MASTER TN 02-004

Environment Canada Environnement Canada

Canada

ANALYSIS OF NORTH TORONTO CSO FACILITY OPERATION BY NUMERICAL AND PHYSICAL MODELLING

C. He, J. Marsalek, Q. Rochfort and
K. Krishnappan

NWRI Technical Note No. AEMRB-TN02-004

...L WATER ...INSTITUTE
...ATIONAL DE ...SUR LES EAUX

Technical Note

Analysis of North Toronto CSO Facility Operation by Numerical and Physical Modelling

C. He, J. Marsalek, Q. Rochfort, K. Krishnappan

Aquatic Ecosystem Management Research Branch
National Water Research Institute
867 Lakeshore Road
Burlington, Ontario L7R 4A6

NWRI TN # 02-004

August 2002

Analysis of North Toronto CSO Facility Operation by Numerical and Physical Modelling

C. He, J. Marsalek, Q. Rochfort, K. Krishnappan

PREAMBLE

The Remedial Action Plan for the Toronto Waterfront Area of Concern calls for the abatement of wet-weather pollution discharged along the waterfront to Lake Ontario. During these studies, a number of issues related to the hydraulics of the facility were raised, particularly with respect to: (a) the feasibility of increasing the hydraulic loading of the facility without bypassing, (b) flow conditioning in CSO storage tanks to induce improved settling, and (c) the feasibility of using the tank outflow weirs for accurate flow measurement. These issues were addressed by numerical and physical modelling of the facility conducted by the National Water Research Institute (NWRI), and described in the technical note that follows. Modelling results indicate that:

(a) The hydraulic loading of the facility could be increased 20 % by reducing the final effluent weir height by 0.4 m and streamlining two bends in the effluent channel,
(b) Flow patterns in the CSO tanks are non-uniform and significant improvements in these patterns could be achieved by means of retrofitted baffles, and
(c) It is unsuitable for flow measurement to use the CSO tank outflow weirs because they would remain submerged for most of inflow rates.

This technical note, as first phase study, represents a NWRI contribution to the City of Toronto led effort with respect to the development of remedial plans for Toronto Waterfront and restoration of beneficial water uses in this Area of Concern.

Analyse du fonctionnement de l'installation de TEU de Toronto-nord, par modélisation numérique et physique

C. He, J. Marsalek, Q. Rochfort, K. Krishnappan

INTRODUCTION

Le plan d'assainissement pour le secteur riverain préoccupant de Toronto nécessite la réduction de la pollution causée par le temps pluvieux, qui affecte le lac Ontario le long du secteur riverain. Lors des études effectuées, un certain nombre de questions ont été soulevées concernant les caractéristiques hydrauliques de l'installation, et notamment les suivantes : a) faisabilité de l'augmentation de la charge hydraulique de l'installation, sans passer par une dérivation; b) conditionnement du débit dans les réservoirs de stockage de TEU afin d'améliorer la décantation; c) faisabilité de l'utilisation de déversoirs sur les réservoirs pour mesurer de façon précise le débit. L'Institut national de recherche sur les eaux (INRE) a étudié ces questions par modélisation numérique et physique de l'installation. Les éléments techniques de l'étude sont décrits dans la note technique qui suit. Les résultats de la modélisation montrent que :

a) La charge hydraulique de l'installation pourrait être haussée de 20 % en réduisant de 0,4 m la hauteur du déversoir de l'effluent final et en adoucissant deux courbures dans la canalisation de l'effluent;
b) Les caractéristiques d'écoulement dans le réservoir de TEU ne sont pas uniformes et elles pourraient être considérablement améliorées à l'aide de l'addition de déflecteurs;
c) Les déversoirs du réservoir de TEU ne conviennent pas pour la mesure du débit car ils sont presque toujours submergés;

La présente note technique, en tant que première phase de l'étude, représente la contribution de l'INRE au travail entrepris par la Ville de Toronto pour mettre sur pied des plans d'assainissement destinés au secteur riverain de Toronto et restaurer les utilisations bénéfiques dans ce secteur préoccupant.

TABLE OF CONTENTS

1. INTRODUCTION

The City of Toronto expanded the North Toronto (NT) CSO settling tanks in 1991 (Fig. 1A) to improve the treatment efficiency of the undersized CSO and storm settling tanks constructed in 1924. In wet weather, CSOs escaping from an adjacent combined sewer enter the facility inlet channel, and continue through four connecting pipes into a distribution channel, and over inlet weirs into three storage tank cells. The weirs are arranged at three different levels, so that storage cells are filled sequentially, starting with the most downstream one. Overflow from the CSO tanks is conveyed by the effluent channel into a stormwater tank where it mixes with the discharge from a separate storm sewer. When the stormwater tank is filled, the wastewater overflows the final effluent weir to be blended with the secondary effluent from the North Toronto Water Pollution Control Plant (WPCP) before being discharged to the Don River. The pipes connecting the inlet and distribution channels limit the flow through the CSO storage tanks, and excess flows escape from the inflow channel over a bypass weir directly into the stormwater tank. After storms, the wastewater retained in the North Toronto CSO Facility (approximately 6,000 m^3) is pumped to the WPCP for treatment.

Depending on the type of the CSO event, this facility operates in two modes. For small volume CSO events ($V < \sim 6000\ m^3$), the CSO volume is fully contained in the facility, and eventually treated at the WPCP. For larger volume events ($V > 6000\ m^3$), the facility overflows and such overflows receive limited treatment (i.e., by dynamic settling) as they pass relatively quickly through the facility. The volume stored in the facility at the end of the event is pumped back to the WPCP for full treatment when plant's capacity allows it. When there are high inflow rates, there is some risk of bypassing of the CSO settling tanks. When the inflow exceeds about 4.0 m^3/s, the inlet channel overflows into the stormwater tank (see Fig. 1A).

During the last four years, the City of Toronto along with several partners have been studying the North Toronto CSO Facility. The overall objective was to improve the performance of the facility in the abatement of CSO pollution by implementing chemically aided settling of overflows at the facility. Towards that end, coagulants and/or flocculants were added to CSOs and induced quicker settling of suspended solids in flows passing through the facility. The treatment efficiency of this process depends on treated medium properties, chemical addition, and favourable hydraulic conditions, which would induce particle coalescence in the coagulation zone and quiescent settling downstream of this zone. The first two factors are being addressed in other studies; the hydraulics of the North Toronto CSO Facility is discussed in this report.

The performance of the existing NT CSO facility can be improved by reducing overflows from the inlet channel into the stormwater tank and increasing the flow through the storage tanks. Excessive CSO flows into the inflow channel cause this small tank to overflow into the secondary stormwater tank which eventually discharges over the final effluent weir. Such overflows bypass the CSO tanks, and after mixing with treated effluent from the North Toronto STP, are discharged into the Don River. Increasing the flow through the CSO settling tanks can reduce these overflows. However, the distribution of flows between the CSO tanks and inflow channel overflow is controlled by the elevations of several weirs, including the three tank outflow weirs, the inlet channel bypass weir, and the final effluent weir. For example, the elevation difference between the inlet tank bypass weir and the CSO tank outflow weirs controls how much flow passes through the three CSO tanks without overflows from the inlet channel. Also, the final effluent weir elevation controls (but not exclusively, as discussed later) the backwater in the effluent channel, which, in turn, will affect the maximum volume of flow that can be treated in the CSO tanks.

A survey of the NT facility showed that the elevation of the final effluent weir is about 2 cm higher than that of the CSO tank outflow weir. As one of the corrective measures at this facility, it was proposed to reduce the elevation (height) of the final effluent weir to increase the flow through the CSO cells (by reducing inlet channel overflows) and facilitate flow measurements at this facility. Before implementing such changes, several questions needed to be answered: (a) how much should be the final effluent weir lowered, (b) what are the effects of such a weir lowering with respect to flow increases, and, (c) is it sufficient just to reduce the weir height without implementing other measures.

Besides the facility capacity issues, the issues related to improving treatment (coagulant/flocculant mixing and settling) by creating favourable hydraulic conditions in the CSO tanks were also of interest. Such conditions include mainly flow patterns, turbulence, flow speed and boundary conditions. These could be even more important for the overall facility performance than just the issues of capacity. With respect to tank hydraulics, the pertinent issues include the flow patterns with respect to both chemical mixing and quiescent settling, and whether adverse conditions can be corrected by structural changes (e.g., installation of baffles). Both types of issues: flow capacities and hydraulics of CSO tanks, will be addressed in the following two sections.

Investigations of the CSO facility hydraulics were based on both numerical CFD and physical modelling. Compared to field investigations, such modelling has numerous advantages; it allows inexpensive analysis of various facility layouts and the effects of proposed structural modifications on facility operation and performance, and good control of conditions tested (e.g., the range and variation of flows). With rapidly increasing power of microcomputers, the use of numerical models as tools for analysis and improvement of performance of various wastewater treatment systems has been steadily growing (Kluck 1996, Shaw et al. 1997, and Pettersson 1997). On the other hand, modelling results always contain uncertainties arising from approximations of actual processes, and consequently, some field verification of modelling results is desirable.

2. NUMERICAL MODELLING

2.1 Numerical Model

The commercial CFD software (PHOENICS) was chosen for this study, recognizing that other products may be equally well applicable. PHOENICS is a general purpose computational fluid dynamics model capable of solving a variety of complex fluid flow problems. The mathematical description of the flow consists of the continuity equation and the three components of the Reynolds equations. The resulting differential equations are solved by the finite volume method, which can be applied in Cartesian, cylindrical-polar and curvilinear coordinate grids. A detailed description of the model can be found in Rosten and Spalding (1984).

To address the hydraulics of the CSO storage facility, it is essential to establish the water surface profile throughout the facility and its changes with time, for varying inflows. The PHOENICS code contains several numerical models developed for this purpose. A multi-phase model with structure mesh was chosen in this study, on the basis of its applicability, computer-running time, stability, and suitability for simulating particle transport (which will become important in future study phases). This method is referred to in the PHOENICS documentation as the "algebraic-slip" method (ASLP), which is embodied in the Advanced Multi-Phase Flow option of PHOENICS. In the literature, it is also known as the "drift-flux" method.

Fig. 1A shows schematization of the NT CSO facility used in the numerical model. The Cartesian coordinate system was adopted in this work. This schematization represents closely the actual facility, with main one simplification. The invert of sloping straight pipes (between the influent and distribution channels) should have been represented in the numerical model with a structural mesh by a step-wise line, which would have produced extra resistance to fluid flow through the pipes. To avoid this, and at the same time to preserve hydrodynamic principles, the interconnecting pipes were placed horizontally at the same invert elevation on the inlet CSO tank.

Two turbulence models were tested in this study. The first one was the well-known two-equation k-⇨ turbulence model, which was proposed for general turbulent flows and described, e.g., in Launder and Spalding (1974). The second model was the level turbulence model, which applies to channel and pipe flows. Due to the complex nature of the CSO facility studied, neither of these two turbulence models is ideally suited for this application, even though they are the two best applicable turbulence models in the PHOENICS package and should perform adequately. Testing results showed that the k-⇨ model results agreed with measurements slightly better than those from the level turbulence model, and consequently, the k-⇨ turbulence model was adopted in this study, in spite of longer computer running time.

The numerical accuracy of flow simulations increases with the number of cells in the grid, but so does the computation time. For structures with complex geometry, a very large number of cells may be needed, and the correspondingly large computer running time may become intolerable. Thus, some balance has to be struck, which is not trivial when simulating complex structures on a PC machine.

Also, the choice of the modelling grid is very important for numerical modelling. It not only directly affects computer running times and the accuracy of final results, but it also greatly affects model stability. Adjustments have to be made in various regions of the schematization to avoid sudden changes in cell sizes, by creating more cells in regions with important hydraulic features. In this study, 55 x 45 x 25 uneven fixed nodes were used in the X, Y, and Z directions, respectively, and the time step was 0.182 seconds, with 15 iteration sweeps. When testing this model set up, all results were found to be practically independent of the grid used and size of time step. These can therefore be regarded as true results of the mathematical model simulations. A typical run required about 24 hours on a powerful PC machine.

2.2 Model Verification

The PHOENICS model was verified in two ways; by checking mass conservation in numerical runs and by comparing simulated results to observations in a physical scale model. A physical scale model (1:11.6) of the North Toronto CSO facility was built in the hydraulics laboratory of the National Water Research Institute and used to validate and verify the PHOENICS numerical model. For simplicity, all results measured in the physical scale model and presented in this report were scaled up to the prototype scale. Details of the physical scale model and its other applications will be discussed in later sections.

A general 3D numerical simulating result of flow conditions in NT CSO facility is shown in Fig. 1B. The red colour represents water level at 5.5m and black arrows are flow vectors. Overall, it qualitatively shows a reasonable flow pattern in a multi-part connecting structure without obvious numerical noises. The quantitative behaviour of flow and water elevation at various locations will be closely examined as the following.

In numerical water mass conservation tests, a constant inflow of 5 m^3/s to the facility was simulated for 25 minutes to reach steady state, and it produced a simulated outflow at the end of the effluent channel of $4.97 m^3/s$, or 99.4% of the inflow. Various inflow rates and running times were also tested and produced similar results, with outflows agreeing with inflows within 1%.

In the second verification step, filling rates and water levels simulated at various locations were compared to physical model measurements. These tests were particularly important in this study, in which the numerical model was used to predict water levels throughout the entire system to find out how structural changes would increase the system capacity.

Fig. 2 shows a comparison of simulated and measured (in the physical model) water levels at various facility locations and water level changes with time. The locations are indicated in Fig. 1A as blue dots labelled 1 to 5. The flow rate applied in this test was 5 m^3/s and the water depths plotted in Fig. 2 were measured from the tank bottom at each reference point. Because of tank depth variations, all water surface levels are not identical, even after reaching steady state flow conditions.

At all locations shown in Fig. 2, simulated and observed (in the physical model) time-varying water surface profiles agreed quite well. Thus, the numerical model was capable of accurately predicting the filling time and water levels for both transient and steady-state flows in the whole system. It took about 18 minutes for the entire system to fill up (i.e., for an inflow of 5 m^3/s), as shown by all profiles reaching a constant elevation at that time. When water starts flowing into the CSO tanks, the water level at upstream locations stays almost constant for a while, until each tank has been filled up. So both the measured and modelled profiles display a step-wise shape. Thus, the curves in Fig. 2.1 can be characterized by four steps; the first three correspond to filling the CSO tanks 1, 2 and 3, in sequence, and the last step represents the attainment of the steady-state flow.

The flow velocities in the four distribution channel feed pipes were also monitored and results are shown in Fig. 3. In this figure, the measured velocities represent point velocities along the longitudinal axis of pipes, but because of limited spatial resolution of the numerical model, the modelled velocities represent average velocities for the entire pipe cross section. This explains why the modelled velocities are slightly smaller than the measured ones. Another numerical model approximation should also be mentioned - in the adopted Cartesian coordinate system, the round feed pipes were represented in the numerical model as square conduits. This approximation was accounted for in the modelled velocities displayed in Fig. 3. The total discharge through feed pipes was about 4.96 m^3/s, or 99.1% of the inflow rate.

All results presented above indicate that the general flow characteristics in the CSO facility, which are mainly controlled by hydrostatic pressure forces, are simulated well by the PHOENICS model. However, a question remains as to how well the model handles other flow characteristics, such as mixing in CSO tanks. Such processes are strongly affected by local flow properties and have to be investigated in more detail.

Velocity vectors in three main coordinates were measured in all three CSO tanks in the hydraulic model using an Acoustic Doppler Velocimeter (ADV). The measured velocities are shown in Fig. 4A, and the corresponding simulated velocities are shown in Fig. 4B. Blue arrows in the figure indicate the velocity scale, which represent a 0.3 m/s velocity scalar. Both numerical and measured results indicate that water enters the three CSO tanks unevenly distributed, with strong vertical sinking velocities.` Tank 1 conveys the largest discharge and Tank 3 the smallest. Ratios 1.13, 1.05 and 1.0 describe flow distribution through tanks T1, T2 and T3, respectively. This uneven distribution can be explained by the fact that at the outflow end, water level has a small slope decreasing from Tank 3 to Tank 1, because in the effluent channel, water flows in the direction from

Tank 3 to Tank 1. Even though an identical inflow was used in both numerical and physical models, the data in Fig. 4 indicate that measured velocities around outflow weir were smaller than the simulated ones. This discrepancy can be explained by difficulties with measuring velocities with a relatively large ADV probe in a narrow space between the scum baffle and the tank wall. It is believed that these limitations led to underestimation of those measured velocities.

When comparing the modelled and measured flow patterns in Fig. 4, it is apparent that the best agreement was found in Tank 3. Both patterns show that main flow advances along the outside wall, with a strong lateral rotation in the inflow zone. The numerical model also reproduced well the strong upward vertical velocity along the inside wall in the inflow zone of Tank 3. Even though the numerical simulations of flow patterns in Tanks 1 and 2 were not as good as for Tank 3, many main flow features were reproduced fairly well. Such features include strong currents along the western sidewall in Tanks 1 and 2. An explanation for low flows passing through certain parts of tanks was found from the measured velocities. In particular, low flows through inside sections of Tanks 1 and 3, and the middle section of Tank 2, are caused by very fast flows from the corresponding feed pipes, which impact on the tank front wall inside the distribution channel, and generate very strong turbulence and upward flow, blocking inflow into the tanks. As expected, the k-⇨ model adopted in PHOENICS was not well suited for this extreme situation, because the two governing equations employed in the k-⇨ model apply to flow turbulence with isotropic features. More complex, higher order of turbulence schemes would be needed to model this situation more accurately, but such models would require unacceptably long running times on a PC.

Recognizing the numerical model limitations, in is important to understand why better results were obtained for Tank 3. The reason is that flow in the distribution channel, at the inflow to Tank 3, has a better-defined pattern. After discharge from feed Pipe 4 (located close to the bottom of the distribution channel and at 90° to the three other pipes), the flow in the channel loses its momentum, is blocked by the discharge from Pipe 3, and forced upwards. In the upper layer, it turns back towards Pipe 4 and enters Tank 3 along the outside wall.

In an overall evaluation, the numerical model was capable of reproducing the measured flow characteristics reasonably well, especially for flow features controlled by the hydrostatic pressure. Comparisons of numerical results with observations in the scale model were helpful in explaining numerical results and should be conducted throughout the study. Thus, the PHOENICS model was found suitable for application to the study of performance of the CSO storage and treatment facility.

3. RESULTS AND DISCUSSION

3.1 Numerical Modelling of Flow Rates

In the initial series of PHOENICS runs, the final effluent weir height was set equal to zero (i.e., the weir was removed). Under such circumstances, the CSO tank outflow weir crest should be always above the water level in the effluent channel, and there should not be any backwater pressure influencing water levels in the CSO tanks. The numerical model was run with different inflow rates, until the water level in the inlet tank reached the crest of the bypass weir and the system reached steady state, as shown in Fig. 2-6. In this figure, the straight horizontal line indicates the elevation of the bypass weir, and the curve represents simulated water levels just upstream of the bypass weir. The results show that the maximum inflow rate to this treatment facility is about 5.46 m^3/s, which agrees with measurements in the physical model. However, in the actual facility, there is a noticeable effect of the final effluent weir, which causes backwater pressure and raises water level in the CSO tanks. Increased water levels in these tanks then cause overflow over the bypass weir, well before the maximum flow can be attained. With the existing final effluent weir, numerical simulations indicate a maximum (no bypass) flow rate of about 3.9 m^3/s. Thus, because of the facility configuration and the final effluent weir, the maximum treatment capacity of the existing facility is reduced by 29 %, and even at this reduced flow rate, the CSO tank outflow weir is submerged by about 0.465 m.

Reducing the height of the final effluent weir seems to be an obvious choice for increasing the treatment capacity of the system. However, if the final effluent weir is too low, the function of the stormwater tank will be diminished, and any flow overflowing the inlet bypass weir will discharge directly into the Don River without any treatment. In order to balance both requirements on increasing the treatment rate and preserving the storm tank volume, three different heights of the final effluent weir were considered. These heights corresponded to the present weir crest elevation of 87.08 m ASL, reduced by 20, 40 and 60 cm, respectively, and the results of such simulations are shown in Fig. 5. The reduction of 60 cm is the maximum that would be allowed by the regulatory agency.

The horizontal straight line in Fig. 5 denotes the elevation of the inlet tank bypass weir and the curves displayed represent water levels simulated in the inlet tank next to the bypass weir, for different final weir heights. In all simulations, the inflow rate was set at 5.46 m^3/s. It can be seen that all water surface profiles reach over the bypass weir crest after 18 minutes of inflow. This means that even if the final weir height is reduced by 60 cm, bypassing from the inlet channel would still occur. It was noted that the water level variation in the inlet channel was not very sensitive to changes in the final effluent weir height. For example, a 40 cm reduction in the weir height (i.e., moving from a 20 cm height reduction to a 60 cm reduction), reduced the water level in the inlet channel just by 10 cm. Since the water level difference between the inlet channel and the effluent channel should be similar for the same inflow, this indicates that the water level in the effluent channel also changes much less than the final effluent weir height. This finding prompted further investigations of flow hydraulic characteristics in the region between the CSO tank outflow weir and the final effluent weir.

Water levels in the effluent channel and in the stormwater tank were plotted in Fig. 6 for different heights of the final effluent weir. Recognizing that the reduction of the final effluent weir height by 20 cm was insufficient to significantly improve the system capacity, the discussion of numerical simulations focuses on reducing the final effluent weir by 40 and 60 cm, respectively. The plots in Fig. 6 represent water level profiles starting from the final effluent weir. In both panels of Fig. 6, curve A indicates the water level for the existing facility without any modifications, except for reducing the final effluent weir height. Water surface profiles indicate that additional hydraulic resistance generated by the right angle corner bends causes two water level peaks at the two 90° abrupt bends in the effluent channel. In order to overcome this obstruction, the flow level upstream of the bend has to rise to increase the hydrostatic pressure differential between the entry and exit sections of the bend. By reducing the final effluent weir height, the velocity in the effluent channel will increase, as will the hydraulic head needed to pass a higher flow through the bend. Consequently, the influence of reducing the final effluent weir height on water levels in the CSO tanks becomes much smaller, because of the obstruction effect of the two 90° abrupt bends. Therefore, further investigation is needed on minimizing the hydraulic resistance at the two bends.

The second (downstream) 90° bend, which is directly connected to the stormwater tank, can be fixed more easily either by widening the exit cross-section of the channel, or by cutting the downstream (left) sidewall of the bend at an angle larger than 45°. However, due to space limitations and structural considerations, the outside dimensions of the upstream 90° bend cannot be enlarged. Therefore, the only possible modifications of this bend have to be made inside the channel, e.g., by curving the inner corner and/or adding curved flow conditioning baffles. Computer simulations were used to explore both alternatives for improving the hydraulic efficiency of this bend.

The upstream 90° bend hydraulics was studied in two steps. The first step focused on simulation of the flow passing through the bend in a simplified setting, which included the bend itself and some upstream and downstream sections of the flow channel. In all simulations, the inflow rate was 5 m^3/s. The purpose of this step was to find out the most effective modification of the bend. With fewer structural components simulated and a higher node density, the accuracy of numerical simulations was increased and the computer running time was reduced. In total, six different bend geometries (A-F) were tested (see Fig. 7), and the corresponding water level profiles in the effluent channel are shown in Fig. 8.

Case A refers to a straight channel, which simulates flow conditions without any bend influence and is used as a reference for other test cases. It does not show any flow obstruction, as expected for a straight channel. Case B refers to a channel with 90° bend, without any modifications. The simulation results show that the water level upstream of the bend is about 13 cm higher than in case A, due to the corner effect. Cases C and D represent layouts, in which curved baffles were inserted into the channel outer corner, with radii of 1.5 and 1.0 m, respectively. Modelling produced somewhat surprising results; curved baffles placed in the outer corner barely affected water levels in the effluent channel. In fact, curves C and D in Fig. 8 are almost identical and quite similar to curve B, which represents the 90° bend. This lack of effectiveness can be attributed to the reduction of the flow cross-sectional area due to the placement of the curved baffle in the outer corner, even though water flow trajectories in the corner would become hydraulically more favourable. Similar tests with curved baffles (R = 0.5 m) were done in the physical model for different flow rates and also confirmed that these outer corner baffles barely influenced water levels.

Case E represents the layout, in which the geometry of the inner corner was rounded, with a radius of 0.5 m. This modification brought about a significant reduction in the water levels in the effluent channel, as shown in Fig. 8. It was felt that the effect of this improvement might have been slightly exaggerated in the numerical simulation. Physical model experiments confirmed that inner corner modifications were more effective in reducing local head losses than changes of the outer corner. The flow velocity fields at corners shown in Fig. 9 provide an explanation. For the inner corner of the 90° bend, illustrated in the upper part of Fig. 9, the flow will produce negative pressure downstream of the inner corner, and thereby generate an eddy, which then functions as a flow obstruction and reduces the effective flow width. After rounding the inner corner, the eddy caused by negative pressure disappeared, as shown in the middle panel of Fig. 9. Consequently, it is now easier for flow to pass through the bend, and a lower hydraulic head is needed to force water through the bend, as confirmed by the smaller difference between the crest and trough of water profile curves in Fig. 8. At the same time, the modelling results in the middle panel of Fig. 9 indicate that the velocity (and the corresponding head loss) along the outer corner is much smaller than that near the inner corner. This may explain why curved outer baffles do not contribute much to reducing the bend hydraulic resistance, even if the flow pattern has been improved (see the bottom panel in Fig. 9).

In the second step of this analysis, the knowledge about the hydraulics of 90° bends was used in numerical simulations of the actual facility with modified bends. The resulting water level profiles in the effluent channel are presented in Fig. 6 as curves B and C, in both panels. The water level in the effluent channel with an improved downstream bend (Curve B) was reduced significantly compared to Curve A, for both 40 and 60 cm reductions of the final effluent weir height. The water level in the region downstream of the first bend decreased more than that upstream of the bend. This can be explained by the fact that the first (upstream) bend acts as a bottleneck, which would require a greater hydraulic pressure build up to convey the increased discharge. This finding is also indicated by the increasing water level oscillations at the upstream bend in Fig. 6.

Curves C in Figs. 6A and 6B (both panels) show simulated water surface profiles for the case, in which the downstream bend was "opened" and the upstream bend inner corner was rounded. Water levels were reduced even further, but the upstream bend was still causing head-loss problems requiring more study.

After addressing the bend problems, further work focused on finding the relationships between the maximum attainable inflow rate (without bypassing), Q_{in}, and various heights of the final effluent weir, focusing on three cases: (a) no changes of the effluent channel (existing max Q_{in} = 3.9 m^3/s), (b) correcting the downstream bend only (max Q_{in} = 4.15 m^3/s), and (c) correcting both bends (max Q_{in} = 4.25 m^3/s). These three cases were then re-examined for reductions of the final effluent weir height by 20, 40 and 60 cm respectively, and the results were plotted in Fig. 10. Star symbols in Fig. 10 represent the maximum flow rates measured in the physical scale model. It can be seen in Fig. 10 that the numerically modelled maximum flow rates were slightly smaller (< 5%) than the measured ones, except for one point. However, these minor discrepancies were considered insignificant.

Thus, the numerical simulations indicate that by lowering the final effluent weir by 60 cm and modifying both 90° bends, the maximum inflow rate could be increased from 3.9 m^3/s to 5.12 m^3/s, which corresponds to about a 31% improvement of the system treatment capacity. An important finding can be made from Fig. 10 - the slopes of all water profiles decline with the decreasing final effluent weir height. In other words, the rate of improving the maximum inflow rate diminishes as the height of the final effluent weir is decreased. This also reduces the storage capacity of the stormwater tank. Depending on how much significance is assigned to maintaining the storage volume in the stormwater tank, it may be preferable to improve the system capacity by reducing the final effluent weir height by some intermediate value (e.g., 40 cm) and correcting the 90° bend problems at the same time.

The main reason for the fact that the magnitude of the final effluent weir height reduction can not be fully reflected in the water level drop in the effluent channel is due to the controlling influence of the flow in the effluent channel. This flow will affect how closely the water level in the stormwater tank will follow the height reduction of the final effluent weir. Finally, it is believed that the slope of the water profiles in Fig. 10 is also affected by the rating equation curve of the final effluent.

3.2 Numerical Modelling of Flow Mixing

A highly turbulent flow in the distribution channel enters the CSO tanks with a non-uniform velocity distribution across the three tank entrances, and at some locations, it also causes strong downward and rotational velocities in the CSO tanks. These flow features may produce velocity shear, flow turbulence and strong bottom shear stress in tanks and impair pollutant removal by settling. In fact, they may even cause resuspension of unconsolidated sediment deposits on the tank bottom. Consequently, numerical simulations were carried out to explore the possibility of modifying flow patterns in the

CSO tanks and thereby increase the effectiveness of the facility in removing pollutants through sedimentation.

One commonly used method for improving flow patterns is the use of flow conditioning baffles. An improved settling of suspended solids in the individual CSO tanks should be facilitated by a flow pattern, which would be characterized by a uniform velocity distribution in the inflow zone near the water surface. This pattern would produce less velocity shear, flow turbulence and bottom stress, and also, after particles move out of the surface flow layer by gravity force, most of them should continue to sink without being entrained by ambient flow. Numerical modelling was used to test various baffle designs, which would help approximate this ideal flow pattern. For brevity, only the most promising baffle layout is discussed in this section.

As illustrated in Fig. 11a flow conditioning baffles were placed at the entrance to all three CSO tanks. The main part of this arrangement was a 2 x 7 m (length x width) horizontal baffle attached to the inflow weir crest and extending into the tank. The purpose of this baffle is to force inflow to enter the tank horizontally along the water surface. On top of this baffle, there are two or three mounted vertical baffles, which divide the tank entrance width into several smaller channels and thereby make the flow distribution uniform in the lateral direction. Because the flow patterns in Tanks 1 and 3 were similar, the design of baffles in these two tanks is identical. The length of the three longitudinal baffles increases from the inner wall to the outer wall, as shown in Fig. 11A. Furthermore, the position of (and hence flow intercepted by) these baffles can be adjusted in the longitudinal direction, to control flow distribution. In Tank 2, flow enters along both sidewalls, and consequently the configuration of baffles is different. Two baffles with a specific angle of attack (see Fig. 11A) were used to restrict the inflow near both sidewalls and enhance the inflow through the middle section.

Tentatively, the following baffle designs were proposed:

Tanks 1 and 3:	3 baffles, 2 m high, and 4.2, 3.2, and 2.6 m long, respectively.
Tank 2:	2 baffles, 2 m high, and 2.13 m long, rotated by 20 degrees

The baffles of these dimensions produced promising results in simulation experiments, but they should be considered preliminary designs. The numerical modelling work is still in progress and further changes in baffle layout and dimensions are possible. Also, other methods of flow conditioning will be examined including suspended flexible cylinders serving to dissipate flow kinetic energy in the inlet zone.

Simulated flow patterns in the CSO tanks with the proposed baffles are shown in Fig. 11B. By comparing these patterns with those for the case without baffles (Fig. 4.2), it is obvious that flow conditions were significantly improved. In particular, at all tank entrances, inflow from the distribution channel is more evenly distributed in the lateral direction, the sinking and rotating flows were greatly reduced, and more uniform flow velocity fields were achieved in all the three tanks. A quantitative evaluation of this improvement, with respect to flow turbulence, shears and other characteristics, will be produced in the next study phase. Furthermore, this improvement should be also assessed in terms of improved settling of suspended solids in the facility. For this purpose, a particle transport model will be applied and model runs with and without baffles will be compared.

4. MEASUREMENTS IN THE PHYSICAL SCALE MODEL

4.1 Physical Model and Experimental Methods

In order to investigate the operation of the North Toronto CSO Facility and verify the numerical model, a physical model with a scale of 1:11.6 was built in the NWRI hydraulic laboratory (Fig. 12). The model consists of a water supply box, the CSO facility model, a water collection vessel, and a measuring weir box. The water supply box is fed from the laboratory water supply system. From the supply box, water enters the facility model, which comprises the inlet channel connected by four pipes to the distribution channel, three parallel CSO tanks, effluent channel, and finally the stormwater tank with the final effluent weir. Below this weir, water is collected in a vessel and piped to the measuring weir box. At the downstream end of the weir box, a calibrated sharp-crested triangular weir was installed and used to measure discharges.

This model was designed and operated according to the Froude similarity, which is applicable to flows driven by the forces of gravity. In such models, equal Froude numbers are maintained in model and prototype. From this condition, the following scaling ratios can be derived:
Length scale λ_l, in a geometrically similar model ($\lambda_x = \lambda_y = \lambda_z$) is defined as:

$$\lambda_l = l_m / l_p$$

where λ is the scaling ratio, l is a characteristic length, m refers to model, and p to prototype;

velocity scale $\lambda_v = \lambda_l^{0.5}$, and

discharge scale $\lambda_Q = \lambda_l^{2.5}$

where subscripts v and Q refer to velocity and discharge, respectively.

Experiments in the physical model were arranged in a similar way as in numerical modelling. Recognizing that the height of the final effluent weir was the focus of early investigations, the weir was designed so that its height could be easily changed. Similarly, allowances were made in the model structure to allow experimentation with 90° bends in the effluent channel. Specifically, the downstream side of the second bend (connected to the stormwater tank) was cut, so that it could be set at 90^0 (current condition) or 135^0 (entrance to tank opened by a further 45°) angles. At the first bend, the inside corner was rounded off with a 0.5 m radius (i.e., a prototype dimension), and the outer corner was rounded by inserting a curved baffle with the same radius.

To monitor water level changes with time at different locations, pressure sensors were mounted on the bottom of the inlet tank, distribution channel and the three CSO tanks.

The locations of these points of measurement are shown in Fig. 1 as blue dots. Instrument carriage rails were mounted on top of the CSO tank sidewalls and allowed carriage movement along the tanks. An ADV velocity probe was attached to the carriage by means of an adjustable instrument holder, moveable in the lateral direction. This arrangement allowed moving the ADV probe in three coordinate directions and measuring the three velocity components at all locations in the three CSO tanks. Furthermore, four Pitot tubes were inserted into the four interconnecting pipes between the influent and distribution channels, to measure the velocity along pipe axes. Overall, the model set up allowed to conduct experiments in a well-controlled environment.

4.2 Results and Discussion

Measurements of filling rates, water level changes in time at five different locations, velocities in the interconnecting pipes, and 3-D velocity fields in CSO tanks were discussed earlier in Section 3. Since the early efforts in this study focused on improving the treatment capacity of the CSO facility by lowering the final effluent weir, systematic experiments with weirs of various heights were carried out in the scale model. In particular, five different weir height reductions were tested in the range from 0 to 150 cm, with the maximum height bringing the weir crest to the same elevation as that of the CSO tank outflow weir. This arrangement agreed with the actual facility, where the final effluent weir crest elevation is about the same as that of the CSO tank outflow weir. The results of such experiments are summarized in Table 1. Thus, the values in the first column of Table 1 represent both the weir height reduction and the superelevation of the CSO tank outflow weir over the final effluent weir.

Table 1. CSO Facility Flow Rates vs. the Final Effluent Weir Height, Observed in the Physical Model

Height reduction of final effluent weir (cm)	Max. flow rates without modification (m^3/s)	Max. flow rates with changing second bend (m^3/s)	Max. flow rates with changing both bends (m^3/s)
0	4.08	4.29	4.38
20	4.52	4.72	4.84
40	4.81	5.01	5.10
60	5.00	5.10	5.14
150	**5.46**	*	*

* not measured

The 150 cm reduction corresponds to a complete removal of the final effluent weir and of its influence on the operation of the CSO tank outflow weir. Under such conditions, the CSO tank outflow weir will not be submerged under the maximum flow rate, as discussed earlier in the section on numerical model verification. The data listed in Table 1 were plotted in Fig. 10 as star dot symbols. A closer examination of data in the second table column indicates, that for the three weir height reductions of 20, 40 and 60 cm, the corresponding discharge increments are 0.48, 0.29 and 0.19 m^3/s, respectively. Thus, as discussed before, the relative increases in the maximum facility discharge diminish with decreasing weir height (see also Fig. 10). Explanations of this phenomenon were offered earlier in Section 3 on numerical modelling of discharge changes with respect to three facility elements, the effluent channel bends, the effluent channel itself, and the final effluent weir.

The results further show that the final effluent weir was the main cause for decreasing flow rate improvements with decreasing weir height. It was also noted that a final weir height reduction of 60 cm rather than 40 cm was rather ineffective; it increased the maximum flow rate by only 0.19 m^3/s, or 4 %. However, if the hydraulic problems at the two effluent channel bends were corrected, particularly at the downstream one, the difference in the flow rates for the two weir heights was only 0.04 m^3/s (0.8% of the influent rate) and 96 m^3 of storage in the stormwater tank would be preserved.

The significance of the second bend obstruction effect was also studied in the physical scale model, as shown in Columns 3 and 4 in Table 1. By opening up the second bend, the maximum flow rates increased by about 0.20 m^3/s (determined by subtracting Column 2 from Column 3), except for the 60 cm reduction. This 0.20 m^3/s improvement is comparable to the average flow rate increment of 0.32 m^3/s obtained by 20 cm height reductions of the final effluent weir. Thus, with respect to improving the facility discharge, hydraulic improvements of effluent channel 90° bends is about as effective as reductions in the final effluent weir height.

As discussed earlier, the first channel bend is more difficult to deal with because of space constraints. The earlier discussed corrective options (rounding off the inner corner edge and adding a curved baffle at outer corner) were also tested in the physical model. The measurements confirmed numerical results and indicated that the outer corner curved baffle barely affected the maximum flow rate. Thus, the maximum flow rate improvement shown in Column 4 of Table 1 was mainly due to the rounding of the inner corner edge.

4.3 Observations on Flow Measurements at the CSO Facility

Another important issue to be addressed in the next study phase is accurate flow measurement at the CSO facility. This information is required for operating the flocculant dosing apparatus and assessing the facility performance. Ideally, discharges should be measured at the upstream end, at the point of inflow into the CSO tanks. However, such an arrangement was not judged to be feasible (pending further review) and in the earlier studies, the flow rate was measured by combining several methods, e.g., using the rate of tank filling and weir formulas for either inflow or outflow weirs. If these practices are continued, the following measurement uncertainties should be recognized:

(a) Recognizing that the length of the bypass and the final effluent weirs is more than 30 m, and the actual weir crowns may not be perfectly level, discharge calculations using weir formulas and head measurements may include large uncertainties. For sloping weir crests, the error in flow measurements may be in the range from 0.085 to 0.95 m^3/s, for level differences (between the two crest ends) of 1 and 5 cm, respectively. These sources of measurement error should be further addressed.

(b) As discussed previously (Section 3), upstream flow characteristics and the weir height control the water head over the final effluent weir. Thus, the weir formula used should be verified by laboratory measurements in the physical scale model.

(c) The rating curve of the final effluent weir may be also affected by the approach flow. Thus, water surface profiles in the stormwater tank may be similar to those shown in the first segment of curves in Fig. 6, rather than approximating straight lines. Such complex water level patterns were also observed in the physical model. In one experiment, the water levels at two points, one just upstream the final effluent weir and the other one outside of the effluent channel, differed by 0.25 m. Because of the complexity of water surface profiles in the stormwater tank, both laboratory and field measurements of the depths to be used in the weir formula must be made at identical locations.

(d) Experiments indicate that for maximum inflow rates, the CSO tank outflow weirs are always submerged, except when tanks start to fill. This is the case even at low flow rates, when at least one or more outflow weirs are partially submerged most of the time. With so much uncertainty in weir flow measurements, their use for flow measurements should be avoided. Flow measurement by the final effluent weir, activated as soon as the CSO tanks fill up, is a better alternative.

(e) While the tanks are being filled, it is feasible to use the rate of filling to determine the discharge through the tanks. For this purpose, water level sensors need to be installed in the tanks, and the rate of filling, multiplied by the tank plan area (903 m^2 for all three combined) yields the discharge. However, this method is not very accurate, because for a 1 mm/s error in the filling rate, the corresponding flow rate error is about 0.9 m^3/s, or 18 % of the maximum inflow rate of 5 m^3/s. The measurement error can be caused by both the instrument itself and water surface perturbations caused by surface waves, turbulence, and other sources. This problem was encountered even in the physical scale model in well-controlled experiments.

5. CONCLUSIONS

The assessment of flow behaviour in the North Toronto CSO Facility by numerical and physical modelling provided a good insight into facility operation and performance improvements by structural changes. The potential improvements addressed in this study phase included increasing the maximum flow capacity and improving flow conditions in storage/settling tanks. Numerical modelling was accomplished using a 3-D hydrodynamic model PHOENICS (a commercial CFD package), which was run on a PC. The results demonstrated that the 3-D multiphase model in the PHOENICS package reproduced well water levels, flow velocities and other physical flow characteristics in this complex structure with many hydraulic interconnections. Most of the numerical model results were verified by measurements in a 1:11.6 physical scale model, and the differences between the numerical model output and measured results were less than 5%, in most cases. The analysis of the facility showed that with respect to passage of flows, the facility is a complex, highly non-linear hydraulic system.

When examining the feasibility of increasing the facility flow capacity, several problems were identified. Water profiles through the facility were affected by the 90° bends in effluent channel and by flow control weirs. Modelling results showed that:

(a) reducing the height of the final effluent weir only may be a simplistic solution for this complex system. A more comprehensive plan may be needed and should address the effluent channel bend problems;

(b) the rates of flow change in the system also depend on the height of the final effluent weir itself. For lower weirs, increases in flow rate become smaller when reducing the final effluent weir by a constant step, because the water level in the stormwater tank is also affected by the hydraulic conditions upstream. Thus, even if further reductions of the final effluent weir height may appear feasible, considering the decreasing efficiency of this measure and the environmental consequences of sacrificing the stormwater tank storage, a more balanced approach combining effluent channel improvements and lowering the final effluent weir should be taken.

Flow patterns in the storage tanks were found to be non-uniform, particularly in the most downstream tank (Tank 3). Disturbances from the distribution channel propagated into the tanks, and caused sinking, rotating currents, which would disrupt settling in tanks, and cause fast flow along tank sidewalls. Preliminary numerical experiments with retrofitted baffles indicated that these baffles could correct adverse flow conditions. The proposed arrangements comprised a horizontal baffle attached to the inflow weir (extended into the tank), and two or three vertical baffles mounted on the horizontal baffle and providing fairly uniform lateral flow distribution. Other ideas, such as the use of flexible cylinders for achieving uniform flow distribution will be tested in the next study phase.

Both numerical and physical modelling showed clearly that the CSO tank outflow weirs remained submerged for most inflow rates, unless the final effluent weir was removed completely. Thus, the outflow weirs cannot be used for accurate flow measurements.

Other options such as measuring flow rate in connecting pipes would be examined more thoroughly in the next study phase.

Finally, it was concluded that the numerical model, used in tandem with a physical model, was a very flexible, powerful tool that could provide a distinct advantage in future investigations of the North Toronto CSO Facility.

6. ACKNOWLEDGEMENTS

The contributions made by Environment Canada's Great Lakes 2020 Sustainability Fund, the City of Toronto staff Patrick Chessie and Sandra Ormonde, and NWRI Research Support Branch staff Bill Warrender, John Cooper and Brian Taylor, are greatly appreciated. Without their continuing support and help, this study would not have been possible.

7. REFERENCES

Kluck, J. (1996) Design of storm water settling tanks for CSOs. 7th International Conference on Urban Storm Drainage, Hannover, Germany.

Launder, B.E. and Spalding, D.B. (1974) The numerical computation of turbulent flows, Comp. Meth. In Appl.Mech. & Eng., Vol.3, pp. 269.

Pettersson, T.J.R. (1997) FEM-Modelling of open stormwater detention ponds. Nordic Hydrology 28(4/5), 339-350.

Rosten, H.I. and Spalding, D.B. (1984) The PHOENICS reference manual, TR/200, CHAM Ltd, Wimbledon, London.

Shaw, J.K.E., Watt, W.E., Marsalek, J., Anderson, B.C. and Crowder A.A. (1997) Flow pattern characterization in an urban stormwater detention pond and implications for water quality. Water Qual. Res. J. Canada, Volume 32, No. 1, 53-71.

LIST OF FIGURE CAPTIONS

Fig. 1. North Toronto CSO Facility. (1A) 3-D Schematization used in numerical modelling, (1B) 3D numerical simulation result

Fig. 2. Comparison of numerically modelled and measured flow level changes in time, at various locations. Blue lines represent numerical model output, green colour represents measured results. (2.1) Inlet tank (2.2) Distribution channel (2.3) CSO Tank 1 (2.4) CSO Tank 2 (2.5) CSO Tank 3, and (2.6) By the bypass weir.

Fig. 3. Numerically modelled and measured flow velocities in the interconnecting pipes between the inlet and distribution channels.

Fig. 4. Numerically simulated and measured flow velocities in the CSO tanks. (4A) Measured velocities (4B) Modelled velocities.

Fig. 5. Numerically modelled flow levels by the bypass weir for different heights of the final effluent weir.

Fig. 6. Water levels in the effluent channel and the adjacent part of the stormwater tank for different bend conditions and the final effluent weir height reductions. (6A) Weir height reduction by 40 cm, (6B) Weir height reduction by 60 cm.

Fig. 7. Different bend geometries used in numerical simulations.

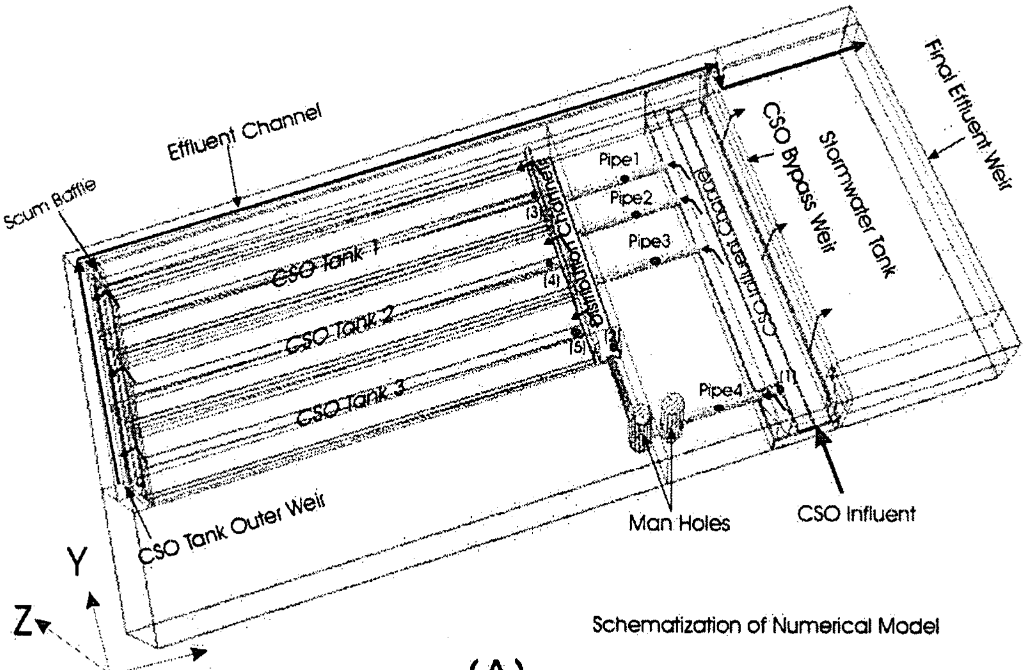
Fig. 8. Numerical experiment results for water surface profiles in the effluent channel with different bend arrangements.

Fig. 9. Computer simulated velocity patterns for three different bend geometries.

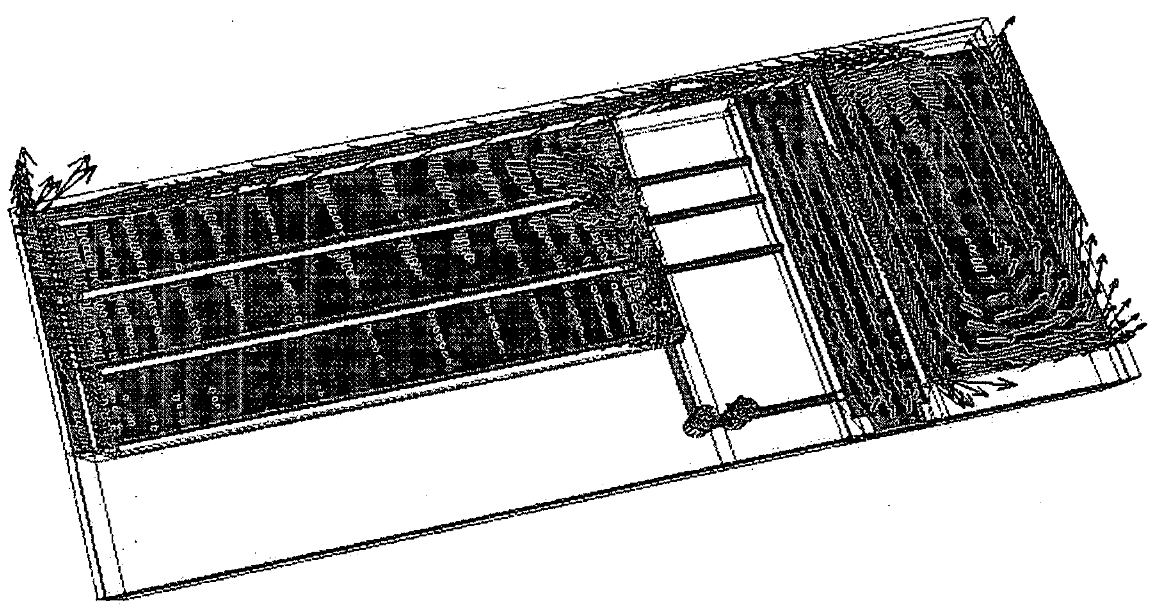
Fig. 10. Maximum flow rates obtained for various heights of the final effluent weir and different corner bend arrangements. Solid lines indicate numerical model output, stars indicate measured results.

Fig. 11. Change of flow patterns achieved by addition of baffles. (11A) Baffle arrangements, (11B) Improved flow fields in the three CSO tanks.

Fig. 12. Photograph of the physical scale model.

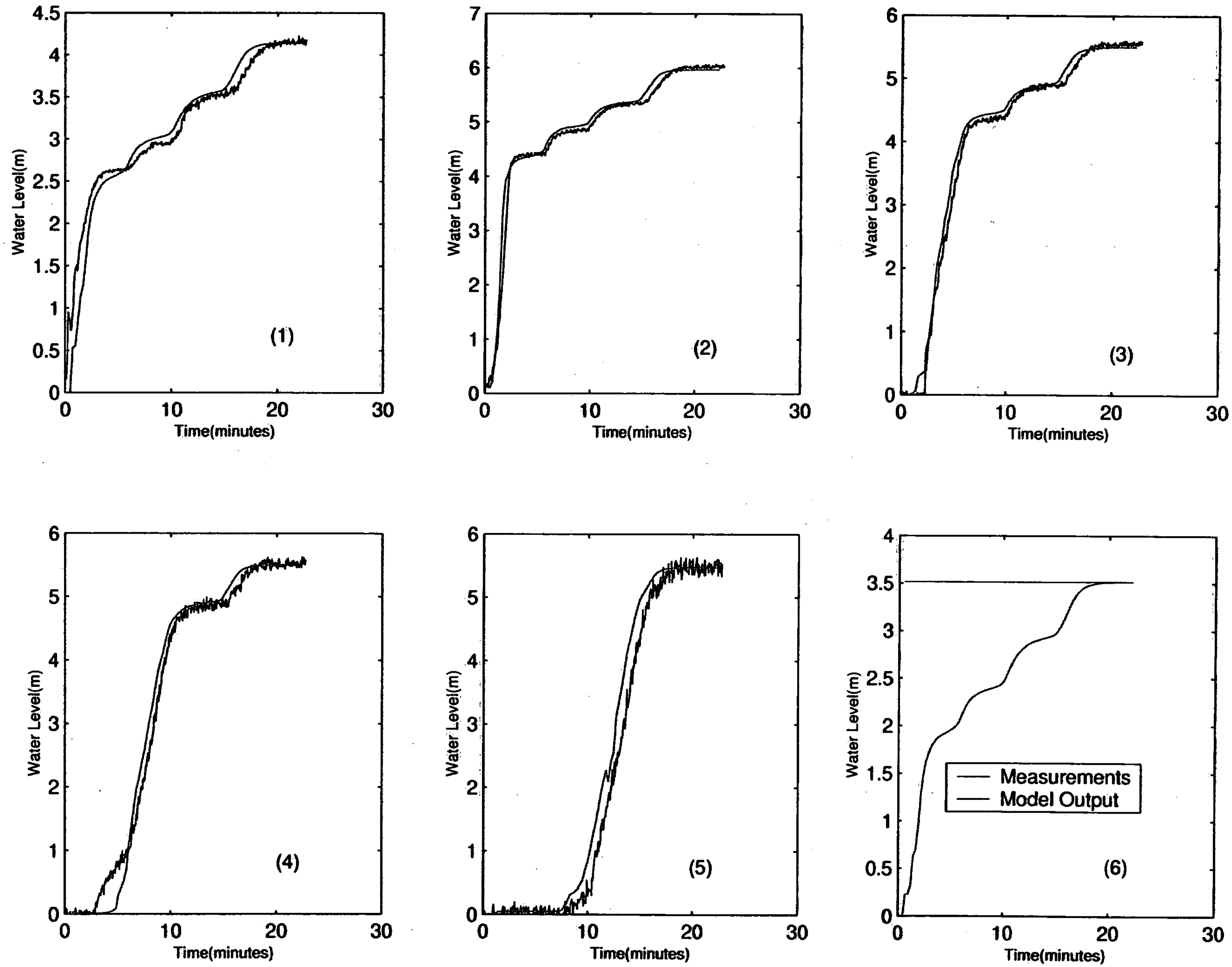

Effluent Channel
Scum Baffle
CSO Tank 1
CSO Tank 2
CSO Tank 3
Distribution Channel
Pipe1
Pipe2
Pipe3
Pipe4
CSO Bypass Weir
Stormwater Tank
Final Effluent Weir
CSO Tank Outer Weir
Man Holes
CSO Influent
Y
Z
X
Schematization of Numerical Model

(A)



(B)

Fig. 1



Water Level(m)
Time(minutes)
(1)
(2)
(3)
(4)
(5)
(6)
Measurements
Model Output

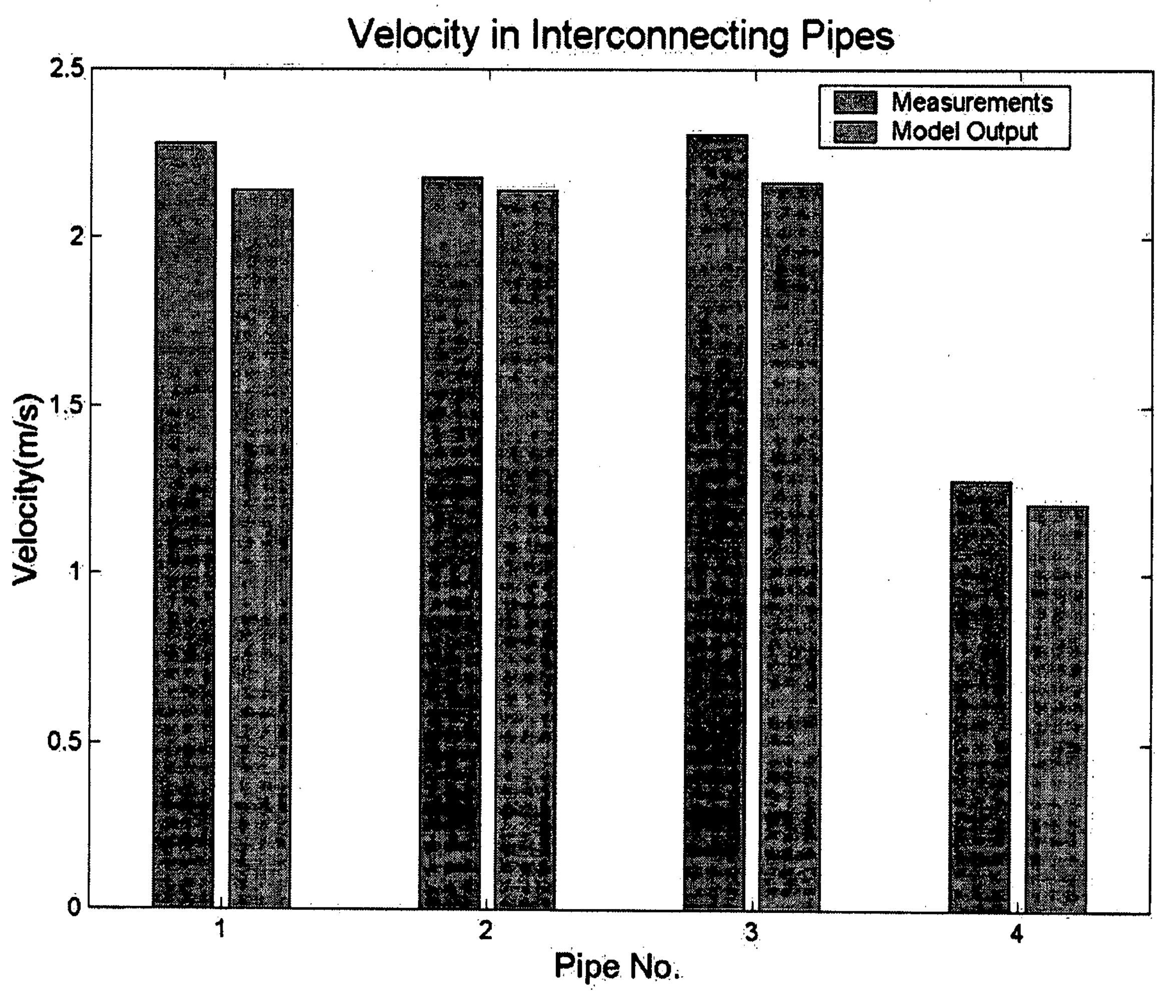
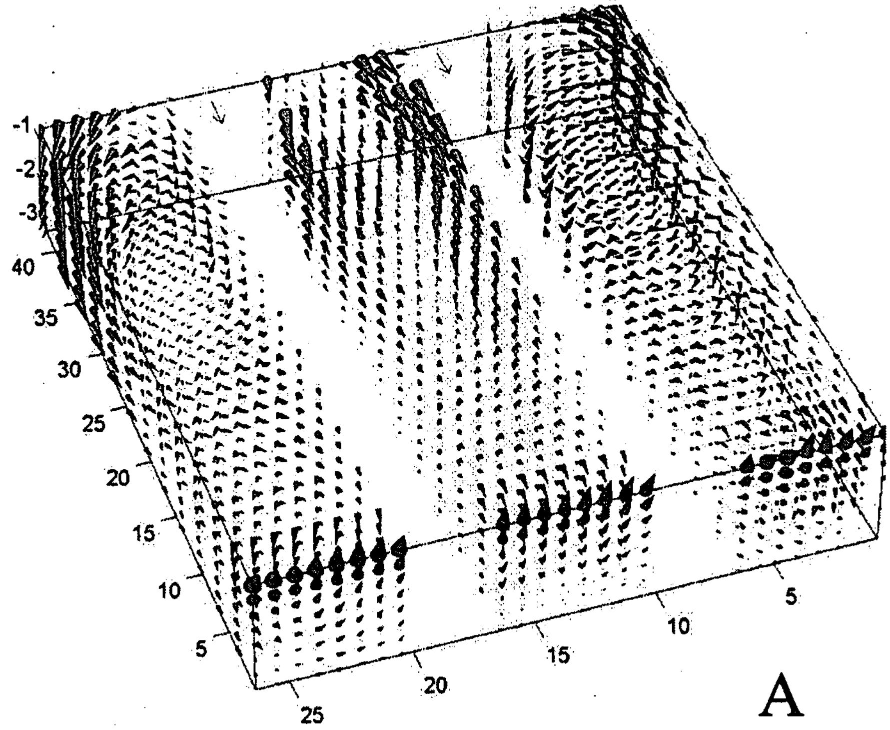

Velocity in Interconnecting Pipes
Measurements
Model Output
Velocity(m/s)
Pipe No.
2.5
2
1.5
1
0.5
0
1
2
3
4

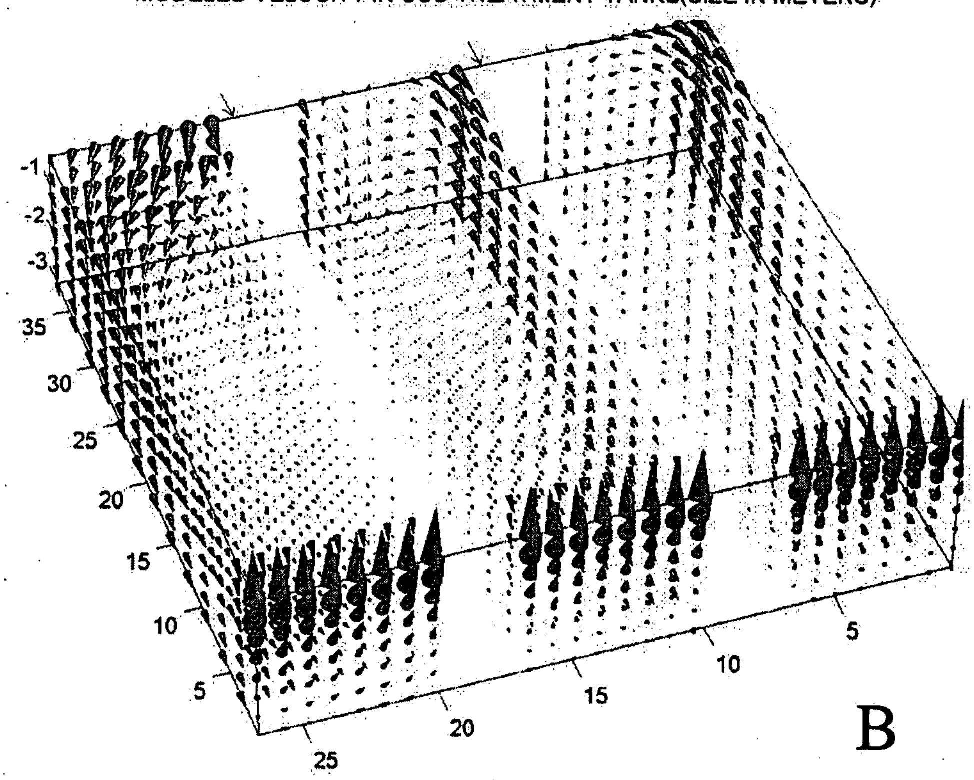
Fig. 3

MEASURED VELOCITY IN CSO TREATMENT TANKS(SIZE IN METERS)



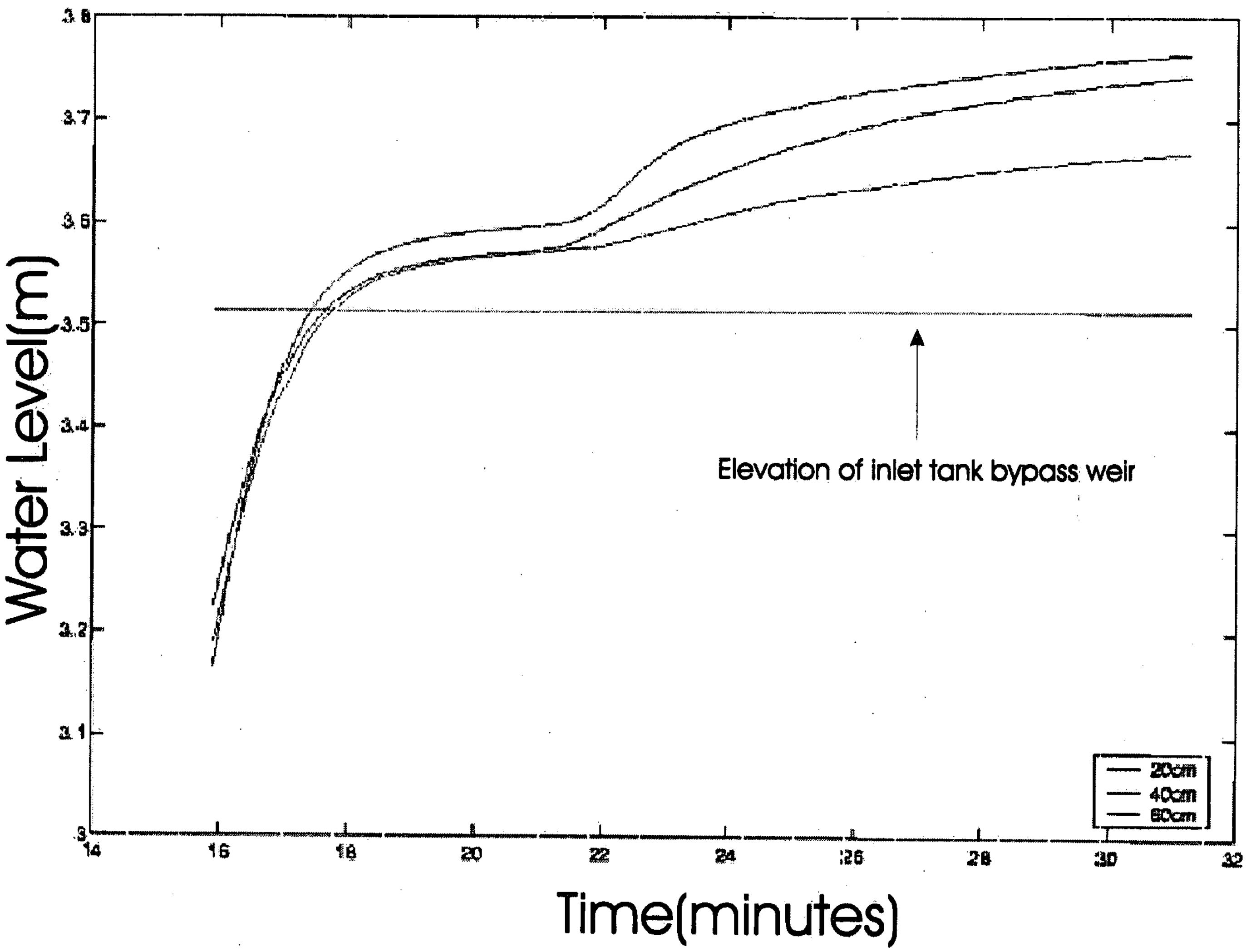
A

MODELED VELOCITY IN CSO TREATMENT TANKS(SIZE IN METERS)



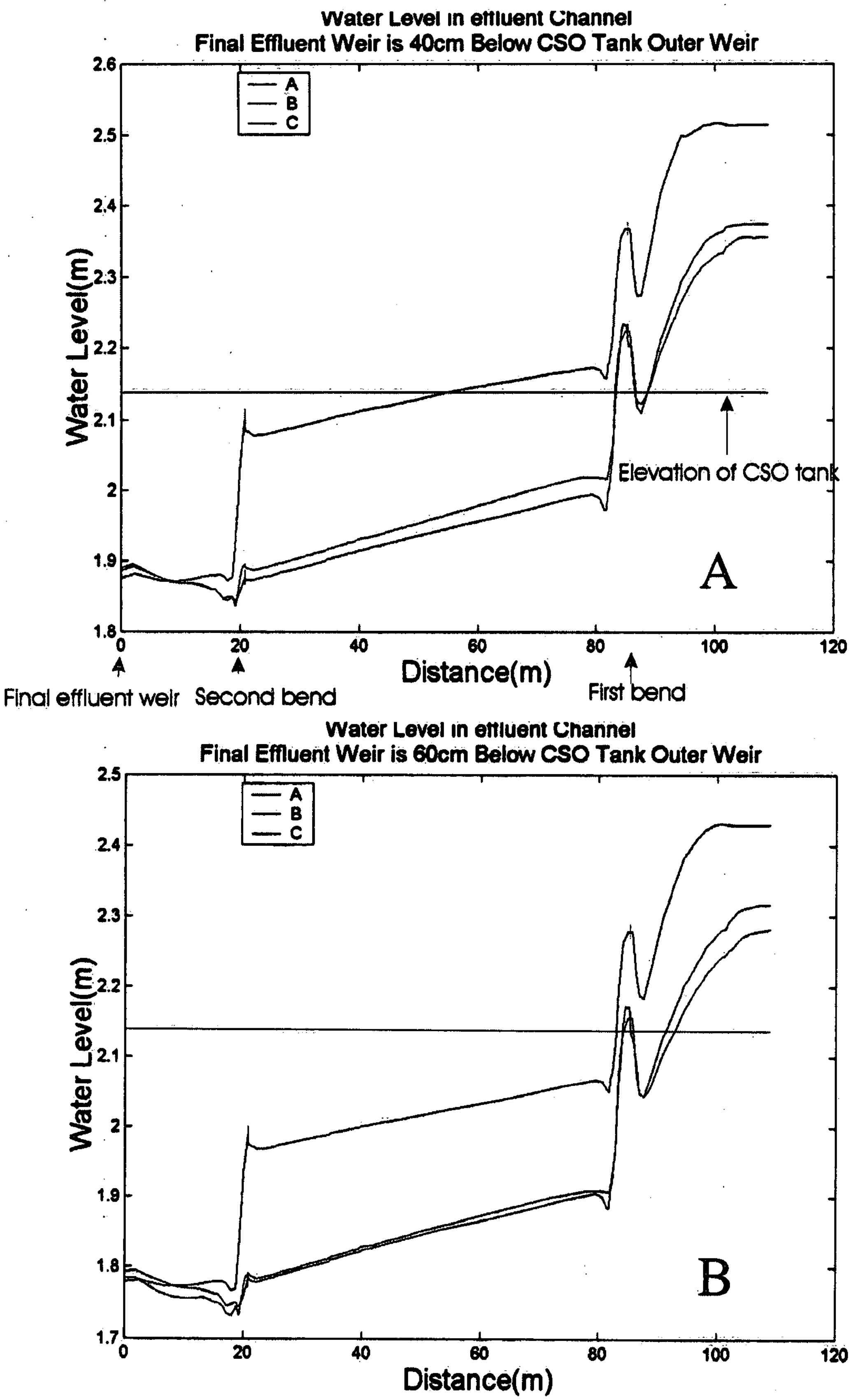
B

Fig. 4



3.8
3.7
3.6
3.5
3.4
3.3
3.2
3.1
3
Water Level(m)
Elevation of inlet tank bypass weir
20cm
40cm
60cm
14
16
18
20
22
24
26
28
30
32
Time(minutes)

Fig. 5



Water Level in effluent Channel
Final Effluent Weir is 40cm Below CSO Tank Outer Weir
A
B
C
Water Level(m)
2.6
2.5
2.4
2.3
2.2
2.1
2
1.9
1.8
0
20
40
60
80
100
120
Distance(m)
Elevation of CSO tank
A
Final effluent weir
Second bend
First bend
Water Level in effluent Channel
Final Effluent Weir is 60cm Below CSO Tank Outer Weir
A
B
C
Water Level(m)
2.5
2.4
2.3
2.2
2.1
2
1.9
1.8
1.7
0
20
40
60
80
100
120
Distance(m)
B

Fig. 6

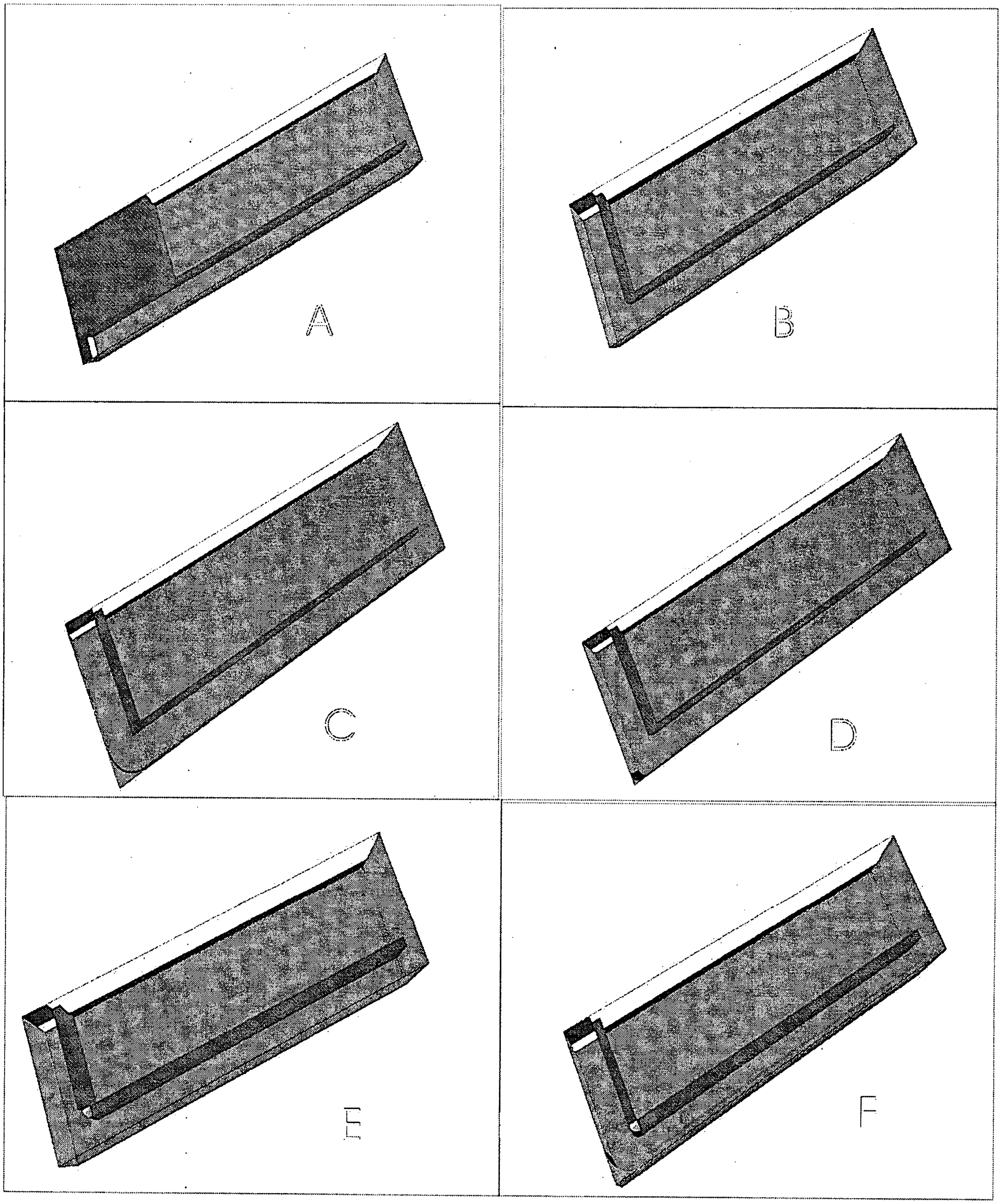


Fig. 7

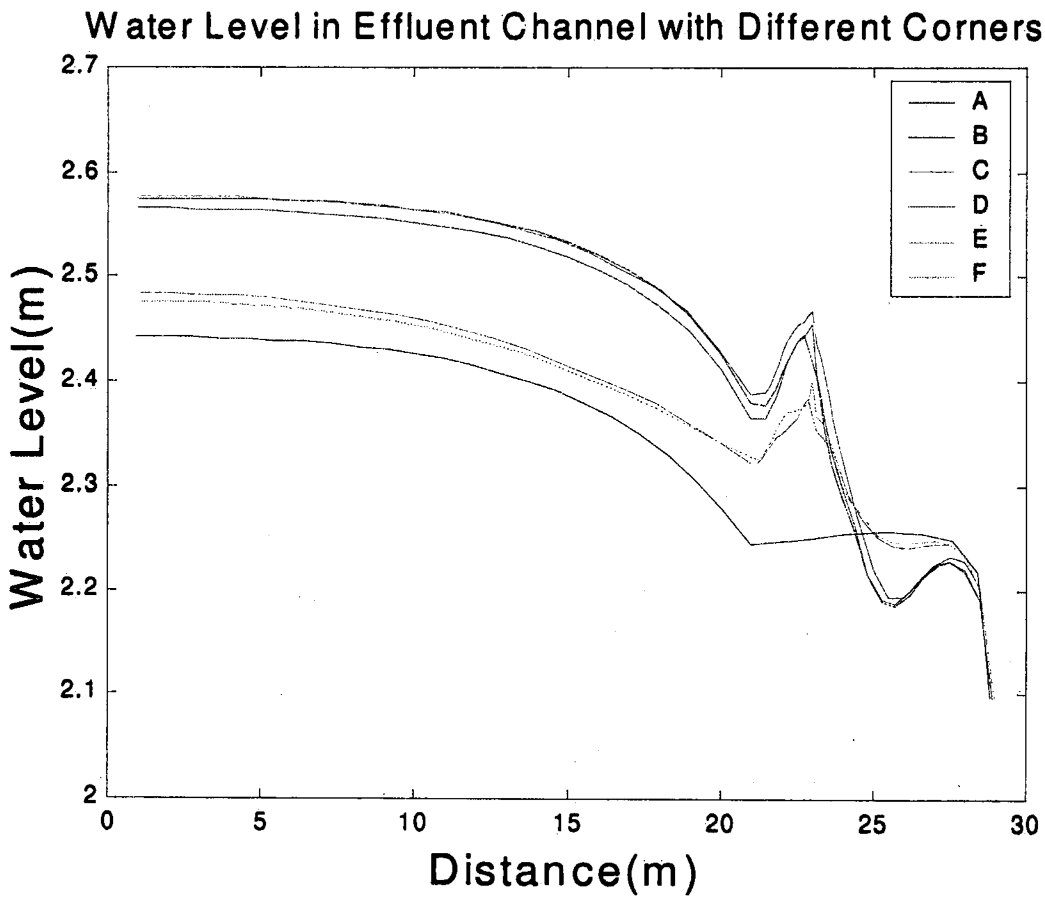

Water Level in Effluent Channel with Different Corners
Water Level(m)
2.7
2.6
2.5
2.4
2.3
2.2
2.1
2
Distance(m)
0
5
10
15
20
25
30
A
B
C
D
E
F

Fig. 8

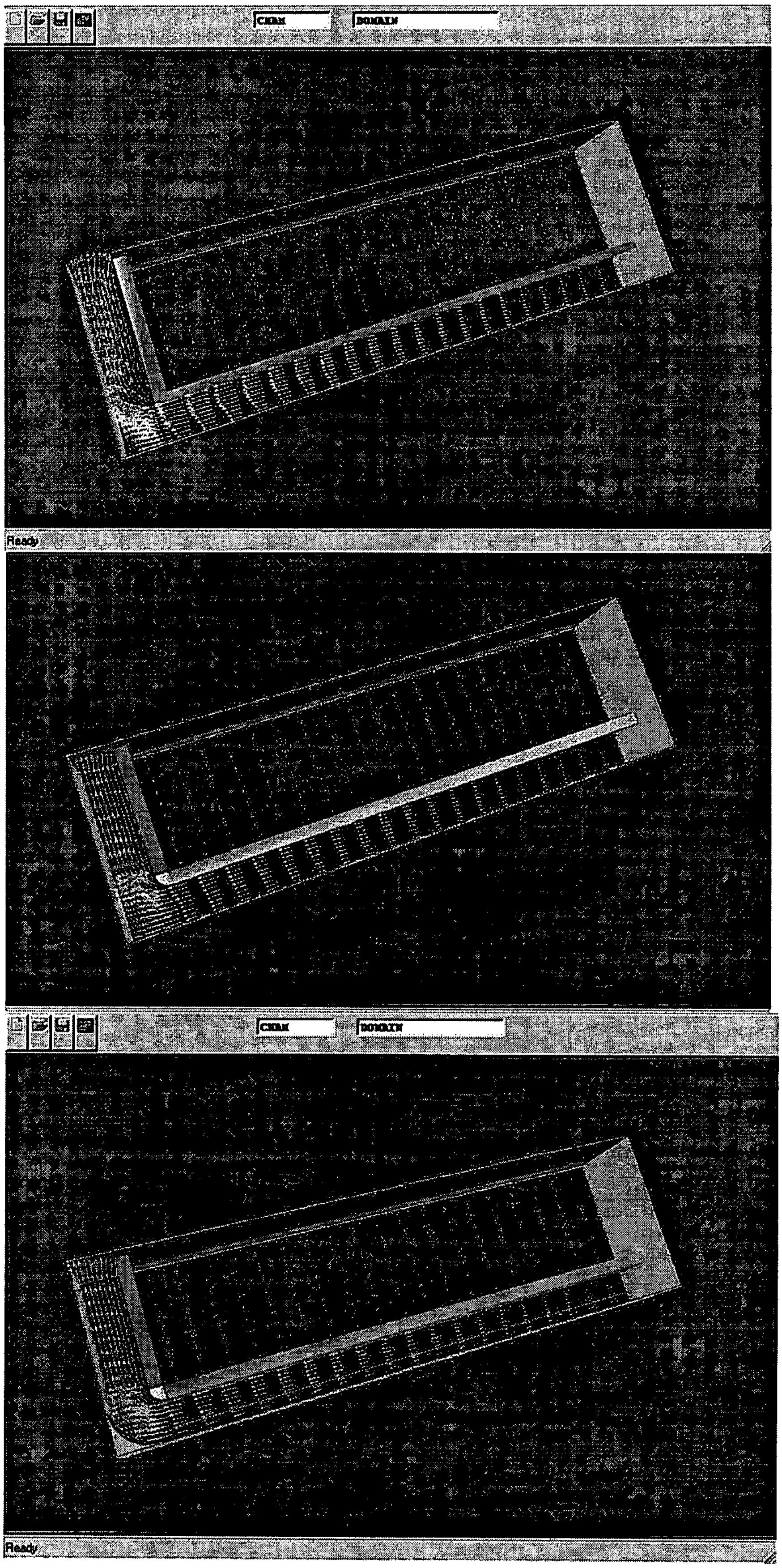
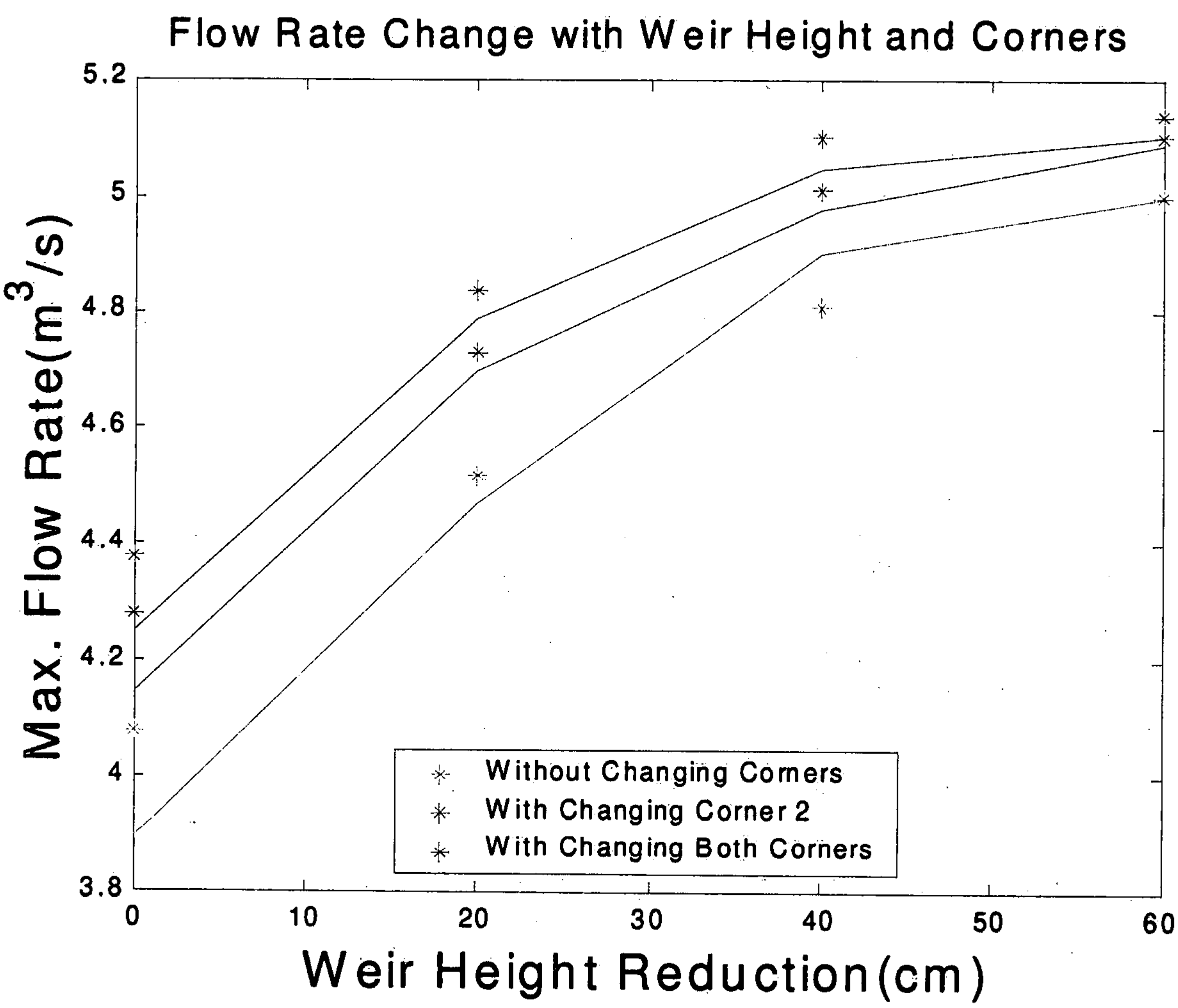
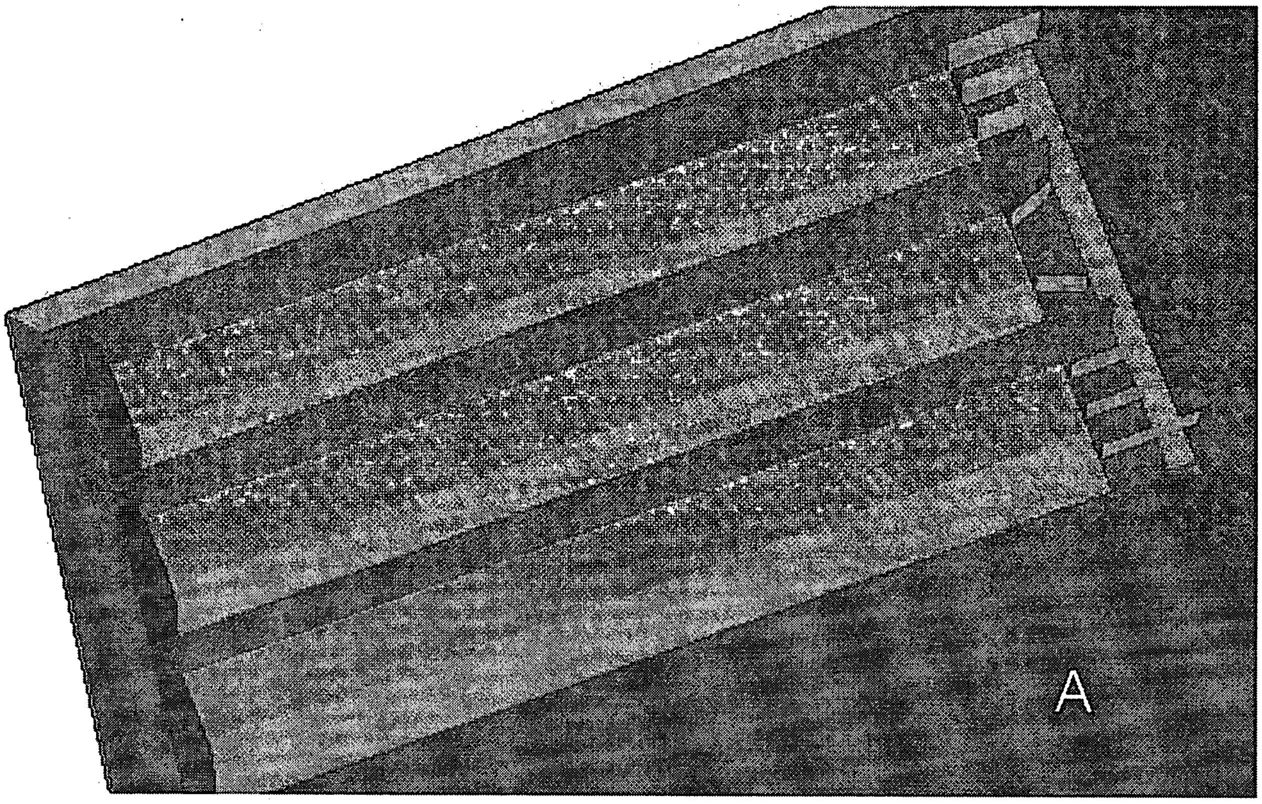


Fig. 9



Flow Rate Change with Weir Height and Corners
5.2
5
4.8
4.6
4.4
4.2
4
3.8
Max. Flow Rate(m^3/s)
0
10
20
30
40
50
60
Weir Height Reduction(cm)
Without Changing Corners
With Changing Corner 2
With Changing Both Corners

Fig. 10


A

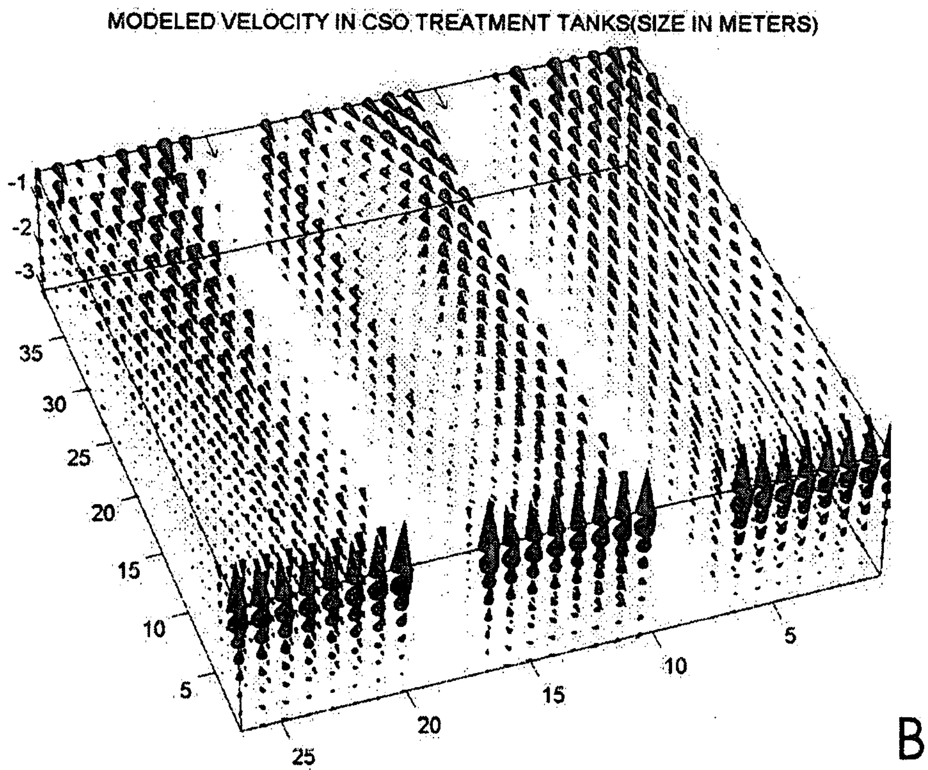

MODELED VELOCITY IN CSO TREATMENT TANKS(SIZE IN METERS)
-1
-2
-3
35
30
25
20
15
10
5
25
20
15
10
5
B

Fig. 11

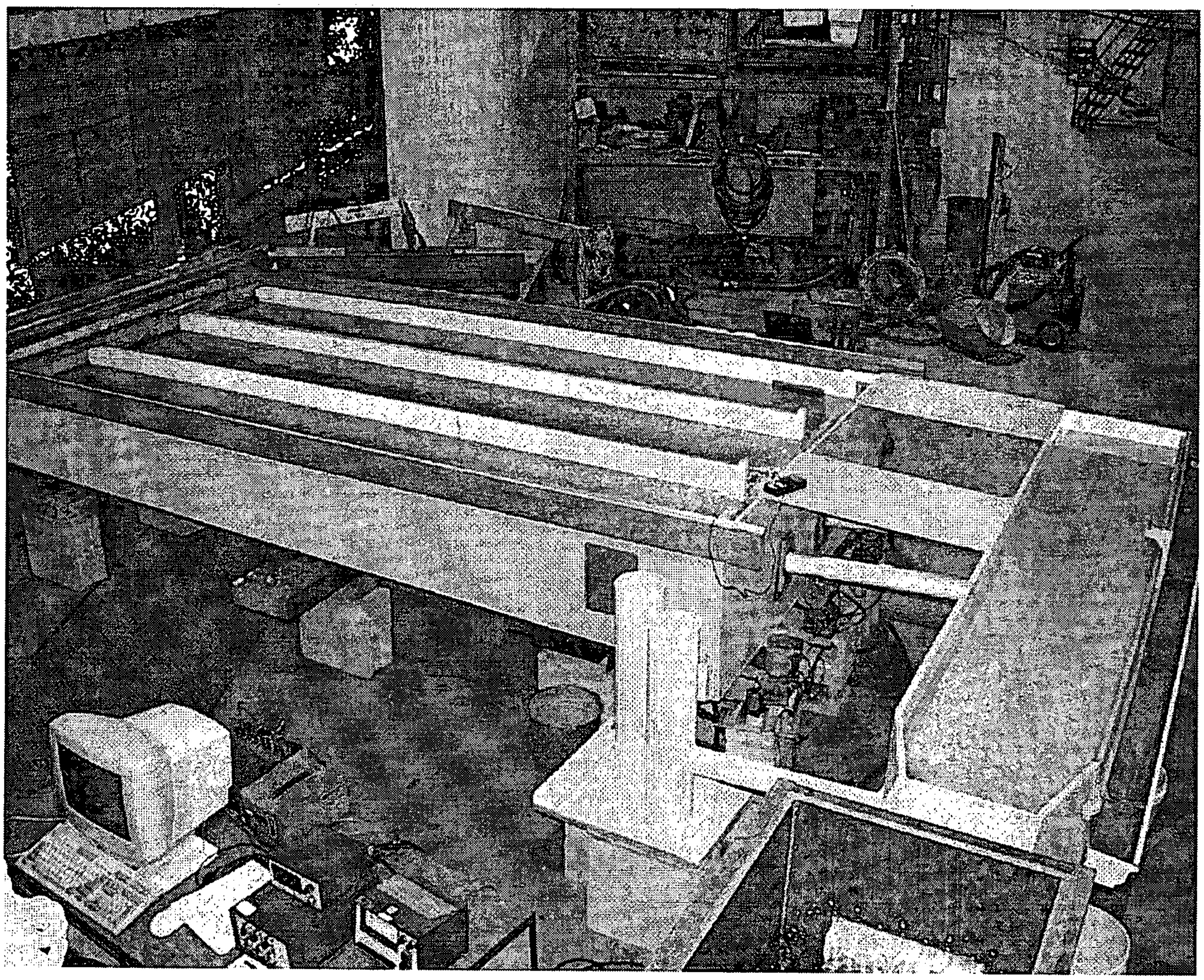


Fig. 12

PRINTED IN CANADA ON RECYCLED PAPER
IMPRIMÉ AU CANADA SUR DU PAPIER RECYCLÉ

National Water Research Institute
Environment Canada
Canada Centre for Inland Waters
P.O. Box 5050
867 Lakeshore Road
Burlington, Ontario
L7R 4A6 Canada

National Hydrology Research Centre
11 Innovation Boulevard
Saskatoon, Saskatchewan
S7N 3H5 Canada

NATIONAL WATER RESEARCH INSTITUTE
INSTITUT NATIONAL DE RECHERCHE SUR LES EAUX

Institut national de recherche sur les eaux
Environnement Canada
Centre canadien des eaux intérieures
Case postale 5050
867, chemin Lakeshore
Burlington, Ontario
L7R 4A6 Canada

Centre national de recherche en hydrologie
11, boul. Innovation
Saskatoon, Saskatchewan
S7N 3H5 Canada

Environment Canada Environnement Canada

Canada