

**AN EXPERIMENTAL STUDY OF FACTORS  
INFLUENCING THE SCOUR DEPTH RESULTING FROM  
THE COLLISION OF WATER JET IN STILLING BASIN  
FLOOR**

دراسة مختبرية للعوامل المؤثرة على عمق النحر الناتج من ارتطام بثق الماء في  
حوض مذب مائي

Fadhil M. Al- Mohammed<sup>1</sup>, Ali T. Jassim<sup>2</sup>, Hamid Adil Abbas<sup>3</sup>

<sup>1</sup> Kerbala Technical Institute, Al-Furat Al-Awsat Technical University,  
56001 Kerbala, Iraq.

Water resources engineering

<sup>2</sup>Department of Civil Engineering, Kufa University

<sup>3</sup>Kufa University

**Abstract:**

When excess water is discharge from a reservoir into the downstream water body, it sometimes flows in the form of a jet. This jet will have a high kinetic energy making it able to transport the sediment particles downstream of the impinging area, thereby forming a scour hole. The purpose of conducting this study was to evaluate the depth of the local scour result by free water jet based on the laboratory experiments and numerical analysis FLOW3D. One hundred and thirty-eight runs were carried out to investigate the local scour downstream the hydraulic structure under clear-water conditions with different discharges, tail water depths, and jet angles. Two types of jets were used, normal jet and dispersion jet. Results indicated that maximum scour depth increase with increasing densimetric Froude number and jet velocity. The increase in tail-water depth increases the scour depth until access to a critical depth at which the maximum depth is occurring after that the scour starts downward when the depth is overstepping the critical value. The optimum jet angle depends on the jet discharge, with a value of 20° for jet discharges 60-80 L/min, and 40° for Jet discharges 100 and 120 L/min, which cause minimal depth of scour. The scour depth values under dispersion jet are smaller than those under normal jet. Generally, the reduction percentage increases with the increase of jet discharge, as it 30.7% and 78.7% for discharge 80 and 120L/min respectively. The numerical simulation was used by (FLOW3D) software program to investigate from the result of the experimental work, the results showed that there was a very good agreement with the experimental work with average 95.25%.

**Keywords:** local scour, plunge pool, free water jet, numerical analysis, FLOW3D.

## الخلاصة

عندما يتم تصريف الماء الزائد من خزان الى مذب النهر، يكون في بعض الاحيان على هيئة بثق مائي. هذه المياه تكون ذات طاقة كامنة عالية تجعلها قادرة على نحر المذب ونقل الترسبات وحبيبات القاع بعيدا عن المنشأ الهيدروليكي مخلقة حفرة في اسفله. ان الغرض من اجراء هذه الدراسة هو لحساب عمق النحر الناتج من البثق المائي اعتمادا على التجارب المختبرية والتحليل العددي بالاستعانة ببرنامج FLOW3D. تم اجراء (138) تجربة مختبرية للتحقق من عمق حفرة النحر أسفل المنشأ الهيدروليكي تحت ظروف الجريان الخالي من حمل الرسوبيات (Clear water) ولقيم متعددة للتصريف وللماء الذيلي ولزوايا البثق. كما تم استخدام نوعين من البثق هما الاعتيادي والمشتت. اظهرت النتائج بان عمق حفرة النحر تزداد بزيادة رقم فرود الكثافي وسرعة البثق. الزيادة في عمق الماء الذيلي يزيد من عمق حفرة النحر لحين الوصول لعمق الماء الذيلي الحرج، الذي يكون عنده عمق حفرة النحر في قيمته العظمى، وبعد ذلك يقل عمق النحر بزيادة عمق الماء الذيلي. كما بينت النتائج بان أفضل زاوية للبثق تعتمد على سرعته، للتصريف 60-80 لتر/ثا فان أفضل زاوية للبثق (ينتج عنها اقل عمق نحر) هي 20° بينما كانت للتصريف بين 100 و 120 لتر/ثا هي 40°. ان قيم عمق النحر في حالة البثق المشتت اقل من قريباتها في البثق الاعتيادي ويعتمد ذلك على قيم تصريف البثق حيث كانت نسبة تقليل عمق النحر عند استخدام البثق المشتت للتصريف العالية والقليلة (78.7%) (30.7%) على التوالي. كما أظهرت النتائج توافقا كبيرا بين نتائج قيم عمق النحر المختبرية للحالات المختلفة وتلك المتحصلة باستخدام أسلوب التحليل العددي بواسطة برنامج FLOW3D وبمعدل 95.25%.

## 1. Introduction

When excess water is discharge from a reservoir into the downstream water body, it sometimes flows in the form of a jet. The jet is an intensified flow, where water with high velocity discharge into fluid downstream, where the fluid either be static or in moving. This high energy water jet can lift and transport the sediment particles downstream of the impinging area, thereby forming a scour hole. As well, the stability of the foundation of a hydraulic structure may undermine because of generating the local scour downstream of these structures due to of the flow in the form of turbulent water jets. Examples of these structures, local scour downstream of a flip bucket, vertical gates, spillways, weirs, drops, culvert, and downstream outlet. In all cases, the complete protection against scouring is often impractical and expensive. Therefore, it is important for hydraulic engineers to understand the scour mechanism to find appropriate solutions and ways to guide and control the scour process so as to reduce the risk of failure.

## 2. Literature review

Some studies have been published on the scouring of cohesionless bed sediment downstream hydraulic structure. [1] studied the influence of submergence on the scour processes using three dimensional jet release by a square cross-section nozzle on a non-cohesive sand bed, three different tail-water depths, and conforming to  $2b_0$ ,  $4b_0$  and  $6b_0$ , where  $b_0$  is a nozzle width. Results indicated that the tail-water depth, densimetric Froude number  $F_0$ , and grain size-to nozzle size ratio, all has an effect on the scour hole extent caused by three-dimensional jet. The results also indicate that at very low tail-water depths, for  $F_0 < 5$ , the jet expansion ratio can have a significant effect. The maximum scour depth isn't necessary deeper at the lower range of submergences, where the depth was large at higher tail-water depths. Also it must be observed that the volume of scour hole is large at higher tail-water depths. [2] studied the effect of many parameters on plunge pool scour of a totally incoherent rock surface. The angles of jet collision changed between 30 and 90°, the result showed that the scour hole depths increase with the jet angle whereas the ridge height decreases with increase the jet angle. [3] studied the local scour in noncohesive sediment downstream of a horizontal rigid apron. Two kinds of bed material were used in the experiments with median size diameters of 0.73 mm and 1.85 mm. For the scour hole, maximum of scour depth, location of maximum scour depth, maximum length of scour, maximum height of scour, and location of maximum ridge height for all the experiments have been measured. Results showed that there are good correlations among scour hole factors. It was found that the maximum extension of the scour

hole, the maximum dune height, the maximum extension of the dune, will increase with the increasing of the maximum depth of scour. [4] studied the local scour due to free fall jets. Four kinds of sediment mixture were used. The results showed that by increasing the geometric standard deviation of the particles, scour hole parameters and the ridge height decrease. The decreases in the maximum scour depth and the height of downstream ridge by increasing the geometric standard deviation from 1.3 to 2.7 is about 32% and 45% respectively. Moreover, the scour hole length and width decrease by about 30% as the geometric standard deviation change from 1.3 to 2.7. [5] studied the local scour by three-dimensional wall jet great by a nozzle with square cross section. Three various tail-water depths were used and the jets expansion ratio was larger than 10. It was noted that coarser particles are deposited towards the downstream end of the scour hole. In fact, the coarsest particles are existing on down slope of the ridge. Also they observed that the volume of scour hole increase with increases of time. However, for 72 hours of test, there is a large increase in scour volume when compared with  $t$  equal to 48 hours. Also the volume of scour hole be increasing with increasing tail-water depth where it's larger for  $H/b_0$  equal to 6 compared with  $H/b_0$  equal to 4 and 2 where  $H$  is a depth of water in the channel.

### **3. Objective**

The present study aims to carry out laboratory experiments and numerical simulation (FLOW3D) on the physical models to investigate and evaluate the scour depth resulting from the collision of water jet in stilling basin floor when using different jet discharges, tail water depths, and angle of flow released from the outlet.

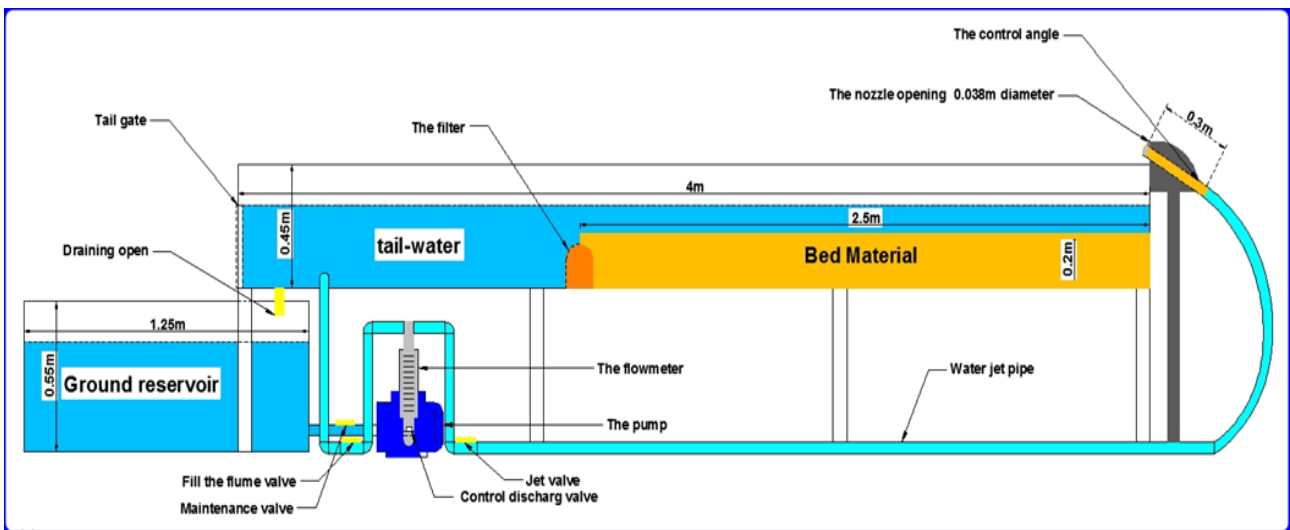
### **4. Experimental work**

The experiments were conducted using a locally made flume. The flume was made from galvanized iron with a long 4 m, width 0.8 m, and height 0.45 m as shown in figure 1. The flume contains a settling basin with 2.5 m long and 0.2 m height that filled with uniform sediment sand. In the end of the flume, there is a gate with a length 0.6 m used to control the tail-water level above the sand bed by setting it on the required level and the water passed on it as a wear. The ground reservoir located at the end of the flume, this reservoir used to store and recirculating the water to the flume. A pump type streen (2 HP) used to draw the water from the ground reservoir to jet nozzle where the water jet is generated. The discharge is controlled by a flowmeter installed vertically after the pump to obtain the required flow at the jet pipe. The jet installation system consists of a circular iron pipe with 3.8 cm inner diameter and 30 cm long installed at the beginning of the experimental flume using a perforated metal plate at an altitude of 27 cm from the sand bed and it's fixed on a plate of iron by tow bolts to control the angle of the jet where the plate was screwed into several holes to control the angle of the jet. The configuration of the sour hole and the maximum depth of scour is taken by using a tool named "measurements mesh". This tool made locally by a mold of wood with a metal mesh. The mesh openings are in is a square shape with rib length is 5 cm as shown in figure 2. In order to find a limited scour time and adopt it in all tests, thirty preliminary tests were made at lowest tail-water and angle of a jet by using tow discharges 120 and 100 L/min. Maximum depth of scour was recorded after each run by using a measurement mesh and measurement tube at different time intervals. These tests show that a scour depth has sharply increased during first 20 minutes of the test duration and the development of scouring has become increases gradually and with deceleration. After two hours, we observed that the increase in scour depth becomes very slowly, so the selected time for all test runs is two hours.

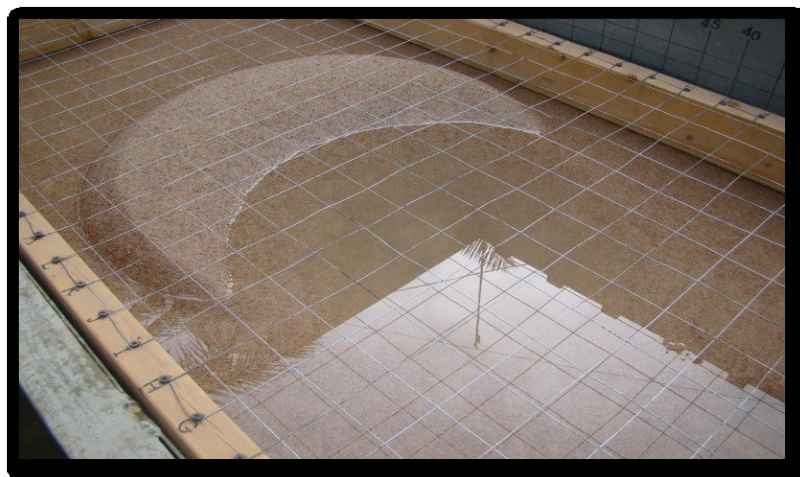
### **5. Normal jet and dispersion jet**

Two types of jet were used in the present study, normal jet NJ, and dispersion jet DJ. The classification of the jet depends on the type of a nozzle. The first type represents the natural state where the water extrude from the nozzle freely, while the second type was made locally by inserting

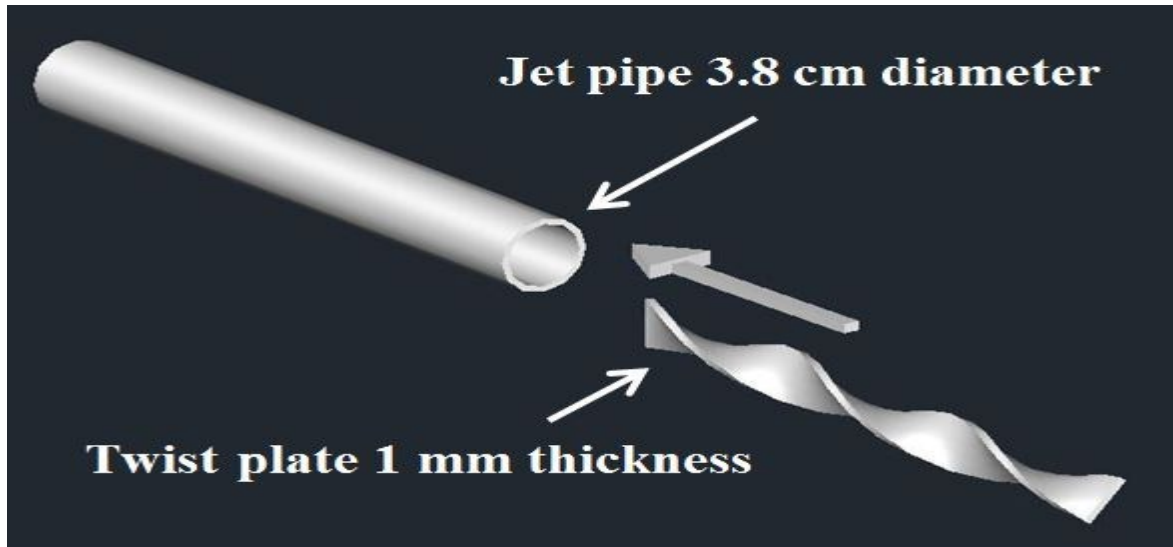
a twisted plate inside the pipe of the jet as shown in figure 3. The stuffing of the twisted plate in an iron pipe works to force the flowing water to move spirally inside the pipe, which helps to spread the water when leaving the nozzle because of the characteristic of centrifugation generated in the flowing water. One hundred and twenty runs were conducted for NJ under different hydraulic condition which contains four discharges (60,80,100,120) L/min, six tail-water depths (5,7.5,10,12.5,15,17.5) cm, and five angles for the jet (10°,20°,30°,40°,50°). Also, additional eighteen runs were conducted, nine tests for each NJ and DJ under same hydraulic conditions, one angle of 30°, three tail-water depths (5, 10, 15) cm, and three various discharges (80,100,120)L/min.



**Figure (1): Sketch for the experimental model detail.**



**Figure (2): Measurements mesh.**



**Figure (3): Sketch for the dispersion jet system.**

## 6. Dimensional analysis

For clear water condition, the maximum depth of scour hole results by free water jet is a function of the following parameters:

$$d_s = f\{ y_t, v, R, d_{50}, \emptyset, g, \rho, \rho_s, \mu, \sigma_g, B, S_o, T \} \quad (1)$$

where,  $y_t$  = tail-water depth;  $v$  = jet velocity ,  $R$  = hydraulic radius ,  $d_{50}$  = median sediment size ,  $\emptyset$  = angle of jet ,  $g$ = gravitational acceleration ,  $\rho$ = density of the fluid ,  $\rho_s$  = density of the sediment ,  $\mu$  = dynamic viscosity of the fluid ,  $\sigma_g$  = standard deviation of particle size distribution ,  $B$  = channel width ,  $S_o$  = channel bed slope ,  $T$  = duration of test. By using dimensional analysis equation 1 can be rewritten as:

$$\frac{d_s}{y_t} = f\left\{ \frac{R}{y_t}, \emptyset, Re, \frac{\rho_s}{\rho}, \frac{d_{50}}{y_t}, Fr, \sigma_g, \frac{B}{y_t}, S_o, \frac{vT}{y_t} \right\} \quad (2)$$

Where  $Re$  = Reynolds number ,  $Fr$  = Froude number

After the simplification the above equations and eliminating the parameters with constant and negligible values and applying the assumption, the functional relationship which describes scour parameter normalized with tail-water depth may be written as:

$$\frac{d_s}{y_t} = f\left( \frac{R}{y_t}, F_o, \emptyset \right) \quad (3)$$

Where  $F_o$  is densimetric Froude number and equal to  $v / \sqrt{gd_{50}\Delta\rho / \rho}$  . Where  $\Delta\rho = \rho - \rho_s$

## 7. Computational fluid dynamics (FLOW3D)

In the present study, the software FLOW3D program was used. It is a CFD package mix between before and after process apparatus. The objective from the numerical simulation in this study is to compare between the experimental and numerical method for NJ state and to find out the compatibility between the two procedures. After that, numerical simulation for rectangular jet

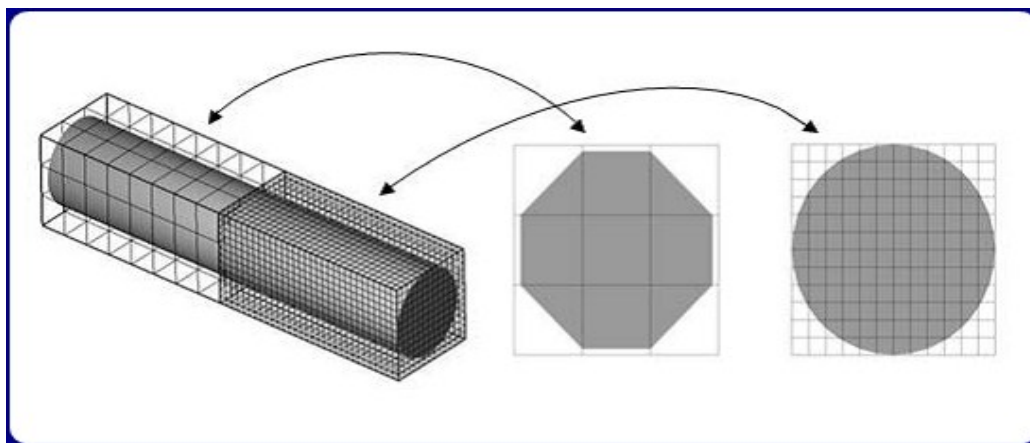
(plane jet) is being done with aspect ratio in the same area of cross section of circular nozzle under same conditions of the jet and compared the result of numerical simulation for circular jet and plan jet.

### **7.1.Numerical model**

Many different physics options are available, only four options were selected is required to get a delicate simulation of the data where it's required in this study, an acceleration of gravity, a sediment scour model with median size particle diameter of 0.81 mm, the viscosity, and the turbulence options. The Geometry of the model represented by a rectangular flume contains a barrier in the upstream to hold the water and the barrier contains circular hole launches water to the downstream in a form of a jet where can be controlled in the angle of the jet.

### **7.2.Meshing**

In FLOW3D, for accurate solution, grid is one of the most important matter. Cell size and grid can influence both on the simulation time and on the results accuracy. If the sizes of the cell are not optimal, geometry problems will happen as shown in figure (4) (FLOW3D manual, 2014).To avoid this problem, FAVOR option helps us to obtain delicate geometric shapes.

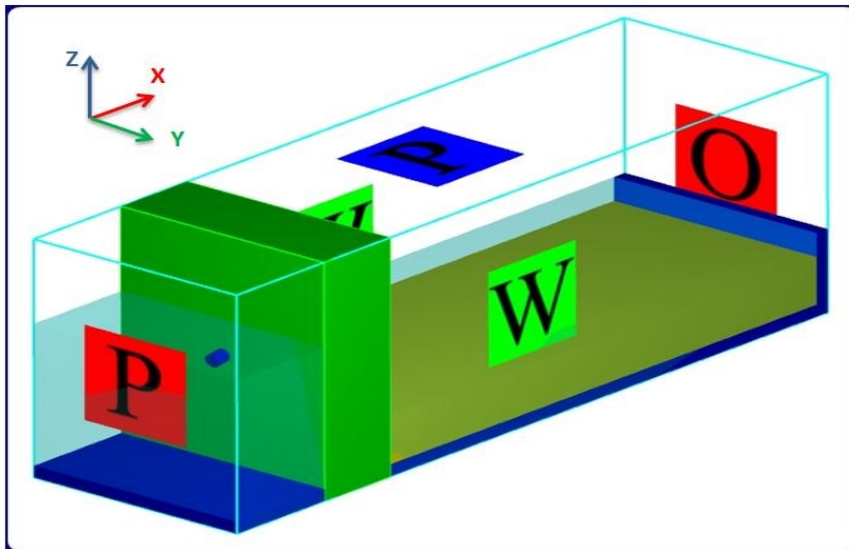


**Figure (4): FAVOR option with different cell size.**

The mesh of FLOW3D software have a cell of cubic shape and considering one of the affecting factors on the simulation process, therefore, different cell sizes are selected as (20, 16, 12, 10 and 8 ) mm to identify the best cell size that satisfies the phenomenon conditions. The best number of cells was 1000000, which the corresponding size of cells was 12 mm were used in this study when the comparison was made among the output results.

### **7.3. Boundary and initial conditions**

In numerical flow analysis, one of the most important stages is the determination of the appropriate boundary conditions. The boundary condition should correspond to the physical condition of the problems duly, on rectangular mesh prism, there are five various types of boundaries were used as shown in figures 5. These boundaries are:



**Figure (5): Boundary and initial conditions.**

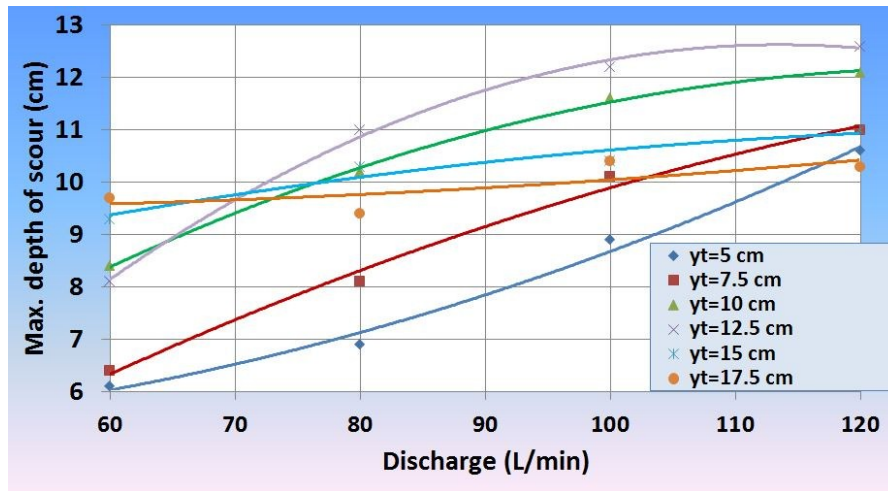
- 1-Upstream boundary (X-min): specified pressure condition (P) used fluid elevation where the head of fluid upstream of the flume is determined based on the Bernoulli equation to get the required discharge for the jet.  $v = \sqrt{2g * y}$  where  $v$  = velocity of the water (m/s),  $y$  = hydraulic head (m).
- 2-Downstream boundary (X-max): An outflow condition (O).
- 3-Top boundary (Z-max): Specified Pressure condition (P) used a fluid fraction and it represent the atmospheric pressure.
- 4-Bottom boundary (Z-min): Wall condition (W) its represent the ground under the packed sediment
- 5-Side boundary(Y-min, Y-max): Wall condition (W) its represent the wall of the flume

## **8. Results and discussion**

This section presents results and analysis of experimental tests that carried out to investigate the influence of nozzle discharge, tail-water depth, angle of the jet, and type of jet on the maximum depth of scour downstream hydraulic structure. It should be mentioned that by focusing the effect of any parameter, the same experimental conditions are held fixed. Thus, the only effect of that parameter can be specified.

### **8.1. Effect of discharge**

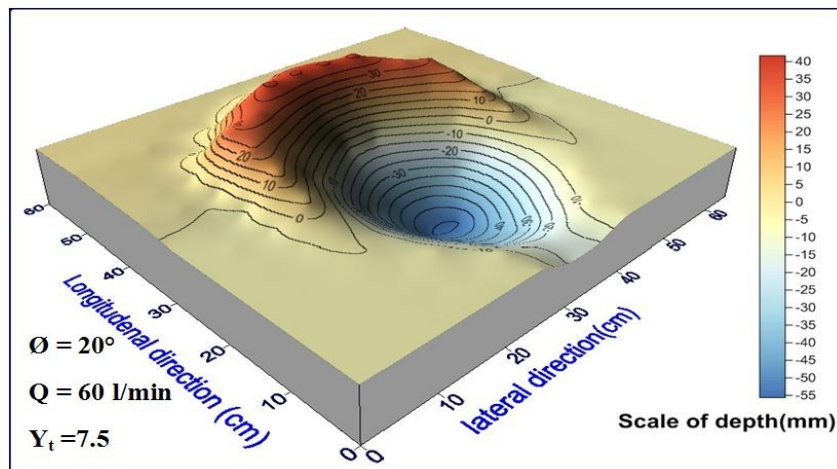
The values of discharge are plotted against the maximum depth of scour for all values of tail water depth and angle of the jet. Figure (6) shows the effect of varying the discharge on the depth of scour hole for an angle of jet equal to 30°.



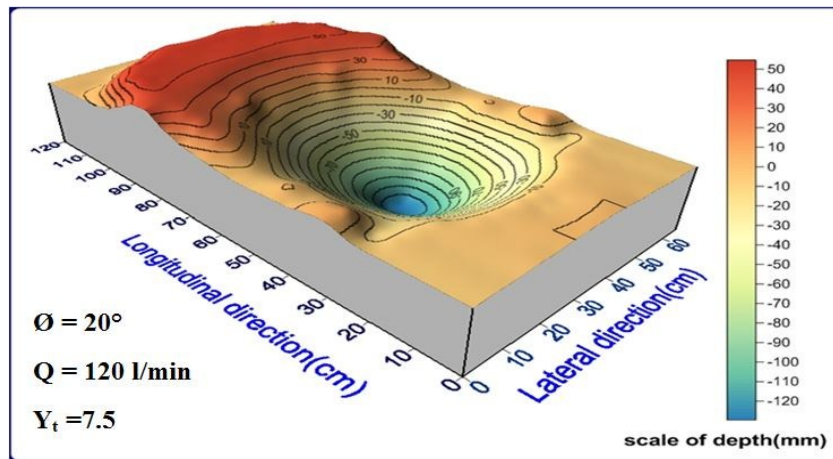
**Figure (6): Development of the maximum scour depth with the discharge at angle of jet ( $\theta$ ) = 30 °.**

It is clear from the figure that the maximum scour depth increases with increasing the discharge for all tail-water depth. The increase in discharge leads to an increase in the water velocity, which leads to increasing its kinetic energy and subsequently increase the depth of the scour hole. Also it's clear that with increase the tail-water depth the effect of the discharge on the maximum depth of scour is reduced.

For more elucidation to the influence of discharge 3D configuration shape for scour hole was drawn by Surfer 10 program as shown in figures 7 and 8 for  $Q = 60$  and  $120$  L/min respectively.



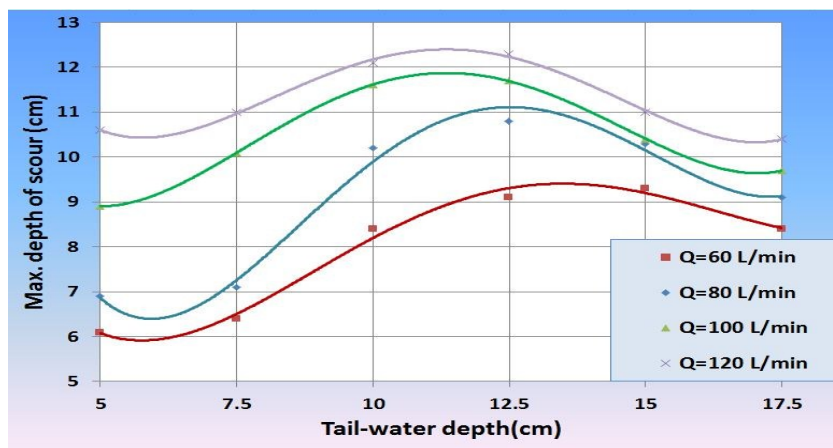
**Figure (7): The contour map and 3D configuration of scour hole for the normal jet.**



**Figure (8): The contour map and 3D configuration of scour hole for the normal jet.**

### 8.2. Effect of tail-water depth

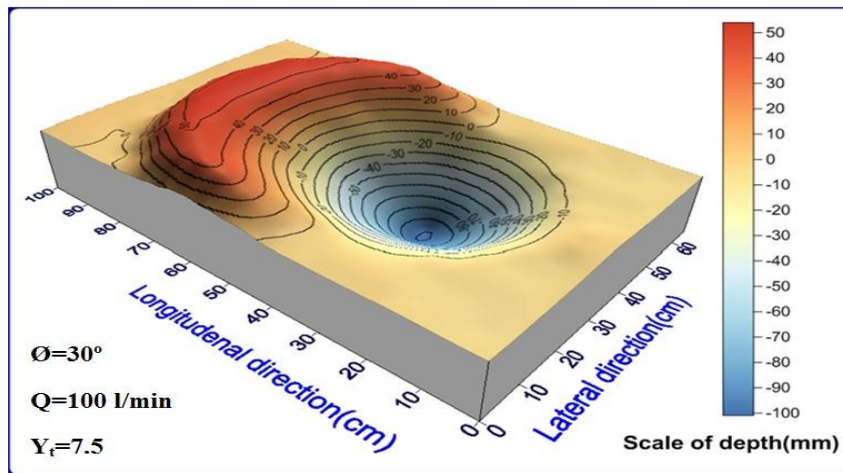
The influence of tail water depth on the depth of scour was investigated by using six levels of tail water depths. Figure 9 show the effect of varying the tail-water depth on the maximum depth of scour hole for all discharge and for the angle of jet equal to 30°.



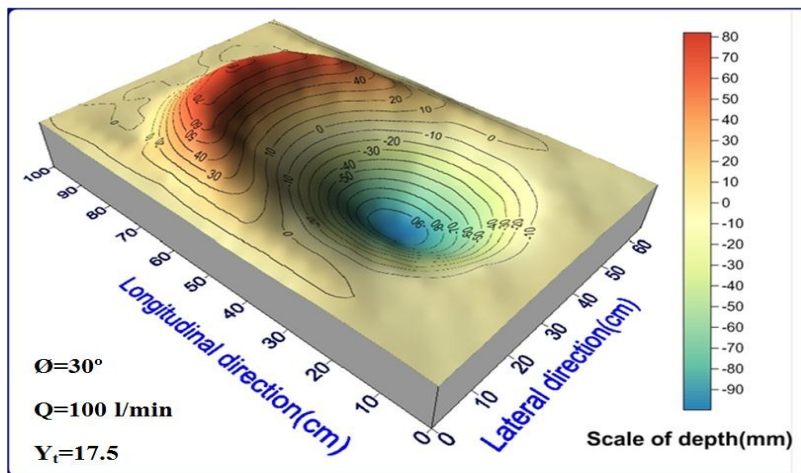
**Figure (9): Development of the maximum scour depth with the tail water depth for angle of jet  $\theta=30^\circ$ .**

Figure 9 shows that the greatest value of scour depth occurs when the value of the tail water is 12.5 cm. The value of the depth of the tail water at which the greatest value of scour depth can occur called the critical tail water depth and the change of the tail-water depth above or under the critical value caused decreases in the depth of scour as mentioned by [6] and [7]. In a low tail-water level, the depth of scour is small and that because of the dynamic movement of particles that happen under these conditions, which represent by digging and refilling processes. Refilling process occurs because of there is not enough flow to transfer particles away from the scour hole and this sequent digging and refilling processes at low tail-water depth confirmed by [7].

The configurations of scour hole were recorded by using measurement mesh results. They were painted by Surfer 10 to show the effect of tail-water depth on the scour hole. Figure 10 and 11 show the difference in the scour hole for tow tail-water depth (7.5, 15) cm.



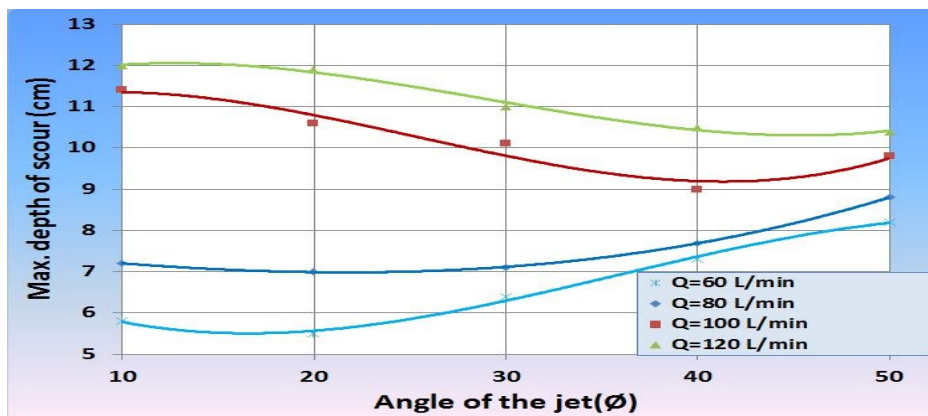
**Figure (10): The contour map and configuration of scour hole for the normal jet.**



**Figure (11): The contour map and configuration of scour hole for the normal jet.**

### 8.3. Effect of the angle of jet

The effect of the jet direction on the maximum depth of scour hole was studied by using 5 angles ( $10^\circ$ ,  $20^\circ$ ,  $30^\circ$ ,  $40^\circ$ ,  $50^\circ$ ). Figure 12 shows the maximum depth of scour hole for all discharges and for tail water depths equal to 7.5.



**Figure (12): Development of the maximum scour depth with the angle of the jet for tail water depth  $y_t=7.5 \text{ cm}$ .**

For discharges 60 and 80 L/min the best angle of the jet that causes little maximum depth of scour was 20° and for discharge equal to 100 and 120 L/min the best angle of the jet that causes little maximum depth of scour was 40°. It can be seen that at the low and high jet velocities, the favorite jet angle are 20° and 40°, respectively.

#### **8.4. Comparison of normal and dispersion jets**

This paragraph deals with a comparison between the effects of NJ and DJ on the scour depth. Table 1 shows results of the comparison of scour depth for 18 runs, nine runs for each jet type.

**Table (1): Comparison in maximum depth of scour between Dispersion jet and normal jet.**

Q (Lmin)	y <sub>t</sub> (cm)	Depth of scour (cm)		Difference(cm)	decrease %
		NJ	DJ		
80	5	6.2	3.8	2.4	38.7
100	5	10.1	7	3.1	30.7
120	5	11.4	7.5	3.9	34.2
80	10	8.6	5.1	3.5	40.7
100	10	11.6	5.6	6	51.7
120	10	12.3	4.8	7.5	61
80	15	8	3.4	4.6	57.5
100	15	10.5	2.7	7.8	74.3
120	15	10.8	2.3	8.5	78.7

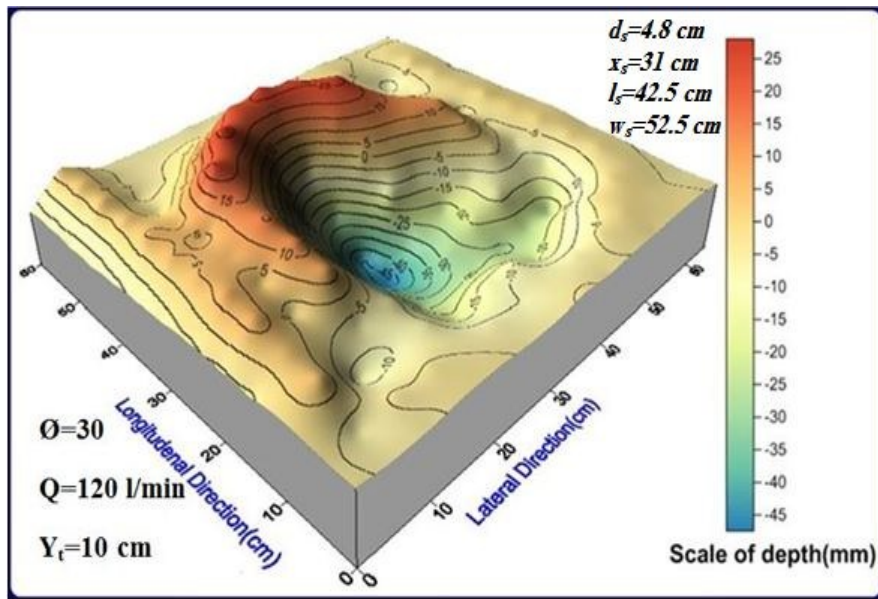
Generally, the results explained that the application of DJ technique for the water extruded from the nozzle considered the most important factor affecting the dimensions of the scour hole especially the depth of scour hole. The percentage of decrease in the scour depth varies between 36.56% and 44.28%.

The greatest effect of the type of jet on the scour hole depth occurs at high values for discharges. Increasing the tail water depth for same discharge leads to increase the difference in the scour depth for NJ and DJ.

This due to the great depth of the water dispersed the energy more effectively in the case of Dispersed jet. The contour map and 3D configuration of scour hole for dispersion jet are shown in figure 13.

#### **9. FLOW3D Results and discussion**

This section aims to compare results of the laboratory experiments of plunge pool results by free water jet were held previously in the laboratory with numerical simulation results of CFD code. To test the effectiveness of the numerical model, this numerical model was applied in similar conditions of the physical model. A twenty laboratory experiment was selected for testing it in the FLOW3D software program.



**Figure (13): The contour map and 3D configuration of scour hole for dispersion jet.**

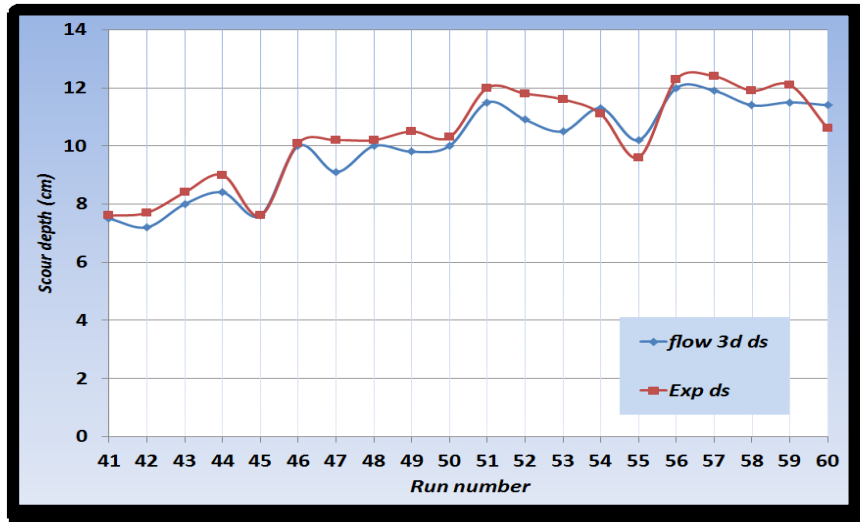
So, we will explain how much compatibility in the maximum depth of scour hole result from FLOW3D software program and the maximum depth in the laboratory work. The results and comparing of the maximum depth of scour hole in the flow 3D and experimental data are shown in Table 2.

Results below shows a good agreement between the experimental work and FLOW3D software program for the maximum scour depth where the average of the error equal to 4.75% that's meaning the agreement equal to 95.25%.

**Table (2): The comparing of maximum scour depth between flow 3D and experimental work.**

Run no	Exp $d_s$ (cm)	FLOW3D $d_s$ (cm)	Difference(cm)	Abs error %
41	7.6	7.5	0.1	1.31
42	7.7	7.2	0.5	6.49
43	8.4	8	0.4	4.76
44	9	8.4	0.6	6.66
45	7.6	7.6	0	0
46	10.1	10	0.1	0.99
47	10.2	9.1	1.1	10.78
48	10.2	10	0.2	1.96
49	10.5	9.8	0.7	6.66
50	10.3	10	0.3	2.91
51	12	11.5	0.5	4.16
52	11.8	10.9	0.9	7.62
53	11.6	10.5	1.1	9.48
54	11.1	11.3	-0.2	1.80
55	9.6	10.2	-0.6	6.25
56	12.3	12	0.3	2.43
57	12.4	11.9	0.5	4.03
58	11.9	11.4	0.5	4.20
59	12.1	11.5	0.6	4.95
60	10.6	11.4	-0.8	7.54

Also, the results of the maximum scour depth for experimental work and FLOW3D are plotted for more comparison as shown in figure 14.



**Figure (14): Comparison of maximum scour depth between experimental work and FLOW3D.**

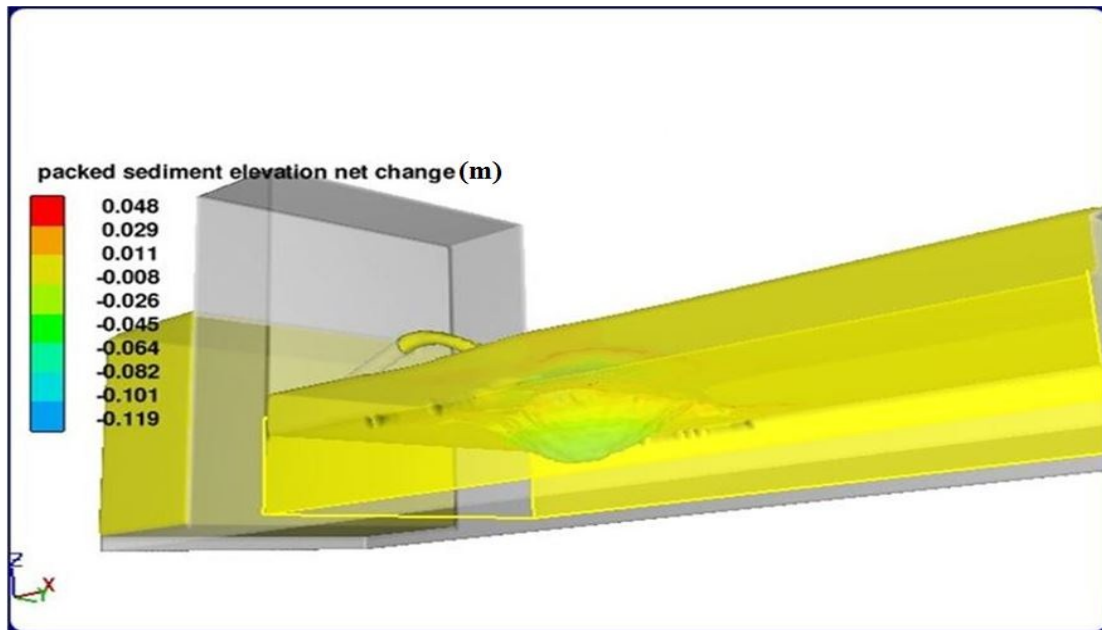
Consequently, the results numerical simulation are very comparable to the results of experimental work especially in relation to the maximum depth of scour. Thus we can rely on the flow 3d program to extract good results and depending on this, tested has been performed for plane jet under the same conditions and with the same cross-sectional area for rectangular slot with dimensions (2\*5.7) cm for comparison between it and circular jet.

Table 3 show the comparison between the results of the maximum depth of scour for circular jet and plane jet.

**Table (3): The comparing of maximum scour depth for flow 3D between the circular jet and plane jet.**

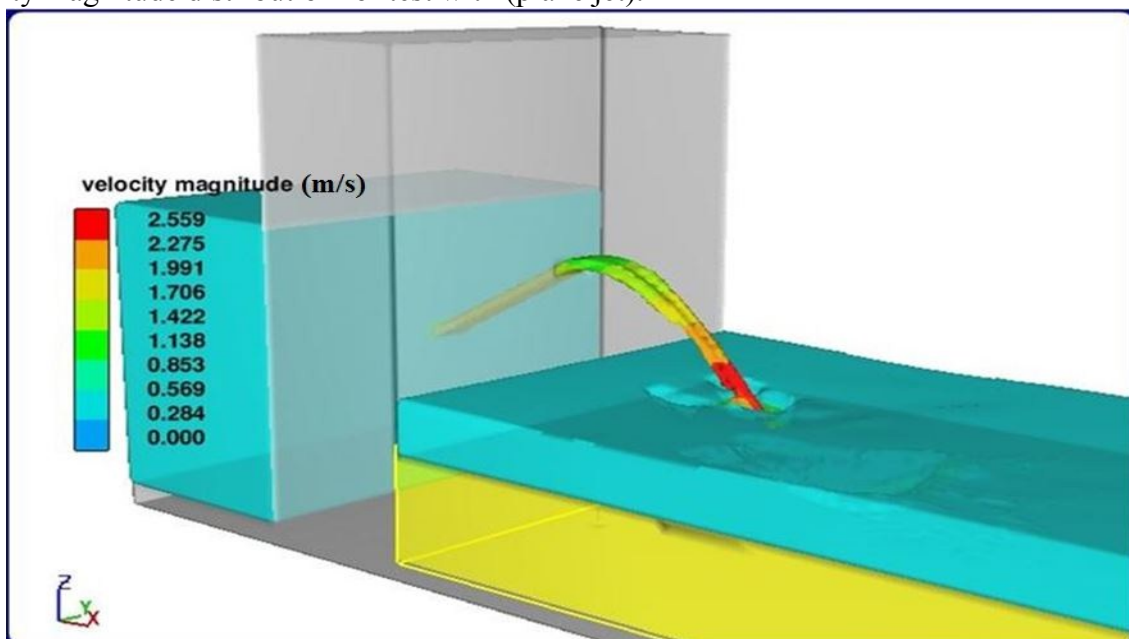
Run no	Cir jet $d_s$ (cm)	plane jet $d_s$ (cm)	Difference(cm)	Abs error %
41	7.5	5.8	1.7	22.66
42	7.2	6.8	0.4	5.55
43	8	7.5	0.5	6.25
44	8.4	6.7	1.7	20.23
45	7.6	6.9	0.7	9.21
48	10	9	1	10
47	9.1	6.6	2.5	27.47
48	10	8	2	20
49	9.8	7.5	2.3	23.46
50	10	7.3	2.7	27
51	11.5	9.1	2.4	20.86
52	10.9	8.9	2	18.34
53	10.5	8.4	2.1	20
54	11.3	7.8	3.5	30.90
55	10.2	7.6	2.6	25.49
56	12	10.7	1.3	10.83
57	11.9	11.1	0.8	6.72
58	11.4	9.2	2.2	19.29
59	11.5	10.2	1.3	11.30
60	11.4	8	3.4	29.82

From the table above we observed that the results of the maximum depth of scour for the plane jet were less than in the case of the circular jet with an average ratio equal to 18.26%. Figure 15 shows the three-dimensional fluid fraction output of numerical model for the maximum scour depth for test with (circular jet).



**Figures (15): Three-dimensional out but for maximum depth of scour result by circular jet (fraction of fluid).**

Also, figure (16) show the fluid fraction three-dimensional output of numerical model for velocity magnitude distribution for test with (plane jet).



**Figure (16): Three dimensional out but for velocity magnitude distribution in the plane jet (fraction of fluid)**

## **10. Development of new formulas**

The computer package (SPSS v21.0) was used to develop the equations through a non-linear regression analysis.

For depth of scour:

Equation 3 can be written as below:-

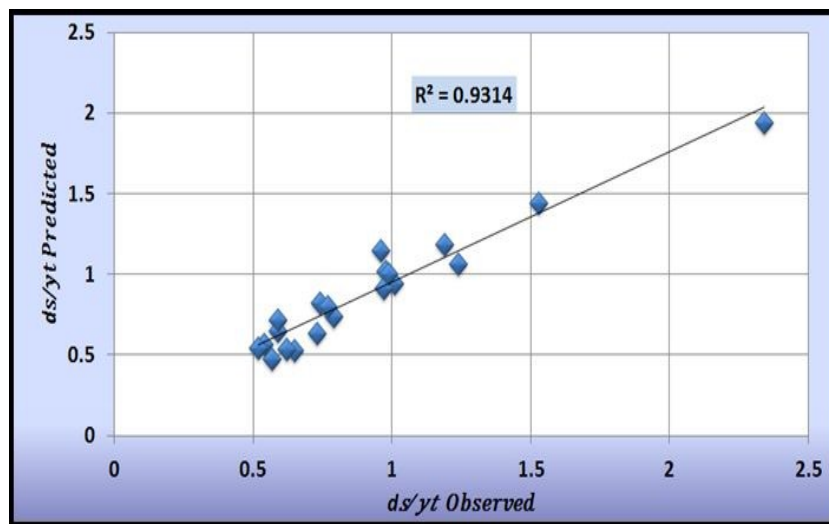
$$\frac{d_s}{y_t} = c_1(F_o)^{c_2} \times \left(\frac{R}{y_t}\right)^{c_3} - c_4 \cos \emptyset \quad (4)$$

$$c_1 = 0.954 \quad c_2 = 0.604 \quad c_3 = 0.751 \quad c_4 = 0.015$$

So, the equation becomes:

$$\frac{ds}{y_t} = 0.954(F_o)^{0.604} \times \left(\frac{R}{y_t}\right)^{0.751} - 0.015 \cos \emptyset \quad (5)$$

Another data was used to test the equations and this by separating twenty values from a total 120 values randomly by the program. The values of  $\frac{ds}{y_t}$  were calculated from the equations 5 a statistical comparison with the experimental data that separating was used to show the convergence of the predicted to observed records see figure 17. The value of  $R^2$  was 0.9314.



**Figure (17): Comparison of equation (5) with experimental data.**

### 11. Conclusion

- 1- The maximum scour depth increase with increasing densimetric Froude number, flow rate, velocity of the jet.
- 2- The increase in tail-water depth increases the depth of scour until access to a critical depth at which the maximum depth is occurring after that the scour starts downward when the depth is overstepped a critical value.
- 3- There is no better angle for the jet in all cases where we found that each velocity jet favorites angle where cause minimum depth for scour, ranging between 20° for low velocity to 40° for high jet velocities, generally we found that the angle equal to 30° is the optimum angle.
- 4- Dispersion technique for the water jet decreases the maximum scour depth by a large percent especially in high discharges where drop to 369.56% less from the normal jet.
- 5- In CFD work the Comparison of the results between the numerical model and experimental model indicates about rate of error equal to % 4.75 for the maximum depth of scour hole this information show a good agreement between the numerical and experimental work, and thus can be relied on the flow 3d to inclusion of other variables.

6- Result of tests conducted in the numerical model for plane jet showed that the depth of scour less than in the case of the circular jet in a percent ranged between percent30.9 to 6.25%.

1. **12. References**

2. Faruque, M, (2004). **Transient local scour by submerged three dimensional wall jets**· effect of the tail-water depth. Msc thesis, University of Windsor
3. Pagliara, S, Hager, W. H, & Minor, H. E. ((2006). **Hydraulics of plane plunge pool scour**. J. Hydr. Engng. 132(5), 450-461.
4. H, Hamidifar, (2011). **Interrelationships between characteristic lengths of local scour hole**, International Transaction Journal of Engineering, Management, & Applied Sciences & Technologies. Vol 2, pp 355-364
5. Ghodsian,M, et al (2012) ,**Local scour due to free fall jets in non-uniform sediment**, Scientia Iranica .19 (6), 1437–1444
6. Abdullah Al Faruque & Frank Hanna (2014). **Scour Caused by Three-Dimensional Submerged Square Wall Jet: Sand Deposition in Scour Hole and Ridge**, International Journal of Emerging Technology, vol 4, pp -323-333.
7. Aderibigbe, O, & Rajaratnam, N. (1996). **Erosion of loose beds by submerged circular impinging vertical turbulent jets**, J. Hydraulic Res, 34(1), 19-33.
8. Ghodsian, M, Melville, B, & Tajkarimi, D. (2006). **Local scour due to free overfall jet**, Ins **Civil Engineers**, Water Management. 159(4), 253-260.

*Symbol used*

<i>Symbol</i>	<i>Description</i>	<i>Dimensions</i>
<b>d<sub>50</sub></b>	Median particle size which is taken as the representative particle size	L
<b>d<sub>s</sub></b>	Maximum depth of scour hole	L
<b>F<sub>o</sub></b>	Densimetric particle Froude number for d <sub>50</sub>	-
<b>g</b>	Gravitational acceleration	LT <sup>-2</sup>
<b>Q</b>	Discharge of the jet	L <sup>3</sup> T <sup>-1</sup>
<b>R</b>	Hydraulic radius	L
<b>V</b>	Mean velocity of approach flow	LT <sup>-1</sup>
<b>y<sub>t</sub></b>	Tail-water depth	L
<b>Ø</b>	Angle of the jet	-
<b>μ</b>	Dynamic viscosity of water	ML <sup>-1</sup> T <sup>-1</sup>
<b>ρ</b>	Mass density of water	ML <sup>-3</sup>
<b>ρ<sub>s</sub></b>	Mass density of sediment	ML <sup>-3</sup>