HYDRAULIC MODEL INVESTIGATION ON BRANCH 3&4 SYPHON UNDER TOSHKA SPILLWAY CANAL

Yasser Shawky

Researcher, National Water Research Center, Delta Barrage P.O. Box 13651, Egypt

ABSTRACT

The main purpose of this paper was to check the design of the syphon which will be constructed on branch 3&4 of El-Sheikh Zayed Canal and will pass under Toshka Spillway Canal. The study included the different scenarios for the syphon operation and calculation of Head Loss at each scenario and its effect on the efficiency of the syphon performance. At each scenario, the separation zones and the vortex formation at all conditions, which may affect on the syphon stability and performance, were observed. The expected sedimentation rates inside the syphon, determination of its removing procedures, the velocities measurements inside the syphon and its effect on the syphon inlet and outlet were also studied in the paper.

For this purpose, a physical model for the Sheikh Zayed canal and the syphon body was constructed with scale 1:27. The model test program included all scenarios of tunnels operations. These scenarios were tested in order to determine the best inflow and outflow conditions (flow pattern and separation zones), when one, two, three or four tunnels in operation. The results indicated that the design of the inlet structure is acceptable although separation zones were created by the piers edges and the transition wall as they considered have very small effect in the inflow pattern. It was recommended to construct a sediment trap upstream the inlet structure and it should be monitored each 6 months to remove the accumulated sediment if necessary. It was recommended also to decrease the horizontal angle of the two vertical transition walls downstream the syphon outlet from 30° to 13°. This is to improve the flow uniformity and reduce the flow resistance.

Keywords: Hydraulic, Syphon, Head Loss, Separation zones, Vortex formation, Sedimentation

INTRODUCTION

The Horizontal Expansion and Projects Sector, Ministry of Water Resources and Irrigation assigned the Hydraulics Research Institute (HRI) to conduct a hydraulic model study of the proposed Syphon of Branch 3&4 – El Sheikh Zayed Canal under Toshka Spillway Canal.
PURPOSE OF STUDY

The purposes of this paper are summarized as follows:

- Study of different scenarios for the syphon operation and calculation of Head Loss at each scenario and its effect on the efficiency of the syphon performance.
- Study of the separation zones and the vortex formation at all conditions, which may affect on the syphon stability and performance.
- Study of the expected sedimentation rate inside the syphon and determination of its removing procedures.
- Study and measure the velocities inside the syphon and its effect on the syphon body and the inlet and outlet.
- Determination of the required tests for insuring non-seepage from the syphon and checking the efficiency of its performance (after put the syphon in operation).

MODEL SIMILITUDE AND SCALE

A well-designed hydraulic scale model may be the best tool to provide the optimal hydraulic performance of the syphon inlet and outlet structures. The hydraulic model gives direct insight into the physical processes, which cannot be obtained otherwise. A 1:27 undistorted hydraulic scale model was constructed at HRI. The model represents 150 m of the branch upstream the syphon, the inlet structure to the syphon, the total length of the syphon, the outlet structure from the syphon and 150 m of the branch downstream the syphon. Figure (1) shows plan and elevation view for the proposed syphon and the transition zones of the branch upstream and downstream the syphon.

Figure (1) Plan and Elevation View for the Proposed Branch 3&4 Syphon under Toshka Spillway Canal
To properly simulate the kinematics and dynamics of the fluid flow field, an undistorted geometric model is required. Since the model has free surface, inertial and gravitational forces are dominant. Therefore, the model has to be based on Froude number criterion with the prototype. The Froude number, which represents the square root of the ratio of fluid inertial stresses to gravitational stresses, is given by:

$$F = \frac{V}{(gL)^{0.5}}$$

in which:  
- $F =$ Froude number,  
- $V =$ velocity in m/s,  
- $L =$ Characteristic length (flow depth) in m, and  
- $G =$ Gravitational acceleration$= 9.81 \text{ m/s}^2$

Froude similarity requires that the Froude number ratio $= 1$, i.e. the Froude numbers in both model and full size prototype are equal. From this condition, the following scales are derived when selecting the geometrical scale.

- Geometrical scale $n_1$
- Velocity and Time scale $n_v = n_1 = (n_1)^{0.5}$
- Discharge scale $n_Q = (n_1)^{2.5}$

Viscous and surface tension forces can also affect the flow field and vortex formations in a model. The Reynolds number $Re$ expresses the ratio of inertial to viscous stresses.

$$Re = \frac{VL}{\nu}$$

where: $\nu =$ kinematic viscosity $\text{m}^2/\text{s}$.

The Weber number $We$ expresses the ratio of inertial to surface tensile stresses.

$$We = \frac{\rho V^2 L}{\sigma}$$

in which:  
- $\rho =$ fluid density, $\text{kg/m}^3$, and  
- $\sigma =$ surface tension, $\text{kg/s}^2$

Because the model flow rate was determined on the basis of equal model and prototype Froude numbers, Reynolds number could not have the same values as they have in the prototype. Typical Reynolds number in the intake model should be greater than $3 \times 10^4$ [Anwar, 1977], so the flow conditions in the model are fully turbulent as the prototype. Therefore, scale effects due to viscous forces were negligible. Also the Weber number in the intake model should be greater than $280 \times h/d$ [(h= submergence, d= suction diameter), Martins, 1984], to ensure that its effect on the scale of vortex formation is negligible. In case the model Weber number is less than its required value tests of vortex formation should be checked at exaggerated model velocity.
According to the available facilities at HRI such as area, pump capacity, electric source and instrumentation, the length scale was selected to be 1:27. As the model was operated according to Froude similarity alone, other scales are:

\[
\begin{align*}
\text{Length scale} &= n_l = 27 \\
\text{Velocity scale} &= n_v = \text{time scale} n_t = 5.196 \\
\text{Discharge scale} &= n_Q = 3787.995
\end{align*}
\]

The selected model scale was checked to ensure enough model turbulence. The minimum Reynolds number of different model locations showed that the model had enough turbulence everywhere (minimum value \(3.4 \times 10^4 > 3 \times 10^4\)). The model which may be based upon Weber number criterion was found higher than the required limit (237 > 119). So, in order to test the vortex formation in the model it was not necessary to exaggerate the model velocities.

**MODEL DESCRIPTIONS**

Figure (2) shows the general layout of the model (model dimensions). The model has recirculation system and consists of 3 main sections: the model entrance, the modelled reach of the canal and the syphon, and the model exit.

![General Layout of the Model](image)

The model entrance consists of a masonry basin of 6.0 m wide, 2.0 m long and 1.5 m high. A pump of capacity 65 l/s. (which is enough to represent the maximum discharge in the model) sucks the water from a sump of 4 m water depth and discharges it into the model entrance. In order to dissipate the energy of the flow, which enters the model as well as to avoid any disturbance of the flow in the canal section, a distributor of wood slides was used at the beginning of the model.
The following reaches of branch 3&4 and the syphon were represented in the model:
- 150 m of branch 3&4 of El-Sheikh Zayed Canal U.S the syphon
- The syphon inlet
- The whole length of the syphon (4 tunnels)
- The syphon outlet
- 150 m of branch 3&4 of El-Sheikh Zayed Canal D.S the syphon

The represented reaches of branch 3&4 in the model were made of a sand-cement mortar. The cross-sections were shaped by 15 mm thick wooden sheets according to the length scale, placed according to their levels and locations, and cemented. Then the area between each two cross sections was filled with cement mortar by considering the relevant bed levels.

The inlet and outlet structures were constructed from wood. The whole length of the four tunnels was represented in the model. The tunnels were manufactured from sheets of iron with 3 mm thickness in HRI workshop. Figures (3) and (4) show the layouts of the transition zones of branch 3&4 and the syphon inlet and outlet (prototype dimensions).

Figure (3) Transition Zone of Branch 3&4 Upstream the Syphon Inlet
The model exit consists of a basin that starts directly at the end of the simulated reach of the branch followed by a tail control gate to adjust the corresponding water surface levels in the model.

The syphon model was provided with a recirculating system. The maximum feeding capacity of the system was 65 l/s. This capacity was sufficient as the maximum scaled discharge of the model is 461/s corresponding to the maximum prototype discharge (175 m³/s). This required discharge was pumped directly from a sump to the model through the suction pipe of the pump (6 inch diameter). The sump takes the water from high reservoir, by a pipe of 3-inch diameter. This reservoir has a total capacity of nearly 100 m³. An ultrasonic flowmeter was installed on the feeding pipe of the pump (6 inches diameter) to measure the required discharge to the model.

To determine the inflow discharging into the model, an ultra-sonic flowmeter was installed on the feeding pipe of the pump as mentioned above. This flowmeter indicated the inflow into the model within accuracy limits of ± 1 %. Velocities in the model were measured using Electro-Magnetic Current meters manufactured by Delft Hydraulics- the Netherlands.

To monitor water surface levels along the modelled reach of the branch and syphon inlet and outlet, 4-point gauges were installed within stilling wells. The accuracy of

---

**Figure (4) Transition Zone of Branch 3&4 Downstream the Syphon Outlet**
these gauges is within $+0.1$ mm. For composing flow patterns and indicating surface flow currents in the vicinity of the syphon inlet and outlet and in the transition zones of the branch upstream and downstream the syphon, wooden floats were applied and their float paths recorded by a photo camera.

The model was calibrated to adjust the canal inflow/outflow arrangement, to have a complete uniform flow upstream the system structure. One canal cross-section was used for calibration. At this cross-section velocities were measured at 14 points. In each point, the measurements were conducted at 0.5 of the local depth. A vertical wood slides at the entrance of the model were used to adjust the inflow.

**MODEL TEST PROGRAM**

The model test program was designed to provide sufficient information about the prediction of the hydraulic performance of the syphon in the prototype and to check the flow distribution in the transition zones upstream and downstream the syphon (inflow and outflow distribution).

All different tunnel combinations were tested in order to determine the best inflow and outflow conditions (flow pattern and separation zones), when one, two, three or four tunnels were operated. Seven cross sections were determined for each transition zone of the branch (upstream and downstream the syphon) to measure the velocity distribution of the inflow and outflow. Figures (5) and (6) show the location of these cross-sections (prototype dimensions). Float tracing technique was also used to test the inflow and outflow distribution. Table (1) shows the different scenarios of the tunnels operations, and the corresponding hydraulic parameters (discharge and water levels).

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Tunnel No.</th>
<th>Flow Discharge (m$^3$/s)</th>
<th>Water Level U.S (m)</th>
<th>Water Level D.S (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Opened</td>
<td>175.00</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>2</td>
<td>Opened</td>
<td>43.75</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>3</td>
<td>Closed</td>
<td>43.75</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>4</td>
<td>Opened</td>
<td>87.50</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>5</td>
<td>Closed</td>
<td>87.50</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>6</td>
<td>Opened</td>
<td>131.25</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>7</td>
<td>Opened</td>
<td>175.00</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>8</td>
<td>Opened</td>
<td>131.25</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>9</td>
<td>Opened</td>
<td>87.50</td>
<td>194.97</td>
<td>194.43</td>
</tr>
<tr>
<td>10</td>
<td>Opened</td>
<td>87.50</td>
<td>194.97</td>
<td>194.43</td>
</tr>
</tbody>
</table>

**Table (1) Test Program of the Different Scenarios of the Syphon Operation**
Figure (5) Location of the Selected Cross Sections for Velocity Measurements (Upstream the Inlet of the Syphon)

Figure (6) Location of the Selected Cross Sections for Velocity Measurements (Downstream the Outlet of the Syphon)
The pressure distribution inside the syphon was measured along the whole length of tunnel No.1 in all tests, which it is in operation. The pressure was measured by means of thirty pressure cells (piezometer) installed above the surface of the tunnel.

The vertical velocity distribution inside the syphon was measured in four tests (test No.1, 2, 6, and 9). The measurements were taken at the locations of the access tower (manholes, which installed above the surface of the tunnels, see Figure1). The purpose of these measurements was to determine the degree of uniformity of the vertical velocity distribution inside the syphon to check its effect on its body. Also it was important to determine the minimum value of the velocity inside the syphon to predict the sedimentation, which may occur inside it.

The vortex formation and submergence availability was also tested in the same four tests mentioned above.

**TESTS RESULTS**

At each test the velocities along the selected cross sections were measured. Each cross section was divided into some points with a distance of 4 m (prototype value) between each two successive points. At each point the velocity was measured at three depths (0.2, 0.5 and 0.8 of the local water depth).

1. **Approach Flow to the Syphon Inlet**

1.1 **Inflow Distribution**

The velocity measurements and photos showed a uniform approach flow, only when four tunnels are in operation. Flow separations around the piers were observed when 1, 2, or 3 tunnels were operated. These separations were caused by the round edge of the piers of the inlet structure. Separations were observed also in the transient wall to the inlet structure but with small effect in the inflow pattern.

1.2 **Vortex Formation**

There are four types of Vortex, which may be formed inside the inlet structure, which are:

- **Type 1:** Rotating surface dimples or short tails without formation of air bubbles.
- **Type 2:** Rotating short tails with distinct air bubbles breaking off from the tail, but not drawn into the intake.
- **Type 3:** Rotating moderate tails with distinct air bubbles breaking off from the tail and drawn into the intake at rather irregular time intervals.
- **Type 4:** Rotating vortex with open-air core reaching into the intake, accompanied by an audible sucking noise. This type of vortex may be steady or unsteady.
Vortex formation tests showed vortices type 1 and 2 only, which are permissible. Air entering vortices (types 3 and 4) never occurred; consequently the available submergence is sufficient.

1.3 Water levels Upstream and Downstream the Syphon

In all tests the water level upstream and downstream the syphon was measured to insure that the syphon losses are within the permissible limits and the required upstream and downstream water levels were verified. As the increase of the syphon losses will affect on the downstream water level.

The measurements indicated that the water levels were nearly verified in all tests (Max. upstream and downstream water levels). For test no. 7 at which the maximum flow (175 m$^3$/s) passed only from three tunnels, the required upstream water level was not verified with the corresponding required downstream water level (upstream water level was 195.24 m instead of 194.97m at downstream water level of 194.43 m). This refers to the increase of the velocity inside the tunnels, which led to the increase of the losses.

1.4 Prevention of Sediment Entrance to the Syphon

In order to prevent some of the sediments to enter the syphon, sediment trap was applied to help for deposition some of it before it reaches to the syphon inlet. A depth of 2 m was suggested for the trap to increase its capacity for deposition and to increase the period at which the accumulated sediment should be removed from it and consequently decrease the maintenance costs.

The suggested sediment trap is shown in Figure (7). Due to the limitation of the bed width of the branch it was recommended to keep one meter at least from its both sides. Then a side slope of 2H: 1V was applied from each side up to level 2 m below the design bed level (187 m). This means that the bed width and the depth of the trap well be equal to 10 m and 2 m consequently. The length of the trap should be at least four times its width approximately. So a length of 80 m was applied. The outlet line of the trap was suggested to be 30 m far from the beginning of the upstream transition zone. Then a slope of 5H: 1V was applied in the upstream direction up to level 2 m below the design bed level and followed by 80 m horizontal and again a slope of 5H: 1V up to the design bed level. The purpose of the slope 5H: 1V in the longitudinal direction was to prevent any disturbance of the outgoing flow from the trap to the natural branch bed.
The suggested sediment trap was represented in the model to check its effect on the flow distribution upstream the syphon inlet. The velocities were measured at the seventh cross sections in the upstream transition zone for four tests [test No.1 (4 tunnels in operation), test No. 2 (1 tunnel in operation), test No. 6 (three tunnels in operation, and test No. 9 (two tunnels in operation)]. The trap had very small effect on the flow distribution for a short distance and far enough from the syphon inlet.

2. Syphon Body

2.1 Pressure Distribution inside the Syphon

As mentioned in the test program, the pressure head along the whole length of tunnel no. 1 was measured in the tests which tunnel 1 was in operation. The pressure was measured by means of thirty pressure cells (piezometer) installed on its surface. The energy head at each pressure cell was computed (potential head, pressure head, and velocity head). The average velocity inside the tunnel was considered constant along the whole length of the tunnel. The purpose of these computations was to determine the energy gridline along the tunnel. The total head loss between the inlet and outlet structure was computed by subtracting the energy head at the outlet structure from the energy head at the inlet structure.

2.2 Vertical Velocity Distribution inside the Syphon

The vertical velocity profile was measured inside the tunnels for four tests (as mentioned in the test program paragraph) at the locations of the access tower (manholes, which installed above the tunnels). The purpose of these measurements
was to determine the degree of uniformity of the vertical velocity distribution inside the syphon to check its effect on its body. Also it was important to determine the minimum value of the velocity inside the syphon to predict the sedimentation, which may occur inside it. From the measurements, it was clear that the vertical distribution of the velocities inside the syphon is considered ideal as in case of uniform flow in closed conduits. This means that there will not be a direct effect of the velocity on the syphon body. The minimum value of the velocity was found nearly 0.80 m/s, which is considered enough to prevent the deposition of the particles of sediment which have mean diameter ($D_{50}$) less than 0.05 mm. This means that the predicted rate of sediment, which may deposited in the syphon well be a little bit small and can be removed easily every period of time.

3. Outflow Distribution

The round piers of the outlet structure created separation areas downstream them. The flow at the end of the transition area, downstream the structure was non-uniform and flow recirculation was observed. The main reason of this non-uniformity was the angle of the vertical transition walls immediately downstream the syphon outlet.

For this reason the transition walls were reshaped in the model with horizontal angle $13^\circ$ instead of $30^\circ$ and extended until they met the banks of the branch. Then four tests were repeated again (test No. 1, 2, 6, and 9). The velocities measurements and photos indicated some improvements in the flow distribution but still there is some recirculation of the flow especially in the case of operating the two tunnels in the middle. Figure (8) shows plan view for the designed and proposed vertical walls of the outlet structure.
CONCLUSIONS AND RECOMMENDATIONS

1. Conclusions

Based on the obtained results, it can be concluded that

- The velocity measurements showed some flow separations at the inlet structure around the piers when 1, 2, or 3 tunnels in operation. This separation was caused by the round edge of the piers of the inlet structure. Separations were observed also in the transient wall to the inlet structure.

- Vortex formation tests showed vortices type 1 and 2 only, which are permissible. Air entering vortices (type 3 and 4) never occurred; consequently the available submergence is sufficient.

- The proposed suggested sediment trap had small effect on the flow distribution for a short distance and far enough from the syphon inlet.

- The results indicated that the water levels were nearly verified in all scenarios of operations. For test no. 7 at which the maximum flow (175 m$^3$/s) passed only from three tunnels (case of maintenance for one tunnel), the required upstream and downstream water levels were not verified. The reason of this refers to the increase of velocity inside the tunnels, which led to the increase of losses.

- The vertical distribution of the velocities inside the tunnels was checked. The results indicated that these distributions are considered ideal as in case of uniform flow in closed conduits. This means that there will not be a direct effect of the velocity on the syphon body. The minimum value of the velocity was found nearly 0.80 m/s, which is considered enough to prevent the deposition of sediment particles, which have $D_{50}$ less than 0.05 mm inside the syphon.

- The energy head at each pressure cell installed above tunnel no. 1 was computed (potential head, pressure head, and velocity head). The total head loss between the inlet and outlet structure was computed by subtracting the energy head at the outlet structure from the energy head at the inlet structure. The losses were within the permissible range in all tests except for the case of operating three tunnels with the maximum discharge.

- The round piers of the outlet structure created separation areas downstream them. The flow at the end of the transition area, downstream the structure was non-uniform and flow recirculation was observed. The main reason of this non-uniformity was the angle of the vertical transition walls immediately downstream the syphon outlet.
2. Recommendations

- Generally the design of the inlet structure is acceptable although a separation zones were created by the piers edges and the transition wall as they considered have very small effect in the inflow pattern.

- It should take into consideration that in case of passing the maximum discharge from only three tunnels (maintenance of one tunnel) and to verify the maximum downstream water level (195.43 m), the upstream water level should be 195.23 m instead of 194.97 m.

- It is recommended to construct the proposed sediment trap and it should be monitored each 6 months to remove the accumulated sediment if necessary.

- Due to the limitation of the sediment trap surface area it is also recommended to equip each tunnel by two steel rods fixed in its bed a long the whole length so that a small trolley can move easily inside the tunnel which can used for collecting the accumulated sediment inside the tunnel. A screen filter at the entrance to the canal is also recommended as a safety device to filter any debris picked up by the pumps.

- The horizontal angle of the two vertical transition walls downstream the syphon outlet should be 13° instead of 30°. This is to improve the flow uniformity and reduce the flow recirculation. But still there is some nonuniformity and recirculation of the flow especially in case of the operation one or two tunnels in the middle. So, in case of two tunnels in operation, it is recommended to operate the tunnels, which lie in the out sides (tunnel no. 1 and 4).

- It is recommended after a certain time of the construction of the syphon is to measure the discharge by means of velocity-area method along two cross sections at the same time (one in the upstream and the other in the downstream of the syphon). The purpose of these measurements is to determine weather there is a seepage from the syphon or not and to compute its value if necessary. These measurements should be carried out for each syphon in operation alone.

ACKNOWLEDGMENTS

This paper was derived from a published technical report performed in the Hydraulics Research Institute (HRI) of the National Water Research Center (NWRC), Egypt. The report was published in May 2004. The author would like to express his appreciation to Prof. Dr. M. Gaweesh (Director of HRI) and Eng. I. El-Desouki (Deputy Director), for their support and assistance in formulating this paper and their valuable comments, which led to improvements in the final manuscript.
REFERENCES

Book

Published Reports and Journals


