D} \left( \frac{\partial U}{\partial x} \right) = C_z \left[ \frac{1}{2} \left( U^2 + V^2 \right) \right] \sigma, \sigma \rightarrow 1 \]  

(9a,b)

\[ C_z = \text{MAX} \left[ \frac{\kappa^2}{\ln \left[ 1 + \sigma_{k-1} H / z_o \right]}, 0.0025 \right] \]  

(9c)

\( \kappa = 0.4 \) is the von Karman constant and \( z_o \) is the roughness parameter, and is taken as 0.01m for the present simulation.

**Lateral boundary conditions**

On the solid lateral boundaries, i.e. land borders, free-slip condition but zero normal velocity are applied. Along open boundaries where the numerical grid cells end but fluid motion is unrestricted, tidal elevations are prescribed as either harmonic series or time series interpolations as forcing for the surface elevation, \( \eta(t) \). Correspondingly for the velocity fluxes, radiation boundary conditions are used as,

\[ \frac{\partial U_n}{\partial n} \pm c_i \frac{\partial U_n}{\partial t} = 0 \]  

(10)

where ‘n’ refers to direction to the normal to the boundary, and \( c_i \) is the internal wave speed.

In the present work, over each open boundary, a single set of measured time-series tidal elevation data, as measured at location along the boundary, is set as the sole forcing function on that boundary. The numerical boundary treatment for the sea-level inputs has further been modified to accommodate (a) time-series amplitude-time input and (b) harmonic amplitude-phase input from field tidal station. For the Singapore water case, the harmonic component input focused on only 13 significant modes, namely the semi-diurnal modes (M2, S2,N2,K2 Tide), the diurnal modes (K1,O1,P1,Q1 Tide) and several Long-Period Modes and Mixed Modes (MF,MM, SS or SSA, M4 and MS4 Tide). These 13 modes typically constituted higher than 94% of the total spectra energy of tides.

**Singapore water model and results**

Using a combination of two bathymetry maps with resolutions of 500m and 1000m spacing respectively, a model on the Singapore water was created with the open boundary locations and geographical domain as shown in Figure 2. This model consists of 115*85 IJ horizontal grids and 11 K vertical grids, with an average horizontal grid spacing about 850m and a vertical spacing of about 0.1 \( \Delta z/H \) scale. In the figure, Stations 4,5,6 refer to the locations of 3 sites where the MPA tidal current data was collected for 2 periods in August and December of 1978 at a depth of 10m. This data are used to calibrate and verify the results of our Singapore Water model. Figures 3 shows the typical comparisons of the 3D simulation results on the tidal horizontal current velocities at one of these locations for the December period. It shows a very good correlation at sigma layer 4 (depth near 10m) with the measured data.

It is noted that the depth-averaged values, computed using the external mode portion of the POM model, give a poorer prediction of the tidal current as compared with the 3D internal mode calculation over each layer. This is because the external mode is essentially a 2D computation that integrates over the vertical layers along the water column to give a single depth-averaged horizontal flow vector value. The external mode portion in the POM model is, however, essential in that it is used to solve for the sea-surface elevations into which the 3D internal mode computation is coupled.

Besides being able to predict the horizontal structures, the POM model is also able to capture the vertical structure. Figure 4 shows the vertical profiles of the computed velocity components at the 3 locations for a specific time. It is interesting to observe that at Station 5 near the greatest depth location in the Singapore Strait as seen in Figure 2, the vertical velocity profile reveals a noticeable downwelling as compared with the other 2 locations with moderate depths with values nearer to the average depth range of 35-45m. The 3D computations also reveal further variation of the current over the internal layers, such as a...
direction reversal phenomena, as depicted at Station 6 in Figure 4. At this location near to the shoreline, this phenomenon may be attributed to mechanism associated to tide turning.

Conclusion

The POM has been successfully applied to predictions of the current for waters off Singapore. This initial phase has also revealed vertical flow structures in the deeper parts of the Singapore Straits. Work is now focusing on developing a simulation for the South China Sea. As part of this simulation, databases of bottom topography, sea surface elevations, surface fluxes and boundary forcing are being researched and assembled.

References


Flow Induced Hydrodynamic Forces on a Slender Ship

Zhou Zexuan (czxzhou@ntu.edu.sg)
Edmond Lo Yat Man (cymlo@ntu.edu.sg)
Tan Soon Keat (ctansk@ntu.edu.sg)

Introduction

Computation of ship hydrodynamics has received significant attention for applications to ship maneuverability problems. Maruo and Song (1990) developed a boundary element solution for a slender ship based on 3-D Green’s functions in deep water and using Kelvin sources that satisfy the linear free surface conditions exactly. The corresponding non-linear wave problem was addressed by Kim and Lucas (1990) using Rankine sources and with iteration on the source distribution and wave profile. More recent work has focused on developing Green’s function in finite water depth (Chen and Nguyen, 2000) and using fully computational fluid dynamics (CFD) approaches (Shin and Takanori, 2000). It is well known that the hydrodynamic loading changes significantly when there is significant blockage of the flow by the hull. Thus the focus here is on developing a slender 2-D hydrodynamic code suitable for engineering predictions of the forces on a slender ship moving at a constant speed and an angle of attack in water of finite depth. Effects of applying the linear versus the full nonlinear surface conditions and finite depth are assessed.

Mathematical model

To develop a mathematical model, we considered a ship moving with a constant forward speed at an attack angle in finite water depth. In the mathematical model, the ship is fixed in a uniform cross flow with the constant velocity \( \vec{U}_0 \) in the horizontal plane. A co-ordinate system is set up with the origin on the undisturbed free surface, the \( x \)- and \( y \)-axes on the horizontal plane and the \( z \)-axis vertically upwards. The steady water motion around a slender ship is considered inviscid, incompressible and irrotational, allowing the use of a total velocity potential as

\[
\phi = \phi + \vec{U}_0 \cdot \vec{r}
\]  

(1)

where \( \phi \) and \( \vec{r} \) are the induced velocity potential caused by the ship and position vector respectively. The velocity potential is governed by Laplace’s equation,

\[
\Delta \phi = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0
\]  

(2)

No flux boundary conditions apply on the rigid ship hull and
sea bottom. The ship-induced velocity potential satisfies the far-field conditions of zero-perturbation. As this zero condition is difficult to simulate, we adopted a less stringent condition of zero perturbation velocity and the far-field boundaries were moved outward until the results converged. The usual non-linear kinematical and dynamic conditions apply on the free water surface. The linear form based on Bernoulli’s equation for pressure is

$$g \frac{\partial \phi}{\partial z} + U \frac{\partial^2 \phi}{\partial x^2} + 2U \frac{\partial^2 \phi}{\partial x \partial y} + U \frac{\partial^2 \phi}{\partial y^2} = 0 \quad z = 0$$  \hspace{1cm} (3)

The corresponding non-linear form at \( z = \eta \) is given as

$$g \left( \frac{\partial \phi}{\partial z} + U \frac{\partial^2 \phi}{\partial x^2} + 2U \frac{\partial^2 \phi}{\partial x \partial y} + U \frac{\partial^2 \phi}{\partial y^2} \right) + U \left( \frac{\partial \phi}{\partial y} \left( \frac{\partial \phi}{\partial y} \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial \phi}{\partial y} \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial \phi}{\partial z} \right) \right) + \frac{\partial \phi}{\partial t} \left( \frac{\partial \phi}{\partial y} \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial \phi}{\partial y} \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial \phi}{\partial z} \right) = 0$$  \hspace{1cm} (4)

The induced flow around the ship for small attack angles is made two-dimensional by invoking the slender body assumption. Thus the flow in each cross-section depends on the body shape and the upstream flow through the surface conditions, Equations (3) and (4).

**Computational method**

Slender body assumption is invoked so that Laplace’s equation need only be solved at each cross sectional plane (Figure 1). The boundary element method with constant elements is adopted to solve the flow-field. The normal direction of the element is assumed to be the normal direction at its mid-point so that numerical difficulties of sharp points are avoided. The discretization of flow boundaries is such that boundary elements on the hull surface are larger than those on the free water surface near the ship. This ensures that the simulation is more quickly convergent in the cases considered. The derivatives in \( x \) in Equations (3) and (4) are approximated using backward differencing and the central differencing is applied to the other derivatives. Iteration is used for the handling the non-linear terms with an approximation of using the induced velocity potential and water surface elevation from the previous cross-section as the starting estimates. Lastly, Gauss’s quadratic interpolation is applied to the integrals over boundary elements distributed on the ship hull.

For calculations using the full nonlinear surface condition Equation (4), iteration is used on both the velocity potential and water surface elevation. The latter is given by the free surface dynamic condition. The convergence criterion is that the sum of absolute values of difference in the velocity potential between successive iterations is less than the specified tolerance of \( 10^{-4} \).

As the ship is fixed in the horizontal plane, only three hydrodynamic forces arise, wave resistance, lateral force and yaw moment. These are evaluated based on pressure integrals over the hull surface. The pressure on the hull body surface is determined by the Bernoulli equation

$$P = \rho \left[ -U_o \cdot \nabla \phi - \frac{1}{2} \nabla \phi \left( \gamma + g \left( \frac{x}{d} \right) \right) \right]$$  \hspace{1cm} (5)

where \( \rho \) is the water density.

**Numerical results**

The Wigley hull is adopted for presentation of results as previous experimental results and fully 3-D results are available for comparison. Its hull surface is given by

$$[x] = f(x,z) = b \left[ 1 - \left( \frac{z}{d} \right)^3 \right] \left[ 1 - \left( \frac{z}{d} \right)^3 \right]$$  \hspace{1cm} (6)

where \( l = 1 \)m is the half ship length, \( b = 0.1 \)m is the half ship breadth and \( d = 0.125 \) m is the ship draft. The first cases considered are those in deep water. Figures 2-4 show the comparisons between the present numerical wave profiles and the experimental results from Namimatsu et al (1985), for the cases of (i) \( F_r = 0.32 \), \( \alpha = 0^\circ \) and (ii) \( F_r = 0.267 \), \( \alpha = 10^\circ \). The hydrodynamic forces and yaw moment for different water depths, attack angles and waves are also listed in Tables 1 and 2. Table 1 shows two cases on using linear wave at zero and \( 10^\circ \) attack angles. The results are given as coefficients where the wave resistance and lateral force are normalised by \( \frac{1}{2} \rho U_o^2 L^2 \) and yaw moment by \( \frac{1}{2} \rho U_o^2 L^3 \). The results indicate good agreement with past experimental water surface profile and force/moment coefficients from fully 3-D treatments. The results also showed that the non-linearity has a small effect on the water surface displacement but reduces the wave loading. The attack angle leads to the asymmetry of wave pattern and lowers the wave resistance but increases the lateral force and yaw moment.

**Table 1. Hydrodynamic forces and yaw moment comparisons for deep-water (\( F_r = 0.267 \))**

<table>
<thead>
<tr>
<th>Sources</th>
<th>( \alpha )</th>
<th>( C_w )</th>
<th>( C_l )</th>
<th>( C_y )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Expt (Namimatsu et. al, 1985)</td>
<td>0º</td>
<td>0.0001251</td>
<td>0.0000000</td>
<td>0.0000000</td>
</tr>
<tr>
<td>Maruo and Song (Linear wave)</td>
<td>0º</td>
<td>0.0001207</td>
<td>0.0000000</td>
<td>0.0000000</td>
</tr>
<tr>
<td>Present results (Linear wave)</td>
<td>0º</td>
<td>0.0001331</td>
<td>0.0000000</td>
<td>0.0003016</td>
</tr>
<tr>
<td>Present results (Linear wave)</td>
<td>10º</td>
<td>0.0001238</td>
<td>0.008407</td>
<td>0.003069</td>
</tr>
</tbody>
</table>
Figures 5-6 and Table 2 show the effect of finite depth on wave pattern, hydrodynamic forces and yaw moment. The finite depth effect is much stronger when $h/d < 2$ with significant increases in the hydrodynamic forces and yaw moment. For $h/d > 5$, the deep-water results are essentially reached.

**Conclusion**

The slender body assumption is invoked to reduce the non-linear flow-field around a slender ship into a series of 2-D sections that are solved using the boundary element method. Iteration is used to handle the non-linearity at the free water surface. Numerical results are presented for a Wigley hull with good agreement with previous fully 3-D results. This indicates that the use of the slender body assumption in simplifying the flow-field computations gave acceptable engineering results and can be extended for predictions in finite depth.

**References**


Towards a Predictive System for Ship Manoeuvres

Li Wei (cwli@ntu.edu.sg)
Edmond Lo Yat Man (cymlo@ntu.edu.sg)
Tan Soon Keat (ctansk@ntu.edu.sg)

Introduction

A computational capability for predicting ship manoeuvres is being developed under a current joint MPA-NTU project. The goal is to enhance navigational safety for ships in coastal port waters. The predictive system is based on solving the rigid body equations of motion for the ship, taking into account the effects of prevailing environmental forces such as from ambient currents and wind, as well as the forces from the ship propulsion and rudder systems.

Modelling of ship motions

As only the ship track is desired, only the surge, sway and yaw motions (i.e. a 3-dof system) are simulated. As a first approach, empirical formulas for deep water are used to calculate the forces from the ship’s propeller and rudder, ambient wind and current, and resistance from wave-making. The standard equations of surge (u), sway (v) and yaw (r) motion in a ship centred coordinate system are:

\[
\begin{align*}
    m(u - rv - x_G r^2) + M_{xx} u &= F_p + F_{wx} + F_{cx} + F_r \\
    m(v + ru + x_G r) + M_{yy} v - M_{yx} r &= F_{wy} + F_{cy} + F_r \\
    I r + m x_G (v + ru) + I_{xz} r + I_{yz} v &= N
\end{align*}
\]

where \( m \) is the ship mass and \( I \) is the moment of inertia. \( M_{ij} \) and \( I_{ij} \) are the added masses and added moments of inertia. The term \( x_G \) denotes the displacement of the centre of gravity from the ship’s centroid, which is used as the origin of the ship centred coordinate system. The forces simulated are denoted by \( F_p, F_r, F_c, \) and \( F_w \) representing propeller, rudder, current (i.e. hull induced including wave-making) and wind forces respectively. The definition of the coordinate system is sketched in Fig. 1 along with the heading angle \( \psi \), speed \( V \), drift angle \( \beta \) and ambient current in a fixed inertial frame \( x_o \) and \( y_o \).

Wind force

The wind forces are formulated as

\[
\begin{align*}
    F_{ws} &= \frac{1}{2} \rho_s C_w V_w^2 A_w \\
    F_{wv} &= -\frac{1}{2} \rho_s C_v V_w^2 A_v \\
    M_w &= \frac{1}{2} \rho_s C_n V_w^2 I_{xx} A_v
\end{align*}
\]

where \( V_w \) is relative wind velocity, and \( A_w \) and \( A_v \) are the projected cross sectional areas above the water line in the longitudinal and transverse directions respectively. The force coefficients \( C_w, C_v, \) and \( C_n \) depend on the relative wind direction from the ship’s bow and can be found from Harvald [1] as reproduced in Fig. 2.

Calculation of current interaction

The current force arises due to the motion of the ship relative to the ambient current in the area. In the standard ship resistance formulation, the total resistance \( R \) is given:

\[
R = R_w + (1+k) R_f
\]

where \( R_w \) is frictional resistance, \( R_m \) is wave-making resistance and \( k \) is a form factor accounting for the form drag arising from the ship’s wake.
For our application, we take the current $V_e$ relative to the ship as the ambient current velocity relative to the ship velocity. Then frictional resistance along the longitudinal direction is assumed to be given by:

$$R_f = 0.5C_f S \rho |V_{ex}| V_{ex}$$

(8)

where $V_{ex}$ is x component of $V_e$, S is the wetted area and $C_f$ is frictional resistance coefficient according to ITTC 1978 friction formulation:

$$C_f = \frac{0.075}{(\log_{10} R_n - 2.0)^2}$$

(9)

with $R_n$ being the Reynolds number based on the ship length.

The wave-making resistance is similarly taken as:

$$R_w = 0.5 \rho C_k S |V_{ex}| V_{ex}$$

(10)

Standard values of $C_k$ as given by the ship’s Froude number $F_n$ and prismatic coefficient can be found in Harvald [1].

The lateral current force and moment is formulated using strip theory and $V_{ey}$, the y-velocity component of $V_e$. Essentially the ship is divided into strips with each strip being treated as a 2-D section. The transverse force is written as an integral over the sections:

$$F_{sy} = 0.5 \rho \int (dC_D(x) D(x)) |V_{ey}| V_{ey}$$

(11)

where D(x) is draught of each ship section and $C_D(x)$ is the 2-D drag coefficient. Similarly the yawing moment about the ship’s centroid is expressed as:

$$N_y = 0.5 \rho \int (dx C_D(x) D(x)) |V_{ey}| V_{ey}$$

$$+ 0.5 (M_{sy} - M_{sx}) |V_{e}|^2 \sin 2\beta$$

(12)

The second term in Equation (12) is the Munk moment and is neglected here.

**Calculation of propeller force**

The thrust T of standard propellers could be written as (see e.g. Carlton [2]):

$$T = K_t \rho n^2 D^4$$

(13)

where $K_t$ is the thrust coefficient, D is the propeller diameter and n is the revolution rate (Hz).

In general, $K_t$ is given as a function of the advance ratio J:

$$J = V_s / (Dn)$$

(14)

where $V_s$ is the ship advance speed ($V_{ex}$ here) modified by a wake fraction $W_t$ to account for the interference of the hull, i.e.

$$V_s = V_{ex} W_t$$

(15)

A form suggested for $W_t$ (reference [3]) is given through the block coefficient $C_i$:

$$W_t = 1 - 0.6C_i$$

(16)

Finally the thrust deduction fraction t further modified the final propeller force $F_p$ delivered as:

$$F_p = (1-t) T$$

(17)

**Calculation of rudder force**

The force on rudder normal to it surface is:

$$F_N = -0.5 A_r \rho |V_e| C_N \sin \delta_r$$

(18)

where $A_r$ is the rudder and $\delta_r$ is the relative rudder angle (i.e. the attack angle relative to the flow). From this, the rudder forces and moment can be written as:

$$F_{rx} = F_N \sin \delta$$

(19)

$$F_{ry} = -F_N \cos \delta$$

(20)

$$N_r = -x_r F_N \cos \delta$$

(21)

where $x_r$ is the location of the rudder. The force coefficient $C_N$ can be estimated using thin airfoil theory with corrections for a finite vertical span (Harvald [1]).

**Preliminary calculations**

The equations of motion (Equation (1) to (3)) are solved in time using a mid-point integration scheme with coding in Visual Basic. The ship mass properties (i.e. mass and moments of inertia) are based on a detailed calculation accounting for the mass and load distribution of the ship. The added mass and moments are estimated using potential flow for an ellipsoid in an infinite fluid domain.

A 45° course change calculation was carried out for a container-sized ship with an approach speed of 22.3 knots and the results are shown in Fig. 4. There are over- and under-shoots in the heading angle due to the lack of hydrodynamic damping.

A turning circle manoeuvre was also calculated at the same approach speed as shown in Fig. 5, where the predictions are

![Figure 4. Course changing calculation](image-url)
Studies are now underway using inviscid hydrodynamic calculations to better determine the ship wave making forces/moment and as well as for the added mass and moments of inertia. Computational fluid dynamic calculations are also being carried out for a double-hull ship geometry to better determine the viscous forces/moment. These calculations are also being performed for water of finite depth where the ship draft can occupy a significant fraction of the water depth. However it is noted that, eventually, a detailed comparison with ship sea trail data will be necessary in order to calibrate and verify the accuracy of the predictions. The forcing coefficients being assembled from various sub-theories will invariably need to be fine tuned for the specific ships simulated.

Acknowledgements

The technical assistance of Det Norske Veritas, Republic of Singapore Navy, Maritime & Port Authority of Singapore and Neptune Ship Management Services Pte Ltd in this work is gratefully acknowledged.

References


A Diffusive Model for Evaluating Thickness of Bedload Layer

N S Cheng (cnscheng@ntu.edu.sg)

Introduction

The thickness of the bedload layer can be defined as the saltation height of particles, which is of the order of the particle size. This thickness is usually required to formulate the bedload transport rate analytically. Einstein (1950) simply took the thickness to be two particle diameters, independent of flow conditions. His assumption is acceptable only on qualitative grounds. In comparison, several subsequent studies show, either experimentally or numerically, that the thickness is closely related to characteristics of flow and particles.

It seems that no analytical results are available in the literature for determining the thickness of the bedload layer. This paper attempts to present a framework for an analysis that can be performed using a diffusive model. This study was partially motivated by recent studies on the hydrodynamic diffusion conducted largely in chemical engineering, which indicate that the hydrodynamic particle-particle interaction can effectively be described as a diffusive process. Depending on flow conditions and properties of particles, different diffusion mechanisms can be identified associated with the transport of particles in fluids. For colloidal particles, the diffusion caused by the thermal effect is significant, leading to the Brownian motion of small particles. If the ambient turbulence is intensive, the diffusive characteristics of particles are largely associated with turbulent eddies. For certain conditions, the turbulent diffusion can be analogised to the momentum transfer of fluid. It is with this assumption that the well-known Rouse equation is derived for computing the concentration profile of suspended particles in open channel flows. The hydrodynamic diffusion refers to the phenomenon that a particle executes a random walk solely because of the particle-particle interaction. It is not due to...
the thermal effect and can take place even at very small particle Reynolds numbers, implying that it is not caused by the fluid inertia either. This phenomenon has been confirmed experimentally and numerically.

Bedload comprises near-bed particles, which experience continuous contacts with the bed. These contacts can be in the mode of rolling, sliding, or saltation. This implies that hydrodynamic interactions among the particles have a commanding influence on the bedload transport. In this study, the turbulence effect on the bedload transport is not considered. A diffusive model is therefore proposed for describing the particle flux perpendicular to the flow direction, which enables evaluation of the thickness of the bedload layer.

**Theory**

Consider a bedload layer. The initial elevation of the bed surface is $z_0$. For a certain flow condition, an equilibrium interface at the elevation, $z_c$, can be identified between the bedload of which the particles are moving and the stationary particles beneath the bedload. The thickness of the bedload layer, $\delta$, is therefore equal to the distance from the top of the bedload to the interface, i.e., $(z_b - z_c)$. If the bed particles move typically in the mode of saltation, then the bedload thickness can be easily measured as the saltation height.

The bed particles, once saltating, will fall down to the bed because of the gravitational effect. The downward flux of the particles, $F_s$, can be related to the settling velocity, $w_{m}$, and the local concentration of the particles, $c$:

$$F_s = w_m c$$  (1)

where the subscript $m$ indicates the settling velocity occurring in the particle-fluid mixture. This settling velocity can be further expressed in terms of the settling velocity, $w_m$, under a very dilute condition,

$$w_m = w_r w_o$$  (2)

where $w_r$ = relative settling velocity depending on the concentration as well as the properties of the particle. Usually, it takes the following form:

$$w_r = (1 - c)^n$$  (3)

where $n$ can be empirically related to the dimensionless particle diameter, $D^*$, or the settling Reynolds number. Alternatively, a generalized approach for evaluating $w_r$ is used in this study, as detailed in the subsequent section.

In addition to the gravity-driven particle flux, the upward hydrodynamic diffusive flux is

$$F_d = -E \frac{dc}{dz}$$  (4)

where $E = \text{shear-induced diffusion coefficient}$. It is this diffusive flux that causes the bed dilatation in the vertical direction. At equilibrium, the net flux of the particle is zero. Equating (1) and (4) yields

$$w_r c = -E \frac{dc}{dz}$$  (5)

Note that the particle concentration for bedload is zero above $z = z_0$, and approaches the maximum value, $c_m$, for the condition of the densely parked particles below $z = z_c$. Integrating (5) from $z = z_c$ to $z = z_b$ for the bedload layer yields

$$\int_{z_c}^{z_b} -w_r c \text{d}z = w_r (z_b - z_c)$$  (6)

Therefore, the elevation difference, $(z_b - z_c)$, i.e., the thickness of the bedload layer, $\delta$, can be expressed as

$$\delta = \frac{1}{w_r} \int_{z_c}^{z_b} E \text{d}c$$  (7)

From (7), it follows that the thickness can be analytically evaluated provided that detailed information on the diffusivity, $E$, and the relative settling velocity, $w_r$, is available.

**Hydrodynamic Diffusivity, $E$**

Previous studies show that the hydrodynamic diffusivity in shear flows, $E$, is generally proportional to the bulk velocity gradient and the square of the particle diameter:

$$E = E_0 D^2 \frac{du_m}{dy}$$  (8)

where $du_m/dy = \text{velocity gradient of the bulk flow}$; $u_m = \text{velocity of the mixture}$; and $E_0 = \text{dimensionless diffusivity depending on the particle concentration}$. The $E_0$-value deduced from experimental measurements generally increase with increasing particle concentration. However, a general formulation of such a relationship is not yet available.

Alternatively, the diffusivity is assumed in this study to be equivalent to that associated with the momentum transfer induced by the random particle motion. This implies that the diffusivity can be related to the particle-related shear stress, $\tau_p$:

$$E = \frac{\tau_p}{\rho_p} \frac{du_m}{dy}$$  (9)

where $\rho_p = \text{density of particles}$. The assumption made here is quite similar to that used for deriving the Rouse equation, where only the turbulent diffusion is concerned.

The particle shear stress, in general, may comprise three components contributed by the presence of particles, random motion of particles, and inter-particle collision, respectively. When particles are simply present in a fluid, even without relative motion among them, the rheological property of the fluid is modified. On the other hand, the random motion and collision effect lead to a ‘turbulent’ component of the particle shear stress. If the ‘turbulent’ component is predominant, the particle shear stress can be derived as

$$\tau_p = \tau_t$$  (10)

where $\tau_t = \text{turbulent shear stress}$. Replacing $\tau_t$ by $\tau_p$ in (9) yields

$$E = \frac{\tau_p}{\rho_p} \frac{du_m}{dy}$$  (9)

where $\rho_p = \text{density of particles}$. The assumption made here is quite similar to that used for deriving the Rouse equation, where only the turbulent diffusion is concerned.

The particle shear stress, in general, may comprise three components contributed by the presence of particles, random motion of particles, and inter-particle collision, respectively. When particles are simply present in a fluid, even without relative motion among them, the rheological property of the fluid is modified. On the other hand, the random motion and collision effect lead to a ‘turbulent’ component of the particle shear stress. If the ‘turbulent’ component is predominant, the particle shear stress can be derived as

$$\tau_p = \tau_t$$  (10)

where $\tau_t = \text{turbulent shear stress}$. Replacing $\tau_t$ by $\tau_p$ in (9) yields

$$E = \frac{\tau_p}{\rho_p} \frac{du_m}{dy}$$  (9)

where $\rho_p = \text{density of particles}$. The assumption made here is quite similar to that used for deriving the Rouse equation, where only the turbulent diffusion is concerned.

The particle shear stress, in general, may comprise three components contributed by the presence of particles, random motion of particles, and inter-particle collision, respectively. When particles are simply present in a fluid, even without relative motion among them, the rheological property of the fluid is modified. On the other hand, the random motion and collision effect lead to a ‘turbulent’ component of the particle shear stress. If the ‘turbulent’ component is predominant, the particle shear stress can be derived as

$$\tau_p = \tau_t$$  (10)

where $\tau_t = \text{turbulent shear stress}$. Replacing $\tau_t$ by $\tau_p$ in (9) yields

$$E = \frac{\tau_p}{\rho_p} \frac{du_m}{dy}$$  (9)

where $\rho_p = \text{density of particles}$. The assumption made here is quite similar to that used for deriving the Rouse equation, where only the turbulent diffusion is concerned.
where \( G \) = average gap among neighbouring particles and \( \alpha \) = coefficient. Substituting (10) into (9) yields
\[
E = \alpha \frac{D^4}{G^2} \frac{\mathrm{d}u_n}{\mathrm{d}y}
\]
Comparing (8) to (11) leads to
\[
E_\ast = \alpha \left( \frac{D}{G} \right)^2
\]

The average gap among neighbouring particles can be determined using the following approach. Consider a certain amount of particles dispersed randomly in a fluid, which yields a volumetric concentration, \( c \). On average, it is assumed that a single particle can be enclosed in a finite volume, \( V \), so that the ratio of the particle volume, \( \pi \frac{D^3}{4} \), to \( V \) is equal to the concentration, i.e.,
\[
\frac{\pi D^3}{4} \frac{1}{V} = c
\]
With (13), if the volume \( V \) is spherical, then its diameter is found to be \( Dc^{-1/3} \), which can be considered as the average distance between the centres of two neighbouring particles. Therefore, the average gap, \( G \), is equal to
\[
G = Dc^{-1/3} - D
\]
Substituting (14) into (12) leads to
\[
E_\ast = \alpha c^{-1/3} \left( 1 - \frac{1}{c} \right)^2
\]
With \( G \) given by (14) and some existing experimental data, one can find that the \( \alpha \)-value varies from 0.06 to 0.19. Furthermore, note that
\[
\frac{\mathrm{d}u_n}{\mathrm{d}y} = \frac{\tau_b}{\mu_n}
\]
where \( \tau_b \) = bed shear stress; \( \mu_n = \mu \mu_e \) = effective viscosity of the mixture; and \( \mu_e = \mu \) = relative viscosity. Substituting (16) into (11), after manipulation, yields
\[
E = E_\ast \frac{D^4 \tau_b}{\mu_e} \frac{\mu_e}{\mu_e^*} \frac{1}{\mu_E\frac{1}{\mu_e^*} \rho}
\]

Relative Viscosity, \( \mu_r \)
Many empirical formulas are available in the literature for computing the relative viscosity. Their predictions differ in particular for high concentrations. These formulas could be represented well by the following exponential function:
\[
\mu_r = \exp \left[ \frac{2.5}{\beta} \left( \frac{1}{1 - c} \right)^{1/3} - 1 \right]
\]
where \( \beta \) = constant. By comparing (18) with the existing empirical relationships, the \( \beta \)-value is found to be equal to approximately 1.0 to 3.9.

Relative Settling Velocity, \( w_r \)
For the particle-fluid mixture, the settling Reynolds number can be defined as
\[
R_m = \frac{\rho_n w_o D}{\mu_n}
\]
where \( \rho_n = \rho_c + \rho(1 - c) = (1 + c\Delta)p = \text{density of the mixture}; \) and \( \Delta = \rho_c / \rho - 1 \). If the particle concentration is vanishingly small, \( R_m \) reduces to \( R_o = \rho w_o D / \mu \). With \( R_m \) and \( R_o \) the relative settling velocity, \( w_r \), can be expressed as
\[
w_r = \frac{\mu_r \rho_n R_m}{\mu R_o} = \frac{\mu_r \rho_n R_m}{1 + c\Delta R_o}
\]
Generally, the settling Reynolds number, \( R_m \), is a function of the dimensionless particle diameter, \( D_\ast \). For natural sediment particles, this function can be formulated as:
\[
R_m = \left( \frac{2.5 + 1.2 D_\ast^2}{5} \right)^{1.5}
\]
As an approximation, (21) can be further applied to the particle-fluid mixture, yielding:
\[
R_m = \left( \frac{2.5 + 1.2 D_{\ast m}^2}{5} \right)^{1.5}
\]

Relative Thickness of Bedload Layer, \( \delta / D \)
Substituting (17) into (7), one can express the relative thickness as
\[
\frac{\delta}{D} = \frac{\tau_b D_{\ast m}^{1/3}}{R_o} \int_0^{c_{\ast m}} \frac{E_\ast}{c \mu_r w} \, dc
\]
where \( E_\ast, \mu_r, \) and \( w_r \) can be evaluated using (15), (18) and (25), respectively.

To the writer’s knowledge, very limited experimental data of the thickness of the bedload layer are available in the literature to fully verify the present study. This may be because the photographic technique used for tracing the particle movement applies only to very low rates of sediment transport, which imply few particles saltating over a flat
bed. In particular, a flat bed comprised of fixed particles is often adopted in experiments. This bed is suitable for conducting measurements but not realistic. On the other hand, with increasing bed shear stress, the sediment bed with movable particles can easily develop from the flat bed into bedforms such as ripples and dunes. In the presence of bedforms, the height of ripples or dunes is more measurable and meaningful than the saltating height of individual particles for sediment transport, even though they may be closely connected.

However, it is noted that at high shear stresses, bedforms may disappear, the bed particles moving largely in the sheet modes. For this extreme condition, the linear relationship, \( \delta/D = \tau^* \), seems to be possible, as reported by Wilson (1987) and Sumer et al. (1996). The relationship approximated first by Wilson is \( \delta/D = 10\tau^* \), which agrees acceptably with Sumer et al.’s experimental data.

**Conclusion**

Applying the concept of the hydrodynamic diffusion, which is caused by particle-particle interactions, to bedload transport leads to an analytical model that enables the evaluation of the thickness of the bedload layer. The derived results indicate that the bedload thickness is related to the dimensionless particle diameter and dimensionless bed shear stress. However, further research needs to be done to fully verify and improve the analysis presented in this study.

**References**


---

**Total Sediment Discharge in Alluvial Channels**

Lim Siow Yong (cslim@ntu.edu.sg)

Yang Shu-Qing

**Introduction**

The mechanism of sediment transport has been a subject of study for decades due to its importance, especially in the design and operation of canal systems and river regulation. To date, there are many formulas for the calculation of the rate of sediment discharge in alluvial channels, but so far few have gained general acceptance. There are basically three types, i.e., bed-load, suspended load and total load formulae. The wash load is usually not included in these formulas. The total load can be computed indirectly as the sum of the bedload and suspended load. This method, however, contradicts what is observed under natural flowing conditions where both modes of sediment motion are inter-changeable, i.e., as bed-load on one instance and as suspended load on the next, depending on the sediment characteristics and flow conditions. The purpose of this paper is to develop an empirical equation to compute the total sediment discharge based on a new total-load transport parameter, \( T_T \) and to test the new equation using 3500 data sets covering a wide range of laboratory and field measurements.

**Dimensional considerations**

In the study of sediment transport capacity of a flow, the following characteristic parameters are considered important: (1) flow conditions: flow depth \( h \), channel width \( B \), mean flow velocity \( V \), energy slope \( S \), and gravitational acceleration \( g \), (2) fluid properties: specific weight \( \gamma \), and kinematic viscosity of water \( \nu \), and (3) sediment properties: median grain diameter \( d_{50} \), specific weight of sediment \( \gamma_s \), and its submerged weight \( \gamma_s - \gamma \), and grain fall velocity \( \omega \). In sediment transport, the shear velocity is more meaningful than the mean velocity. We will adopt the effective grain shear velocity, \( u_s \) proposed by van Rijn (1984) to replace \( V \). This is physically more logical because for mobile bed channels with bedforms, the grain shear is responsible for transporting the bed material and the form drag does not contribute to this process. The grain shear velocity can be computed using:

\[
\frac{V}{u_s} = 2.5 \ln \left( \frac{11h}{2d_{50}} \right)
\]  

Therefore, the total sediment discharge \( g_t \), in dry weight per second per unit width (N/s/m) for a steady and uniform flow in an alluvial channel can be expressed as:

\[
g_t = f(h, B, u_s', S, \gamma_s, \gamma, \nu, d_{50}, g, \omega) \]  

Using dimensional analysis, the dimensionless form of Eq. Column2
In Eq. (3), the term $B/h$ reflects the influence of the channel aspect ratio and needs to be considered if the wall effects are significant, i.e., if $B/h < 5$. The second term on the right side of Eq. (3) is the Froude number based on $u_c$. The term $u_c h/v$ is the Reynolds number which reflects the influence of $v$. For fully developed channel flows, this term can be ignored. The term $h/d_50$ reflects the influence of the relative roughness on the bed, and would be important if it becomes too small. The terms, $\gamma$ and $\gamma_s - \gamma$ need to be included to reflect the specific weight of the sand used and it is more meaningful when lightweight material is used in the experiments or if air is used as the transporting medium. The last term $u_f^2/\omega$ reflects the importance of the grain fall velocity in the study of sediment transport.

Data analysis

An initial assessment of the influence of the dimensionless parameters in Eq. (3) on $g_t$ has been conducted using Stein’s (1965) data. The 56 data sets include all stages of bed form development.

Figure 1 shows the relationships of $g_t$ against $u_c$, $S$ and $u_f^2/\omega$ and the correlation coefficients, $R$ obtained were 0.66, 0.95 and 0.98, respectively. The latter R value indicates that the sediment discharge has the best correlation with the product, $u_f^2/\omega S$ and the trend in Figure 1 shows a linear relationship of the form:

$$g_t \propto c_1 \left( \frac{u_f^2}{\omega} S \right)$$

$$g_t \propto c_1 \left( \frac{\gamma_s - \gamma}{\gamma} \frac{u_f^2}{\omega} \right)$$

where $\tau_o = \gamma h S = $ bed shear stress. Eq. (5) can be rewritten as:

$$g_t = c_1 \left( \frac{\gamma_s - \gamma}{\gamma} \frac{u_f^2}{\omega} + b \right)$$

where $b = $ constant. Substituting the initial condition for sediment transport, i.e., $g_t = 0$, when $u_f = u_c$, where $u_c = $ critical shear velocity from the Shields curve, into Eq. (6) yields:

$$g_t = c_1 \left( \frac{\gamma_s - \gamma}{\gamma} \frac{u_f^2 - u_{c0}^2}{\omega} \right)$$

Figure 2 shows the variation of $c_1$ with $\gamma / \gamma_s - \gamma$. The figure shows that $c_1$ increases almost linearly as the bed material becomes lighter, i.e. as $\gamma / \gamma_s - \gamma$ increases. For most practical purposes, $c_1$ can be expressed as:

$$c_1 = k \left( \frac{\gamma}{\gamma_s - \gamma} \right)$$

where $k = $ constant = 12.5. Substituting Eqs. (8) into (7), we get a total sediment discharge formula of the form:

$$g_t = k \left( \frac{\gamma_s - \gamma}{\gamma_s - \gamma} \right) \tau_o \left( \frac{u_f^2 - u_{c0}^2}{\omega} \right)$$

where $T_T = \tau_o \left( \frac{u_f^2 - u_{c0}^2}{\omega} \right)$ is defined as the total-load transport parameter. The variables in $T_T$ can be obtained easily from field or flume measurement. The fall velocity in $T_T$ can be calculated using the formula proposed by Cheng (1997).

Figure 3 shows comparisons between the calculated and measured total load for the US Waterway Experimental Station data. The agreement is reasonably good with most of
Figure 3. Effect of sediment density on the predictability of (9), based on US Waterway Experiment Station’s data (1936).

The data within the ±100% error band. Other data were also used to verify the applicability of Eq. (9). Altogether 3500 published total load data from field and flume studies have been analyzed. Figure 4 shows a typical comparison between the calculated and measured total load with data from various sources. Generally, the results show that 84% of the 3500 data were predicted within the 0.5 and 2 times of the measured values, and \( k = 12.5 \) remains a constant irrespective of the hydraulic conditions.

A comparison on the predictability of Eq. (9) with a few formulae (Engelund and Hanson 1972, van Rijn 1984, and Karim 1998) for lightweight materials is given in Table 1, where \( r = \) discrepancy ratio = \( g_t \) (measured)/\( g_t \) (computed). The last row in Table 1 shows the overall results and it can be seen that Eq. (9) yields the best results. van Rijn’s equation also score well for this type of sediment.

### Table 1. Comparisons of various predictive formulas for lightweight materials using the US Waterway Experiment Station data (1936)

<table>
<thead>
<tr>
<th>( \rho_s ) (T/m³)</th>
<th>Runs</th>
<th>( d_{50} ) (mm)</th>
<th>% Score of predicted sediment discharge and range of discrepancy r</th>
<th>( 0.75 &lt; r &lt; 1.5 )</th>
<th>( 0.5 &lt; r &lt; 2 )</th>
<th>( 0.33 &lt; r &lt; 3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.85</td>
<td>26</td>
<td>0.96</td>
<td>50% 50% 61% 15% 69% 76% 85% 23% 92% 88% 92% 65%</td>
<td>0.96</td>
<td>45%</td>
<td>51% 40% 10% 69% 78% 68% 25% 81% 84% 81% 56%</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.317</td>
<td>Y-L*, E-H, VR, K</td>
<td>0.317</td>
<td>45%</td>
<td>51% 40% 10% 69% 78% 68% 25% 81% 84% 81% 56%</td>
</tr>
<tr>
<td>Total</td>
<td>298</td>
<td>48% 36% 40% 13%</td>
<td>72% 58% 63% 24% 88% 73% 85% 52%</td>
<td>72% 58% 63% 24% 88% 73% 85% 52%</td>
<td>72% 58% 63% 24% 88% 73% 85% 52%</td>
<td>72% 58% 63% 24% 88% 73% 85% 52%</td>
</tr>
</tbody>
</table>

*Note on formulas: Y-L = Yang and Lim, E-H = Engelund and Hansen, VR = van Rijn, K = Karim

Conclusion

A new and user-friendly total sediment discharge formula (Eq. 9) for the computation of total load under equilibrium transport condition has been developed based on dimensional analysis. The total sediment discharge, \( g_t \), is linearly related to the new total-load transport parameter, \( T_r \), and a factor of
proportionality, \( k \). Using 3500 sets of published total-load data from both field and flume studies, \( k \) has been found to be a constant = 12.5. The range of the variables tested were: median sediment size from 0.011mm to 28mm, sediment gradation up to 2.07, specific gravity of sediment from 1.03 to 2.65, flow Froude number up to 2.8, sediment concentration up to 110 kg/m³, water temperature from 1.6°C to 63°C, water depth from 0.03 m to 16.4 m, and channel aspect ratio as small as 0.3. The verification exercise for the new equation show, on average, 84% of the data were predicted within the 0.5 and 2 times of the measured values. Considering the large database used and the range of applicability of the formula, the result obtained is comparable, if not better than most of the existing total sediment discharge formulae.

References


Double Diffusive Effect on Mixed Brine Discharges from Desalination

A W K Law (cwklaw@ntu.edu.sg)

Introduction

A concern with brine discharges from desalination plants is the increase in salinity in the surrounding waters near the sea bed level due to poor initial mixing. The induced stratification can inhibit the exchange of matter between the benthic layer and the overlying ambient waters, and lead to an accumulation of chemicals from the pre-treatment process in the bottom sediment. Thus, the brine discharge is sometimes merged with neighboring cooling water before being released through outfalls. Besides the dilution with the merging, the mixture of heat and salinity is also deliberately arranged to yield an almost neutrally buoyant discharge (i.e. the density of the discharge being the same as the ambient waters), so that the mixed brine plume would not have the tendency to sink and hence can be pushed further out to the sea. A mixed discharge of heat and salinity is also the natural outcome of desalination with a multi-stage distillation approach.

To assess the initial mixing of the mixed discharges, the possibility of differential transport of heat and salinity through double diffusion needs to be addressed. The much larger molecular diffusivity of heat than salt can lead to the phenomenon of salt-fingering in the case of an overlaying warm salty layer. This drives a downward salinity flux as a result of their relatively cooler and saltier anomalies than the surrounding ambient. With salt-fingering, the trajectory of a mixed jet can differ significantly from what is intended to be a neutrally buoyant jet.

In this study, we investigate the effect of double diffusion on the mixed brine discharge through laboratory experiments. The study is divided into two parts. The first is with a surface rectangular jet that emulates the prototype geometry, while a submerged round jet is studied in the second part. For both, the discharge has the same density as the ambient water, with the increase in density due to the excess salinity compensated by the equivalent reduction due to the elevated temperature. Without initial density differences, classical analysis would suggest that the jet would simply propagate horizontally. Due to space constraint here, we shall only discuss the results and analysis of the submerged round jet.

Experiments

The experiments were conducted in a glass test tank with dimensions 2.86 m length x 0.86 m width x 1 m depth. The mixed brine was issued from a horizontal circular pipe with an inside diameter \( D \) of 5 mm and located at approximately mid-depth. The densities of both the ambient and discharged fluid were monitored by a digital density probe. Fluorescent dye was used to track the path of salt transport.

12 tests were conducted that can be grouped into three different sets, TA, TB and TC, with varying discharge velocity \( U_o \) of 1.6, 0.4 and 0.08 m/s respectively. Sets TA and TB were in the turbulence range, while the flow in Set TC was actually laminar due to the low \( U_o \). Controlled experiments
were performed without any temperature rise for the first test in each of the three sets. For all the controlled experiments, the jet propagated horizontally towards the end wall as anticipated.

In the first Set TA, the trajectories were also near horizontal up to approximately 60D for the three tests with \( \Delta T \) at 4, 8 and 12°C respectively. Thereafter the jet showed a downward inclination of up to 3D towards the end of the monitoring range.

Set B saw similar actions with the jet gaining negative buoyancy due to the accumulation of buoyancy anomaly and curving downwards. Sample snapshots for Set TB are shown in Figure 1, and the trajectories in Figure 2. The jet trajectory for Tests TB3 and TB4, with a \( \Delta T \) of 8 and 12°C respectively, are quite similar. Test TB2 with \( \Delta T = 4 \)°C has a smaller curvature comparatively. A further decrease of \( U_o \) in Set TC showed even more significant double-diffusive effect.

**Analysis**

In this section, we shall perform a quantitative analysis of the trajectory of a round jet that is subjected to double diffusion action. The analysis begins with the common characteristic of a buoyant jet, that the horizontal momentum flux \( M_x \) remains constant along the trajectory.

\[
M_x = \int_0^\infty u^2 \pi r \cos \theta dr = U_o \frac{\pi D^2}{4} \tag{1}
\]

where \( u \) is the axial velocity and \( \theta \) the jet inclination. The vertical momentum, on the other hand, is accelerated by the downward buoyancy as:

\[
\frac{dM_z}{ds} = \frac{d}{ds} \int_0^\infty u^2 2\pi r \cos \theta dr = \frac{\Delta \rho}{\rho_s} g \frac{\pi b^3}{3} \tag{2}
\]

where \( b \) the jet width and \( \Delta \rho \) the average density excess over the cross-section.

When salt fingers develop over the bottom interface, excess density accumulates inside the fingers due to the salinity surplus over the surrounding ambient. The excess density is the source of the negative buoyancy that forces the downward migration of the salinity. The rate of increase of the total density excess inside the fingers should thus be proportional to the salinity flux across the interface. Hence, we can express the cross-sectional average density buildup, \( \Delta \rho / \rho_s \), as follows:

\[
\frac{\Delta \rho}{\rho_s} = \frac{2\beta \hat{F}_s bdx}{Q} \tag{3}
\]

\[
\beta \hat{F}_s = \gamma (gK_f)^{\frac{1}{2}} (\beta \Delta S)^{\frac{1}{2}} \tag{4}
\]

where \( \Delta S \) is the average salinity and \( Q \) the volume flux. From Wang and Law (2002), the increase in the discharge and jet width of a round jet with distance can be quantified as:

\[
b = \eta_{m_w} x \tag{5}
\]

\[
Q = k_{m_w} U_s \Delta m_{m_w} x \tag{6}
\]

with \( k_{m_w} = 6.48, \eta_{m_w} = 0.105 \).

In the presence of shear turbulence, the change in salinity inside the jet due to turbulent entrainment is large compared to the salt-fingering flux. The average salinity can then be computed as:

\[
\Delta \hat{S} = -\frac{\pi D^2 U_o \Delta S_o}{Q} \tag{7}
\]

Substituting the above equations into (2) and assuming that the jet is near-horizontal yield

\[
M_y = K x \hat{\gamma} \tag{8}
\]
where

\[ K = \frac{9(2)^n \gamma e^2 K_x}{64 k_x^2 \eta_x^2} \left( D/\Delta S_x \right)^{\gamma} \]  

(9)

The trajectory of the jet can now be estimated as:

\[ \frac{dy}{dx} = \frac{M_y}{M_x} \]  

(10)

Therefore

\[ \gamma = \left( \frac{9 K D^{\gamma_x}}{88 M_x} \right) \left( \frac{x}{D} \right)^{\frac{\gamma}{\gamma_x}} \]  

(11)

The power law in Equation (11) shows a high exponent of $11/3$ for the horizontal distance, which explains why the jet can bend downward rapidly despite having the same density as the ambient initially. The relatively longer trajectory for Test TB2 was used to calibrate the value of $\gamma$, yielding $\gamma = 5.14$. The calculated trajectories for Set TB are shown in Fig 2 respectively.

A few observations can be drawn here. First, the overall agreement between the calculated and measured trajectories is satisfactory given the uncertainties. Second, the relative effect on the trajectory due to the salinity increase over the three experiments in each individual set is well captured, implying the validity of (4) in correlating the excess density with the cross-sectional average salinity. Third, the coefficient $\gamma$ remains unchanged throughout the range of discharge velocity. Since Set TA is well into the turbulent range, the magnitude of $\gamma$ should also be applicable in prototype conditions.

Summary

The present study shows that the double diffusive salt-fingering action can lead to a strong migration of the brine salinity towards the seabed, and should be taken into account in the environmental assessment. Analysis shows that the downward salinity migration is triggered by the excess density buildup inside the salt-fingering interface. Further verification of the results in the field scale will be necessary in the future.

Reference