NEW MODELING TECHNIQUES IN THE HYDRAULICS LABORATORY OF THE WATER AND POWER RESOURCES SERVICE (SUBJECT D.C.)

D. L. KING

SUBMITTED FOR XIXTH CONGRESS OF IAHR
NEW DALHI, INDIA, FEBRUARY 1981

PAP - 400

INTERNATIONAL ASSOCIATION FOR
HYDRAULIC RESEARCH

NEW MODELING TECHNIQUES IN THE HYDRAULICS LABORATORY OF THE WATER AND POWER RESOURCES SERVICE (Subject D.C.)

by

Danny L. King
Chief, Hydraulics Branch
Water and Power Resources Service
Denver, Colorado, U.S.A.

SYNOPSIS

Three unique or unusual modeling techniques are described: (1) use of a loose-bed physical model to determine shear forces on river channel boundaries, (2) simulation of atmospheric flows with a stratified liquid model, and (3) a low-ambient-pressure chamber for research in cavitation.

RESUME

Trois techniques de modelage rares ou uniques sont décrites: (1) usage d'un modèle physgue à lit transportable pour determiner les forces de frottement sur les lits d'un fleuve, (2) simulation des écoulements atmosphérique avec un modèle liquide stratifié, et (3) une chambre de sous-pression pour recherche en cavitation.

Submitted for XIXth Congress of IAHR
New Delhi, India February, 1981

Movable bed models are often necessary in order to properly simulate physical processes occurring in a stream or canal. However, such models are expensive in comparison with fixed bed models. Besides hydraulic data, sediment data or computed estimates of transport, which are often difficult to obtain, are required to design and verify a model. Long runs to obtain equilibrium are often needed and changes are more difficult to make. Simulations of hydrographs may require determination of sediment time scales which are variable in terms of discharge.

Water and Power Resources Service experience in sediment modeling has concentrated on exclusion of sediment from canals since the early 1950's. More recent studies have involved diversion dams built in steep and fast flowing mountain streams where sediment is more coarse. A very recent study of the Columbia River downstream from Grand Coulee Dam, figure 1, was aimed at determining the shear or tractive force occurring on the bottom and side slopes of the river channel, for the purpose of providing data for design of protective material.

The early studies of sediment exclusion at diversion dams ordinarily used find uniform sand. The 50 percent sizes were usually scaled by settling velocity according to the Froude Law, which is important in determining when a particle will remain at rest or how long it will travel once suspended in the flow. These models were usually force-fed sediment to develop river slopes sufficient to move sediment at equilibrium rates estimated by sediment bedload computations. Slopes obtained were generally exaggerated because of friction differences between model and prototype. Recent models, concerned with movement of coarse sediment in steep mountain streams, have required that as much as possible of the particle distribution be represented by scaling settling velocity. Although there is generally less hydraulic and sediment data available for these smaller streams, prototype structures are small and the models are relatively large and do not need to be vertically distorted. Sediment sizes are larger and velocity scaling results in grain sizes in which the coarser particles nearly scale geometrically.

Shear or tractive force is directly related to friction and sediment entrainment. The energy equation for nonuniform flow, including work due to shear on the boundary, can be written as follows:

$$\tau_o dx/\gamma D = -VdV/g - dD + dh$$

Where τ_o is the boundary shear, γ is the specific weight of water, V is velocity, D is the depth of flow, x is distance along the bed in the direction of flow, g is the acceleration of gravity, and h is elevation of the bed. Introducing the Weisbach friction relationship and normalizing results in the dimensionless equation:

$$\tau_o{}^*dx^*/D^* \, (V_c^2/gx_c)(f/B) = -V^*dV^*(V_c^2/gx_c) - dD^* + dh^*$$

where the * denotes dimensionless variables, c denotes characteristic values, and f is the Weisbach friction coefficient which is a function of Reynolds number and relative roughness. The dimensionless parameters in parentheses are coefficients associated with the dimensionless variable groups in front of them. To apply the equation to both the model and prototype, the Froude number, V_c^2/gX_c, and f must have the same values in model and prototype. The model Froude number is set equal to the prototype requiring the use of analysis, field data, or best hydrologic estimates to check the friction coefficient, time, and transport scaling.

Roughness in the model as determined from analysis of bed material in the Comumbia River was verified as sufficiently correct to produce model staging within 2-1/3 percent, in terms of hydraulic radius. Since friction was accurately reproduced in the model, point velocities, velocity profiles, and secondary flows were expected to scale correctly provided that there were no major defects in geometric similitude. Since friction and velocity were properly scaled, it was expected that velocity profiles and; therefore, boundary shear would be correctly simulated within the precision of measuring velocity and differences in elevation in the model.

Analysis of the model scaling in terms of Shields', Gessler's, and Lane's entrainment functions indicated that when sediment simultaneously scaled by geometry and settling velocity, only particles greater than 13-mm in the model or about 1.6-m prototype would have general movement similarity (Gessler's 85 percent chance of moving). For Lane's critical tractive force (approximately Gessler's 50 percent chance of movement), movement would scale for model diameters equal to and greater than 8-mm model or about 1-m prototype. Gessler's 5-percent chance of moving would scale for particles equal to and greater than 6.5-mm model or about 0.8-m prototype. It was recommended that 0.05 probability of movement be used in the design of the protective material. Velocity distributions measured in the hydraulic model generally confirmed the results of a mathematical model.

Atmospheric Simulation Using a Stratified Liquid Model

In general, two types of scale models have been used for simulating atmospheric flows. The first and most common type uses air as the working fluid. Flow patterns, eddies, and diffusion characteristics can be studied. Because of difficulties in establishing density gradients, this type of model is usually limited to studies in the lower atmosphere. Air models are extremely useful in studying flows around buildings and diffusion from ground sources.

The second type of model, the subject of this paper, uses liquid as the working fluid. Studies using liquid models have included (1) wave motion and mixing at a density interface, (2) the progress of a density flow, and (3) the effect of a schematic mountain range upon successively higher zones of the atmosphere.

The model and studies described herein, figure 2, were designed to study orographic deflection of atmospheric flow. The distance scale studied was of the order of 2 to 20 km in nature. The model uses a stratified liquid to simulate the entire depth of the atmosphere and to study mean flow patterns and patterns of major circulations. Model flow patterns simulated represent flow patterns at only one instant during the time history of a storm. The model cannot exactly reproduce the flow conditions over mountains; however, the surface geometry and stratifications in the atmosphere introduce complexities in the flow patterns that are so great that even a qualitative reproduction of the flow field is beneficial. Topography is simulated with commercially available plastic relief maps having a 2:1 vertical distortion. The maps are rotated to change wind direction, density gradients can be varied, and the horizontal curvature of the free streamline trajectories can be reproduced.

Velocity and density gradient similitudes are developed in reference (1). A basic similitude equation is:

$$(\rho/\rho_o)^{(k-n)/n} = ce^{(S_o-S)/C_p}$$

which is referenced to conditions at ground level. Terms are defined as: ρ = density of fluid; ρ_o = fluid density at ground level; k = ratio of specific heats, C_p/C_v, in dry air (1.4); n = o for constant pressure, 1 for isothermal, k for isentropic, and ∞ for constant volume; S = entropy; S_o = entropy at ground level; C_p = specific heat at constant pressure; and C_v = specific heat at constant volume.

The right side of the equation is an expression which indicates the variation of entropy in the atmosphere with elevation. Entropy can be conveniently used to describe quantitatively the ability of the atmosphere to change its energy level. Since entropy is a property, changes in its value are independent of the actual way in which the change was accomplished. Thus, entropy can be used to describe a dry adiabatic atmosphere, a pseudomoist atmosphere, or any other atmosphere that may not even obey reversible gas laws. The equation can be regarded as the equation of state for the polytropic compressible atmosphere.

The Richardson number is ordinarily used to create models of the atmosphere, incorporating density variations and velocity gradients into one expression. Claus (2), in numerical studies of linear equations, discovered the method of essentially breaking the Richardson number criterion into two parts. He found that the flow of an incompressible fluid with stratification of density is similar to the flow of a compressible fluid with stratification of entropy if (1) the potential density distribution of the compressible fluid is identical with the potential density distribution of the incompressible fluid, and (2) the velocity profiles are nearly identical. Reference (1) demonstrates the applicability of these conditions. No effort was made to achieve Reynolds similarity in the model. Therefore, flow phenomena related to turbulence such as diffusivity, spreading of a plume, and boundary layer development were not expected to be determined from the model. Only mainstream or plume centerline flow was represented.

Because the maps used in the model study had a 2:1 vertical distortion in scale, it was necessary to verify model-prototype conformance by comparison with field data. This comparison was accomplished satisfactorily.

The motion of the atmosphere is described by Newton's second law which must be written relative to an inertial coordinate system. If the motion is viewed from a moving coordinate system, the basic relationships must account for this movement. Again, the details of this simulation are described in reference (1). The pressure field is defined by a pressure parameter (the Rossby number) and the Froude number.

In the conventional method of designing a model, careful attention is paid to properly reproducing the magnitudes of the dimensionless parameters. If this is done reasonably well, the model represents a solution of the normalized differential equation. That is, dimensionless velocities, flow directions, etc., in the model are identical with the corresponding dimensionless quantities in the atmosphere. Von Arx (3) demonstrates, for instance, how rapidly a model must be rotated about its axis to properly simulate tidal and Coriolis effects in marginal and small mediterranean seas of the earth. This rotation is necessary to maintain equality of the Rossby number in the model and nature.

The liquid simulation model was designed from a slightly different concept. Instead of reproducing the magnitudes of the dimensionless parameters, their resultant effect at a given elevation is reproduced. This is achieved by duplicating, in the model, the horizontal curvature and velocities observed in the field. This procedure is based upon the assumption that if the flow condition in the model (in dimensionless terms) reproduces a specified flow condition measured in the field at a given elevation, then the flow conditions at all elevations are properly simulated. Therefore, the liquid simulation model essentially duplicates the Coriolis and pressure fields implicitly. This technique can be viewed as one in which known flow conditions at one elevation (the upper atmosphere) are extrapolated into a region where the flow conditions are unknown (near the ground).

In summary, the boundary conditions which must be met in the model simulation are (1) correct variation of entropy with elevation, (2) correct free stream velocity, and (3) correct horizontal curvature of the free stream trajectory.

The simulation of the correct variation of entropy requires that the density at any elevation in the liquid relative to the ground level reference density varies as:

$$(\rho/\rho_o)_{liquid} = \left(\frac{T_o}{T} \cdot \frac{P_o}{P}\right)^{\frac{k-n}{n}}_{air}$$

where T = temperature, T_o = temperature at ground level and other terms are as defined above.

The correct free stream velocity is determined from the Froude parameter. The model velocity V_m is given by:

$$V_m = V_p(H_m/H_{atm})^{1/2}$$

where V_p = prototype velocity, H_m = height in model, and H_{atm} = height of atmosphere.

For a vertical scale of 1:125,000, the model velocity is given by $V_m = V_p/353.6$. For example with a field velocity of 25 m/s, the model velocity is 71 mm/s.

The establishment of the correct horizontal free stream trajectory is limited in the model. The horizontal radius of curvature R_m in the model varies between 0 and 2 m. The correlation between the model and the field values is based upon a simple geometric ratio: $R_m = R_p \cdot L_m/L_{atm}$. The ratio L_m/L_{atm} represents the relationship between horizontal lengths in the model and in the atmosphere. With a horizontal scale of 1:250,000, the model radius of curvature is $R_m = R_{atm}/250{,}000$. For example, with a free stream trajectory radius of curvature of 355 km in the atmosphere, the model value is 1.42 m.

The model includes a tank 150 mm deep with a diameter of 4000 mm. The depth of the liquid above sea level is about 88 mm, which roughly corresponds to the boundary between the atmosphere and the troposphere when a vertical scale of 1:125 000 is used. Motion in the model is generated by a rotating disk floating on the liquid. Production of potential flow is based on the rotational Reynolds number. If the disk is rotated at too large a rotational speed, turbulence develops just below the disk. This tends to destratify the gradient and destroys the potential flow field. Experimentally, the upper limit for the rotational Reynolds number of the disk with stratified fluids was determined to be 200,000. The maximum rotational speed of the disk is thus 90 seconds per revolution using the kinematic viscosity for fresh water. This condition puts an upper limit on the maximum upper air velocities in the atmosphere which can be simulated.

The original method of producing a density gradient consisted of floating a fresh water layer on top of a 150,000-mg/L saline solution. Molecular diffusion was then utilized in producing the desired density gradient. A computer program based on Fickian diffusion was written to predict the density gradient as a function of time. Although the density gradient produced in this manner does not duplicate exactly the potential density distribution of simple atmospheric profiles, the difference is small enough to be ignored for all practical purposes.

Dye injection tubes placed flush with the map surface simulated ground-based seeders. Aerial seeding was simulated by injection tubes which projected variable distances above the map surface. Since the dye cloud follows the plume trajectory, the cloud indicates what would have occurred downwind if aerial seeding had been performed anywhere along the path of the cloud. The height of the aerial seeders could be changed during a run by adjusting the height of the injection tube which passed through suitable seals in the tank floor.

A method to measure the density gradient was developed, based on the diffraction of light when it passes through a density gradient. By measuring the amount a light beam is diffracted from a known location, the gradient can be determined through a trial and error process involving a numerical integration. A computer program was developed for performing the numerical integration. The method is practically noninvasive and provides a continuous record with respect to elevation rather than discrete sample points. The atmospheric simulation model has been used to design cloud seeding operations and is presently being modified to include studies for siting wind farms by locating wind concentration areas.

Low-Ambient Pressure Chamber

A new low-ambient pressure chamber was recently operated for the first time in the Water and Power Resources Service hydraulics laboratory. Initial runs have included testing the ability of the chamber to hold the design vacuum of 15 kilopascals (0.15 atmosphere) and calibration of flow and pressure monitoring instruments. The facility will ultimately include a microprocessor controller to allow programed operation and acquisition of test data.

The chamber will be used in a long-term program of cavitation research. The test chamber, figure 3, has dimensions of approximately 4 by 1 by 3 m and will accommodate reasonably large models of structures and equipment, such as spillway chutes or tunnels, gates and valves, and stilling basins. Cavitation does not ordinarily occur in hydraulic models because of the relatively low velocities associated with the model scale. The low-ambient pressure chamber will allow scaling of the atmospheric pressure so that the location and intensity of the cavitation-vapor cloud will appear in the model. Thus corrective measures can be determined.

In addition to studies of structuresand equipment specified for Service projects, use of the chamber will include a 10-year program in cavitation research which will involve redefinition of criteria for surface irregularities, determination of the mechanisms of air entrainment and cavitation damage reduction due to aeration slots and offsets, study of pressure fluctuations in a confined jet, and investigation of the effects of cavitation on draft tube surging in hydraulic turbines. These are all aimed at solving current problems.

An early configuration to be installed in the low-ambient pressure chamber will be a simple offset into the flow. The findings of these tests will be compared with the results of early water tunnel tests. The first major research effort will be the investigation of surface irregularities. The facility has recently been used to determine the cavitation characteristics of a stepped spillway.

References

1. Falvey, H. T. and R. A. Dodge, "Atmospheric Simulation Using Stratified Liquid Models," Water and Power Resources Service Report REC-ERC-77-8, July, 1977.

2. Claus, A. J., "Large-Amplitude Motion of a Compressible Fluid in the Atmosphere," J. Fluid Mech., vol. 19, pp. 267-289, 1964.

3. Von Arx, W. S., "An Introduction to Physical Oceanography," Addison Wesley, 1964.

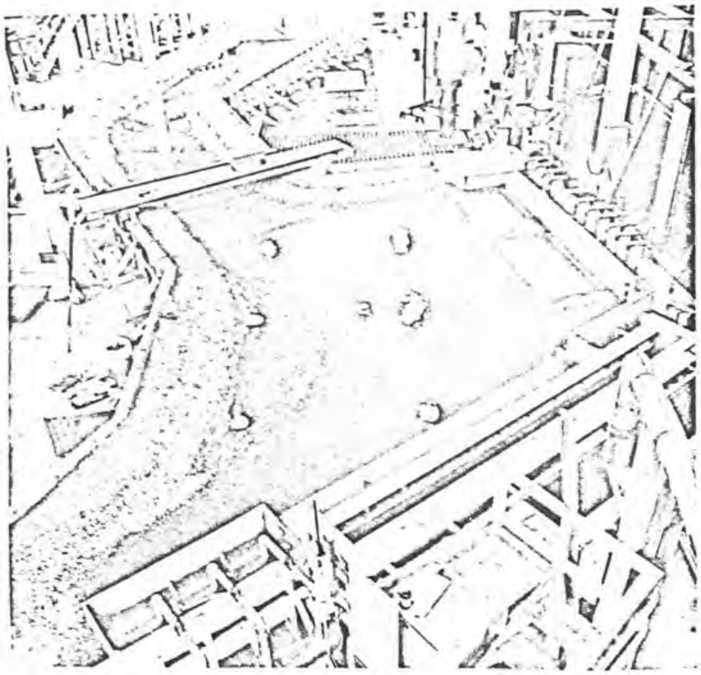

Figure 1. - 1:120 scale model of the Columbia River below Grand Coulee Dam

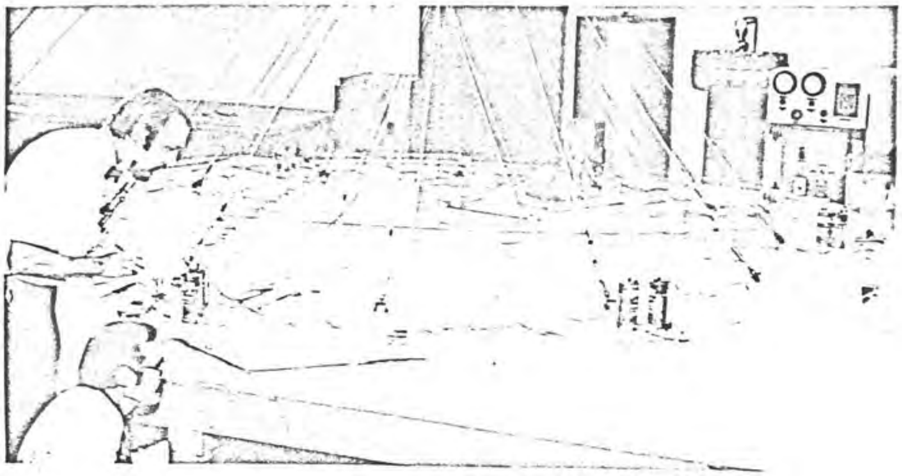

Figure 2. - Atmospheric simulation model

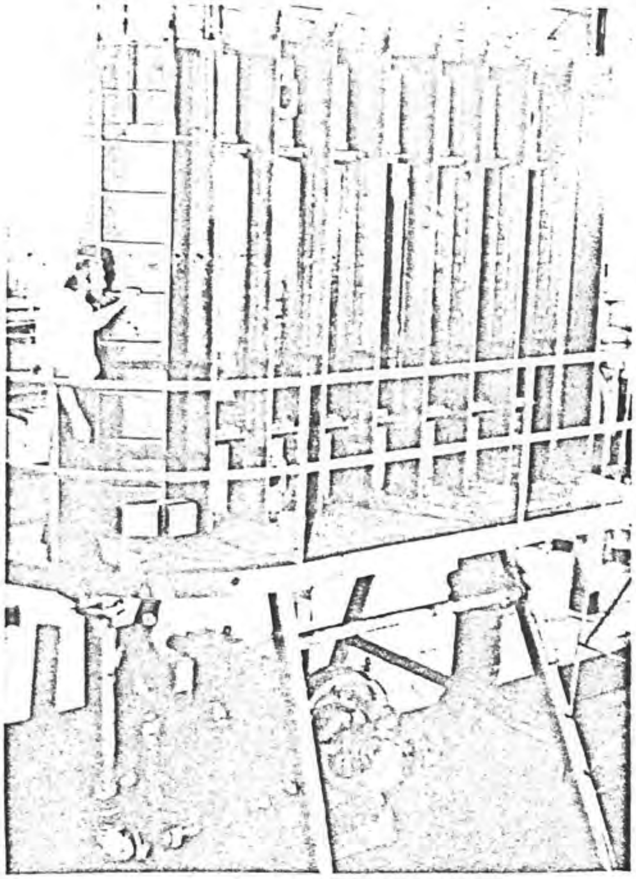

Figure 3. - Low-ambient pressure chamber

FIGURE CAPTIONS

Figure 1. - 1:120 scale model of the Columbia River below Grand Coulee Dam

1:120 modèle reduit de la fleuve Columbia sous le barrage Grand Coulee

Figure 2. - Atmospheric simulation model

Modèle de simulation de l'atmosphère

Figure 3. - Low-ambient pressure chamber

Une chambre de sous-pression