E-proceedings of the 38th IAHR World Congress
September 1-6, 2019, Panama City, Panama
doi:10.3850/38WC092019-0528

COMPARISON OF CALCULATION METHODS FOR HEAD LOSSES IN MULTIPORT DIFFUSER OUTFALL DESIGN

DR. TOBIAS BLENINGER[1], FRANK OPILA[2], DR. ROBERT L. DONEKER[3], ADITHYA S. RAMACHANDRAN[4]

1. Prof. Dr.-Ing., Departamento de Engenharia Ambiental, Universidade Federal do Paraná, Curitiba, Brazil. ***email****: bleninger@ufpr.br*

2. Environmental Engineer, MixZon Inc., Portland, OR, USA. ***email****: franko@mixzon.com*

3. President, MixZon Inc., Portland, OR, USA. ***email****: doneker@mixzon.com*

4. Sr. Software Systems and Modeling Engineer, MixZon Inc., Portland, OR, USA. ***email****: adiram@mixzon.com*

ABSTRACT

Multiport diffusers are efficient components of outfall infrastructure for river or submarine effluent disposal. They are utilized to avoid pollutant accumulation and to provide rapid dispersion of the effluent. A multiport diffuser design program has been developed to improve the current level of design technology for such installations.

Good multiport diffuser outfall design must address the hydraulics outside and inside the diffuser. Several methodologies for the analysis of the diffuser internal hydraulics have been adopted by various authors. These include a 1-D pipe flow port-to-port analysis (Fischer et al., 1979, Wood et al., 1993). Fischer et al. (1979) define a bulk loss coefficient C_d to estimate the loss from simple riser geometries for sharp-edged and bell-mouthed ports. These discharge coefficients are empirical and do not consider diameter ratios and flow separation effects in detail.

Other methodologies include a fictitious porous conduit and a 2 or 3-D field Eulerian grid for every point of the diffuser (Shannon, 2002, Mort, 1989). These two have the advantage that unsteady, stratified flow (i.e. saltwater intrusion) calculations are easier to implement than the port-to-port analysis. However, they do not specify appropriate local loss formulations for common or complex diffuser and port geometries.

The present approach involves calculating the numerous local losses due to expansions, contractions, bends, friction, etc., as well as additional flow forcing due to density differences. This approach accounts for more complex geometries. The CorHyd model presented here computes the flow distribution along the diffuser and the related pressure losses in the pipe system. It considers different pipe materials and geometric configurations and overcomes restrictions of previous diffuser programs by considering flexible geometry specifications, high risers, and variable area orifices, all with the automatic definition of loss coefficients. CorHyd employs this approach using a steady state pipe flow analysis. It employs the continuity equation at each flow division and the work-energy equation along pipe segments, along with detailed loss calculations. The diffuser pressure upstream and downstream of each port is calculated.

Additional features regarding blocked ports, sensitivity analysis and performance evaluation for varying parameters ensure proper diffuser design and reduced costs for installation, operation and maintenance. Coupling with the regulatory mixing zone model, CORMIX, allows for outfall design, optimization, and computation of the subsequent mixing zone characteristics in the receiving waters.

Keywords: diffuser hydraulics, CorHyd, multiport diffuser design, CORMIX, mixing zone

1 INTRODUCTION

Multiport diffusers are efficient components of outfall infrastructure for river or submarine effluent disposal. They are utilized to avoid pollutant accumulation and provide rapid dispersion of the effluent. A multiport diffuser design program has been developed to improve the current level of design technology for such installations.

The program computes the flow distribution along the diffuser and the related pressure losses in the pipe system. It considers different pipe materials and geometric configurations. CorHyd allows flexible geometry specifications, high risers, and variable area orifices, all with the automatic definition of loss coefficients. Additional features such as blocked ports, sensitivity analysis, and performance evaluation for varying parameters ensure proper diffuser design and operation and reduced costs for installation, operation, and maintenance. A coupling to the near-field mixing expert-system CORMIX can be used to model the mixing characteristics in the receiving waters.

2 OVERVIEW OF SOLVING METHODOLOGIES

2.1 Multiport Diffuser Design Goals

Good multiport diffuser outfall design must address the hydraulics outside and inside the diffuser. Multiport diffuser hydraulics design objectives are:

1. Increased initial mixing.
2. Uniform or optimized discharge distribution along the diffuser in order to meet dilution requirements.
3. Avoidance of wake flows and near field attachments.
4. Minimized investment, operation, and maintenance costs using simple, flow optimized manifold geometries with small pressure losses.
5. Prevention of off-design operational problems like particle deposition or salt-water intrusion in the pipe system. This requires full flowing pipe sections and reasonably high velocities.
6. Robustness to unsteady hydraulic conditions in order to reach steady flow conditions after short purging during start-up periods, optimized intermittent pumping cycles, and considerations of wave-induced circulations and transients.

2.2 Bulk Loss Coefficient

Several methodologies for the analysis of the diffuser internal hydraulics have been adopted by various authors. These include a 1-D pipe flow port-to-port analysis (Fischer et al.,1979, Wood et al., 1993). Fischer et al. (1979) define a bulk loss coefficient C_d to estimate the loss from simple riser geometries for sharp-edged and bell-mouthed ports. These discharge coefficients are empirically based on discharge coefficient curves developed earlier (Rawn et al., 1960). Wu et al. (2013) improved upon the constants in the equations using the original curves (Rawn et al., 1960).

For sharp-edged ports,

$$C_D = 0.91 - 0.94\frac{V_n^2}{2gE_n} \qquad [1]$$

in addition, for bell-mouthed ports:

$$C_D = 0.61 - 0.635\frac{V_n^2}{2gE_n} \qquad [2]$$

While the discharge coefficients are useful for many diffuser applications, they do not consider diameter ratios and flow separation effects in detail.

2.3 Solving Methodologies

Mort (1989) developed a one dimensional (1-D) finite difference model. Shannon (2002) utilized a 2-D or 3-D field Eulerian grid for every point of the diffuser. These methodologies have the advantage that unsteady, stratified flow (i.e. saltwater intrusion) calculations are easier to implement than the port-to-port analysis. However, they do not specify appropriate local loss formulations for common or complex diffuser and port geometries.

Conflicting design objectives require compromises that are not sufficiently resolved in many cases (Bleninger et al., 2004). Existing diffuser hydraulic programs (Fischer et al., 1979, implemented in code PLUMEHYD; and Wood et al., 1993, implemented as DIFF) work well for simple diffusers with uniform geometries. However, they consider only short risers with negligible friction and local pressure losses. More complex designs may require long risers (like in deep-tunneled outfalls) with significant frictional and local pressure losses, Y-shaped diffusers, complex port/riser configurations including rosette-like arrangements, multiple ports on one riser, duckbill valves, or other port pressure losses. Available design rules regarding velocity ratios (Fischer et al., 1979) or loss ratios (Weitbrecht et al., 2002) for diffuser sections and downstream ports are only applicable for simple and uniform geometries (i.e. no geometrical changes along the diffuser). In some cases, unnecessarily conservative designs are produced, because in the actual diffuser installation, the velocities and pressure losses along the diffuser line may change in an irregular manner.

3 CORHYD SIMULATION MODEL

CorHyd is a computer model that simulates flow characteristics for multiport diffuser design. It includes loss calculations for complex geometries, as well as additional flow forcing due to density differences.

3.1 CorHyd Governing Equations

Manifold hydraulics is characterized especially by several flow separations, where local pressure losses not only depend on geometrical relations but also on the discharge ratios. For wastewater diffuser manifolds, discharge ratios furthermore are influenced by the pressure difference between the effluent and the ambient.

Figure 1 shows the definition diagram with the geometries defined as sets of diffuser pipe segment locations x, y, z, riser/port segment geometries

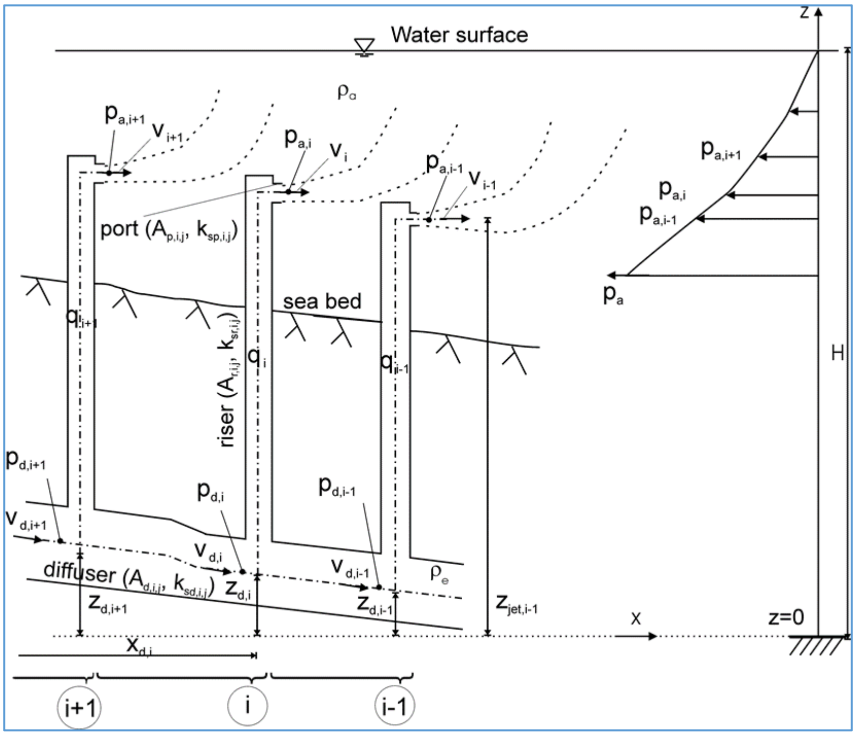

Figure 1 Schematic for the port-to-port analysis

The governing equations are continuity equations at each flow division

$$Q_1 = Q_2 \Leftrightarrow V_1 A_1 = V_2 A_2 \qquad [3]$$

Where, Q denotes the flowrate in one pipe section and the work-energy equation along pipe segments with a constant or known flowrate

$$\frac{V_1^2}{2g} + z_1 + \frac{p_1}{\gamma_e} = \frac{V_2^2}{2g} + z_2 + \frac{p_2}{\gamma_e} + h_\ell \qquad [4]$$

The definition of h_ℓ can be seen as the turbulence closure for pipe flow hydraulics. The dissipation of turbulent kinetic energy causes heat losses in the system. This energy loss is compensated by decreasing pressure. Pressure losses in pipe flows are classified according to the source of turbulence production in continuous pressure losses due to wall friction and local pressure losses due to geometrical changes. Wall friction causes cross-sectional flow non-uniformities, whereas geometrical changes cause streamwise flow non-uniformities.

3.2 Friction Losses

The friction coefficient λ describes the relative influence of fluid viscosity and wall roughness by the dimensionless parameters k_s/D with the equivalent sand roughness k_s (after Nikuradse) and the Reynolds number *Re*. For laminar flows, it is possible to derive λ analytically to $\lambda = 64/Re$ (for $Re < 2000$). However, for turbulent flows only empirical values from experimental studies exist. For computer applications, an explicit formulation after Swamee and Jain (1976) is used λ:

$$\lambda = \frac{0.25}{\left(\lg\left(\frac{k_s}{3.7D} + \frac{5.74}{Re^{0.9}}\right)\right)^2}, \qquad [5]$$

which is valid for $10^{-6} < k_s/D < 10^{-2}$ and $10^{-3} < Re < 10^8$

3.3 Local Losses

Streamwise changes in the pipe system cause non-uniform velocities in the system. Even gradual changes may cause internal flow detachment processes, reverse currents in dead zones, locally increased accelerations or decelerations, and overall increased turbulence. This increase is associated with additional energy and thus pressure losses. Typical geometrical differences occurring between one cross-sectional area and the adjacent area include inlets at the headworks, orifices at the outlet ports, pipe diameter expansions, contractions, bends, or combining and dividing flows.

Local pressure losses $h_{\ell,\ell}$ are generally parameterized as:

$$h_{\ell,\ell} = \zeta \frac{V^2}{2g} \qquad [6]$$

where ζ denotes the dimensionless loss coefficient.

ζ depends on the geometrical configuration (diameter ratios, angles, and radii of bends, gradual or immediate (rounded or sharp-edged) changes), and the Reynolds number.

There are numerous publications defining local loss coefficients ζ for a large number of different geometries under different flow conditions. Comparisons among these publications show considerable discrepancies even for simple geometries. More accurate works stem from Idelchik (1986) and Miller (1990), which are used in CorHyd. A few analytical solutions (e.g. for sudden expansions) are also included.

Additional optional pressure losses, which are not automatically calculated (e.g. obstructions due to valves or monitoring instruments), can be added manually. Loss coefficients can be combined to reduce the amount of computation. The bulk loss coefficient is the sum of all coefficients, providing that the reference velocity is the same.

3.4 Modeling Assumptions

CorHyd assumes that the entire pipe system flows full under all conditions. A two-phase flow due to air entrance at the inlet or a stratified flow due to saltwater intrusion at the outlets is not considered. However, CorHyd allows hydraulic conditions to be specified to determine at what flowrate a full flowing system can be assured. Air entrance at the inlet is avoided by keeping the top pipe inverted under the minimum sea level or using backpressure valves or deaeration chambers. Saltwater intrusion can be avoided by keeping the port densimetric Froude number larger than unity (Wilkinson, 1988):

$$\mathrm{F_p} = \frac{V_p}{\sqrt{g\frac{\Delta\rho}{\rho}D_p}} > 1 \qquad [7]$$

where V_p and D_p denote the port exit velocity and the port diameter, respectively, and $g\Delta\rho/\rho = g'$ is the reduced gravity with density difference $\Delta\rho$ between the effluent and the ambient water at the port orifice. This condition can be achieved by designing either small enough port diameters or using variable area orifices (duckbill valves).

4 MULTIPORT DIFFUSER DESIGN USING CORMIX AND CORHYD

4.1 CORMIX Hydrodynamic Mixing Zone Model

CORMIX is a USEPA-supported mixing zone model and decision support system for environmental impact assessment of regulatory mixing zones (RMZ) resulting from continuous point source discharges. The system emphasizes the role of boundary interaction to predict steady-state mixing behavior and plume geometry. CORMIX is a software modeling system for the analysis, prediction, and design of aqueous toxic or conventional pollutant discharges into diverse water bodies. The major emphasis of the system is on the geometry and dilution characteristics of the initial mixing zone, including compliance with regulatory constraints as well as predicting the behavior of the discharge plume at larger distances. The CORMIX regulatory mixing zone model features the following:

- Accounts for discharge plume “stability” and the possibility of local full vertical mixing and recirculation of the effluent. The assessment of near-field discharge stability, i.e. “stable” or “unstable” discharge conditions, is a key aspect of effluent near-field dilution analyses and mixing zone modeling, Stable discharges occur in conditions of strong buoyancy, weak momentum, and deep water. Unstable discharges occur when recirculation phenomena develop in the discharge vicinity (**Figure 2**, Left).

- Explicitly simulates plume-boundary interaction process (vertical and lateral), near-field dynamic Wake and Coanda attachments which are very important processes to characterize initial mixing behavior (**Figure 2**, Right).
- Simulates density current mixing with the possibility of upstream intruding plumes which are important components of the physical process for dense effluent discharges.
- Predicts near-field and far-field plume trajectory and dilutions for various source configurations.
- Has been validated for a wide range of discharge and mixing zone conditions.

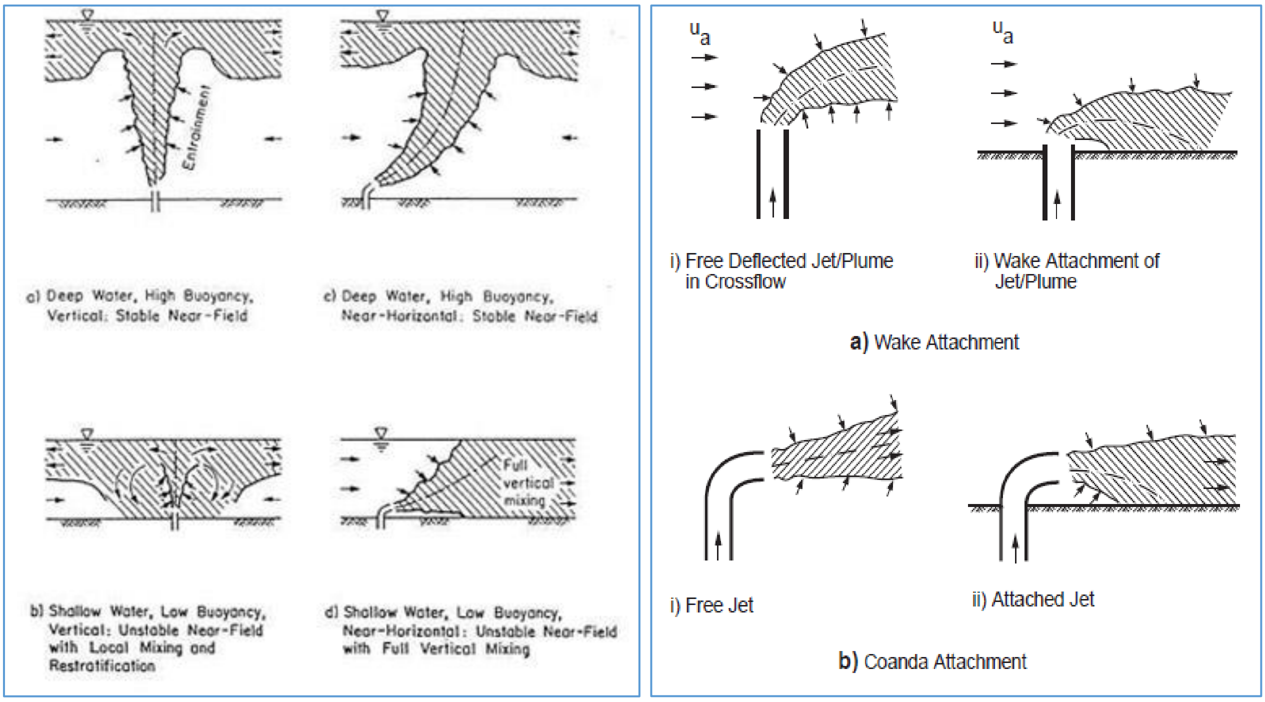

Figure 2 (Left) Examples of near-field discharge stability; (Right) Examples of wake (crossflow induced) attachment and Coanda attachment conditions for jets discharging near bottom boundaries.

4.2 Diffuser Design Types and Performance Features

CORMIX can analyze discharges from the three major multiport diffuser types used in common engineering practice. These diffuser types are illustrated in **Figure 3**:

1. **Unidirectional** diffuser where all ports/nozzles point to one side of the diffuser line and are oriented more or less normally to the diffuser line and more or less horizontally (**Figure 3**a).
2. **Staged** diffuser where all ports point in one direction generally following the diffuser line with small deviations to either side of the diffuser line and are oriented more or less horizontally (**Figure 3**b).
3. **Alternating** diffuser where ports do not point in a nearly single horizontal direction (**Figure 3**c). This diffuser type produces no net horizontal momentum flux. In this case, the ports may point more or less horizontally in an alternating fashion to both sides of the diffuser line or they may point upward, more or less vertically.

For multiport diffusers, CORMIX assumes uniform discharge conditions along the diffuser line. This includes the local ambient receiving water depth (HD) and discharge parameters such as port size, port spacing, and discharge per port, etc. Mean values should be used to specify variable diffuser geometry when it occurs.

4.3 CORMIX Multiport Diffuser Design Criteria

CORMIX simulation results can be used to evaluate the performance of a multiport diffuser design. Dilutions reported at the end of a regulatory mixing zone can be compared against dilution requirements needed to meet ambient water quality standards. The individual port plumes should merge in the near-field. If they do not merge at the RMZ, running CORMIX for a single port may be needed to estimate dilutions. CORMIX may issue warnings for near-field dynamic attachments, including Wake and Coanda attachments. Diffuser performance in off-design conditions should also be considered.

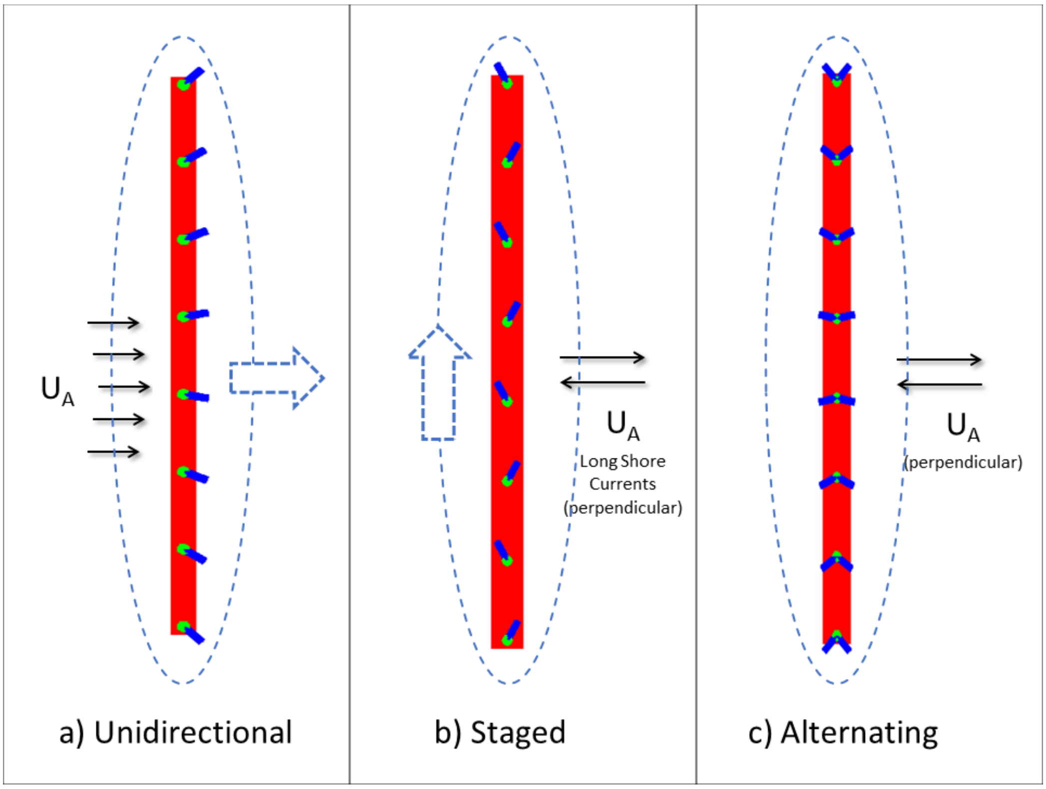

Figure 3 Multiport diffuser geometry types used in CORMIX

4.4 Hydraulic Diffuser Design and Optimization

The first design flow rate should be the maximum foreseen at the end of design life. Diffusers are generally operated under varying flow conditions due to diurnal or seasonal changes. CorHyd includes a batch mode for diffuser analysis of varying effluent flow or varying total head respectively in combination with varying ambient water level elevations (still steady state).

Varying inflows affect the discharge distribution for non-horizontal diffusers. Under low-discharge conditions, diffusers are furthermore confronted with issues of sediment deposition and/or intrusion of seawater. Seawater intrusion can seldom be avoided for all discharges. Duckbill valves and small diameter pipes prevent those problems, but may require additional pumping costs or higher headworks storage buildings. Intrusion can be prevented if the port densimetric Froude number is greater than one (Wilkinson, 1988). Particle deposition can be avoided by achieving pipe velocities greater than critical velocity (≈ 0.5 m/s) at least once a day.

CorHyd simulation results can be used to evaluate the hydraulic diffuser design. Pipe velocities in diffuser sections and risers should be within reasonable ranges, generally $0.5 \text{ m/s} < V < 5 \text{ m/s}$. Port and jet velocities should generally be in the range $0.5 \text{ m/s} < V < 12 \text{ m/s}$. Pipe diameters may need to be adjusted to fit in the desired ranges.

The necessary total head or final flow should be in the desired order of magnitude, otherwise, velocities and/or locations of high-pressure losses should be reduced. Pipe diameters can be increased or geometries can be simplified to reduce the total head required.

Uniform flow distribution along the diffuser should be checked. In general, the discharge from each port q_i should be within 10% of the average port discharge q_{mean} for at least a majority of port/riser configurations. Riser and port diameters can be adjusted to achieve this. Alternatively, duckbill valves can be utilized to obtain more consistent jet velocities. Duckbill valves change their effective open port area related to the pressure difference between inside and outside the valve. They also avoid debris and salt-water intrusion during low flow periods and allow high discharges during peak flow periods (Lee et al., 1998). CorHyd incorporates characteristic curves (headloss, jet velocity, effective open area) for a number of duckbill valves for pipe diameters from 1.3 to 152.4 cm (0.5 to 60 inches).

Figure 4 presents an example of the flow distribution and discharge deviations among the ports of a 30.5 m long multiport diffuser design with 20 ports. This design features three pipe sections of increasingly smaller diameters. Note that the discharges from the individual ports q_i are all within 10% of q_{mean}. Thus, the CORMIX assumption of uniform discharge along the diffuser line is satisfied.

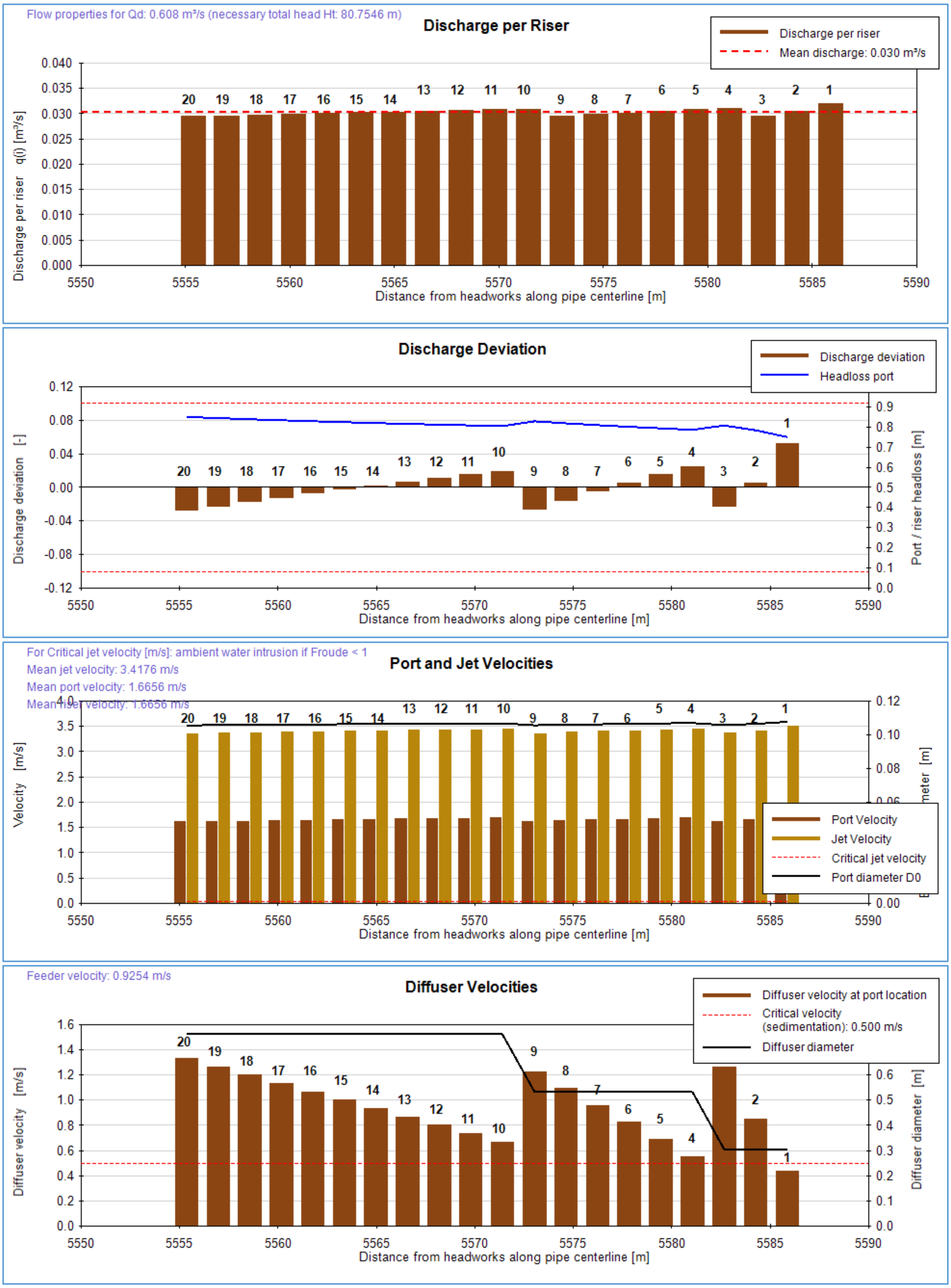

Figure 4 Flow distribution and discharge deviations among the ports of a multiport diffuser design.

4.5 CORMIX / CorHyd Iterative Design

Effective multiport diffuser design can be accomplished using both CORMIX and CorHyd. An iterative approach is used to achieve both dilution and hydraulic requirements. CORMIX can be used to design a multiport diffuser as a line source that meets the required dilution. Next, the design can be refined to meet hydraulic requirements using CorHyd. CorHyd is also used to validate the uniform flow distribution assumption. The revised design may require changes to CORMIX input parameters. CORMIX is then rerun. This process proceeds iteratively until a satisfactory design is achieved. **Figure 5** presents a flow diagram of the iterative process.

The iterative approach was used to design a multiport diffuser for discharge into a river in the state of Oregon, USA. The river has a 1Q10 low flow of 96 m^3/s (3400 cfs). After applying various design options to both CORMIX and CorHyd, a 30.5 m long unidirectional, fanned diffuser with 20 risers and 0.15 m duckbill ports was selected. The unidirectional design (**Figure 3**) discharges effluent in the downstream direction. The port angles relative to the current are “fanned” to limit the effects of a near-field acceleration zone. Diffuser fanning also enhances the cross-section available for near-field plume entrainment and mixing. The design maintains an optimal exit velocity of 3.5 m/s. Adequate mixing is accomplished with bulk dilutions of 150 for an RMZ of 91.4 m from discharge. **Figure 4** presents the results of the CorHyd hydraulics simulation.

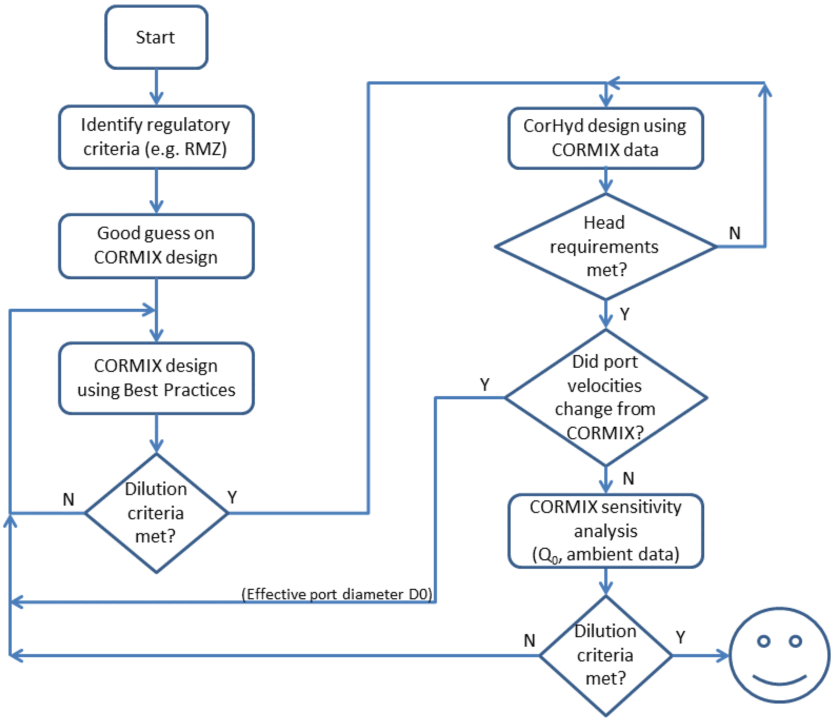

Figure 5 Flow diagram for CORMIX / CorHyd iterative design

5 CONCLUSIONS

The CorHyd hydraulics model simulates flow characteristics for multiport diffuser design. It includes loss calculations for complex geometries, as well as additional flow forcing due to density differences. The model incorporates friction and local pressure losses for risers of various lengths, including long risers (like in deep-tunneled outfalls) with significant frictional and local pressure losses. CorHyd is capable of simulating Y-shaped diffusers, complex port/riser configurations like rosette-like arrangements, multiple ports on one riser, duckbill valves or other port pressure losses.

Effective multiport diffuser design can be accomplished using both CORMIX and CorHyd. An iterative approach is used to achieve both dilution and hydraulic requirements.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Gerhard H. Jirka for his work on both CORMIX and CorHyd at Cornell University and the Institute for Hydromechanics, University Karlsruhe.

REFERENCES

Bleninger T., Avanzini, C.A., and Jirka, G.H. (2004). “Hydraulic and technical evaluation of single diameter diffusers with flow rate control through calibrated, replaceable port exits”, Proc. Int. Conf. Marine Waste Water Discharges and Marine Environment, Catania, Italy.

Doneker, R.L. and G.H. Jirka. (1990). CORMIX1: An Expert System for Mixing Zone Analysis of Conventional and Toxic Single Port Aquatic Discharges. Athens, GA, USEPA.

Doneker, R.L. and G.H. Jirka. (2007). CORMIX User Manual, USEPA**:** EPA-823-K-07-001.

Fischer, H.B., List, E.J., Koh, R.C.Y., Imberger, J. and Brooks, N.H., 1979, “*Mixing in Inland and Coastal Waters*", Academic Press, New York.

French, J., May. (1972). “Internal hydraulics of multiport diffusers”, *Journal WPCF*, Vol. 44, No. 5, p. 782pp.

Idelchik, I.E. (1986). “*Handbook of Hydraulic Resistance*”, Springer-Verlag, Berlin.

Lee J.H.W., Karandikar J., and Horton, P.R. (1998). “Hydraulics of DuckBill Elastomer Check Valves”, Journal of Hydraulic Engineering, April.

Miller, D.S. (1990). “Internal Flow Systems”, BHRA, Cranfield.

Mort, R. B. (1989). “The Effects of wave action on long sea outfalls”, *Ph.D. thesis*, University of Liverpool.

Rawn, A.M., Bowerman, F.R. and Brooks, N.H. 1960. Diffusers for Disposal of Sewage in Sea Water. Journal of the Sanitary Engineering Division, ASCE, 86(SA2): 65-105.

Shannon, N.R., Mackinnon, P.A., Hamill, G.A., and Doyle, B.M. (2002). “Collection of Data to Validate a Numerical Model of Wave Induced Intrusion in a Marine Outfall”, WIT Transactions on Engineering Sciences (ISSN: 1743-3533).

Weitbrecht V., Lehmann D. and Richter A. (2002). “Flow distribution in solar collectors with laminar flow conditions”, *Solar Energy*, Vol. 73, No. 6, 2002.

Wilkinson, D. L. (1988). “Avoidance of seawater intrusion into ports of ocean outfalls”, *Journal of Hydraulic Engineering*, Vol. 114, No. 2, February, 1988

Wood, I.R.; Bell R.G.; Wilkinson D.L. (1993). “Ocean Disposal of Wastewater”, World Scientific, Singapore.

Wu, S. and Mukto, M. (2013). Head loss calculation in multiport diffuser pipes. *21st Canadian Hydrotechnical Conference,* Banff, Alberta, May 14-17, 2013.