XVIII International Conference on Water Resources
CMWR 2010
J. Carrera (Ed)
CIMNE, Barcelona 2010

THE NUMERICAL INVESTIGATION OF FREE FALLING JET'S EFFECT ON THE SCOUR OF PLUNGE POOL

Shahrokh Amiraslani*, Jafar Fahimi†, Hossein Mehdinezhad†

* Zanjan Regional Water Co., M.Sc, Department of Hydraulic Engineering, Islamic Azad University - South Tehran Branch, Tehran, Iran,
(Email: betonpaye@yahoo.com)

†Managing director of Zanjan Regional Water Co., B.S, Department of Irrigation and Drainage Engineering, Shiraz University

†Vice President of Zanjan Regional Water Co., B.S, Department of Civil Engineering, Power and Water University of Technology, Tehran, Iran

Key words: Free Falling Jet, scour, finite Volume, k-ε model, DOT Equation

Summary. A free falling jet is associated with an abrupt break in the river bed and in hydraulic structure such as dam outlets, cantilevered culverts, and pipe. The free falling whether in nature or in downstream of hydraulic structure can cause to form local scour. The most of researches on scour by free falling jet have done is laboratorial, and in some case 2D numerical simulation have done. In this comparison, the 3D scour process by free falling jet is simulated and bed deformation and scour pit in specific time domain is considered. Sediment is non- cohesive and water of jet in clear. For flow simulation, momentum equation and continuity equation is used. The *k-ε* turbulence model for simulation of flow turbulent in plunge pool is used. For computational domain a general 3D orthogonal grid in Cartesian coordinate are used and governing equations in scour process solved by finite volume method. Results of to obtain from 3D simulation are compared by results that presented with DOT (U.S. Department of transportation) equation. Also results shows that hole shape and downstream hill is completely similar to laboratory data .About depth, width and length of scour pit exist agreement between simulated sample and results that obtain by DOT equation.

1 INTRODUCTION

Using of free falling jet in hydraulic structure is one of the economical ways for energy dissipation. Structures exemplar outlet of culvert, orifices, overfall flow spillway have been seen a kind of jets. Although use turbulent flow in place jet enter in tailwater is useful factor to dissipation of energy. If downstream of this jet, have been a movable bed (bottom of river) can occur scour hole and damage for structure stability. A number of researchers have considered by experimental studying, for example, Rouse (1938), Laursen (1952), Bohan (1970), Ruffetal (1981), Rajratnam & Beltaos (1977), Milt& Kalin (1983), all of them that mentioned had studied in scour of outlet culvert. One of the earliest formulas defining the scour depth, proposed by Schoklitsch (1932) is

$$y_{me} + h_t = C_s \frac{q^{0.57} H^{0.2}}{d_{90}^{0.32}} \tag{1}$$

Where Cs is dimensional coefficient, d90 is particle diameter that do not exceed by 90% of sediment grains, h_t is tailwater depth, H is height between head and tailwater levels; q is discharge per unit width and y_{me} is equilibrium scour depth. One of the recent experimental research have been done by Pagliara & Hager (2006) general equation for maximum deep scour ($Z_m = \frac{z_m}{D}$) is expressed in Equation (2):

$$Z_m = f_1\,(Fd).\ f_2\,(\alpha).\ f_3\,(\beta).\ f_4\,(T).\ f_5\,(\delta).\ f_6\,(F_u) \tag{2}$$

densmetric Froud number:

$$F_d = V_W / (g' d_{90})^{1/2}, f1(F_d) = F_d$$

jet impact angle:

$$f_2(\alpha) = -\left[0.38 \sin(\alpha + 22.5)\right]$$

jet air entrainmentβ:

$$f_3(\beta) = (1+\beta)^{-m}$$

tail water effect T

$$f_4(T) = \left[0.12 \ln(1/T) + Cr\right] / 0.3$$

sediment non-uniformity σ

$$f_5(\sigma) = -\left[0.33 + 0.57\sigma\right]$$

upstream flow effect, F_u

$$f_6(F_u) = 1 + F_u^{0.5}$$

Because of expensive hydraulic models and limitation of domain and varieties experiments in laboratory, using of numerical methods is advantaged, and we can mention some of researchers like Hoffmens & Booij(1993), Olsen & Kjellesvig (1999, 1998), Olsen & Melaaen (1993) , Ushijima & Shimizu & 2D, 3D simulation of scour by free falling jet. All of equation that expressed depends on velocity of jet, height of tail water, discharge of jet, height of falling jet, angel of impacting jet, grain size of sediment, sediment uniformity or non-uniformity. In some of research, parameters like volume of air entrainment and submerged or unsubmeged in plunge pool have been considered.

In this research scour process in downstream in free falling jet have been simulated in 3D by flow 3d software for 300 second .properties of geometry hole scour include depth , height ,length is compared with valid experimental so ,variation of bed profile in this time obtained and compared with experimental results.

2. SCOURING BY FALLING JET

Scouring is to move the solid particles from initial situation and settling in another place effect of flow fluid. When the bottom shear stress is not sufficient to entrain the sediment particles in the flow, the deposition process begins. The stress for which there is an initial net deposition rate is called the critical stress. The deposition process depends on the critical stress, the concentration of suspended sediment, the settling velocity of the particles and the water depth. According to Alberston et al (1948) the falling jet can be divided in two distinct regions (fig.1) a zone of flow establishment [also potential core ,Rajratnam (1976)]and a zone of establishment is a wedge like region in which the flow velocity equal the efflux velocity . The velocity in diffused zone is less than velocity in potential core.

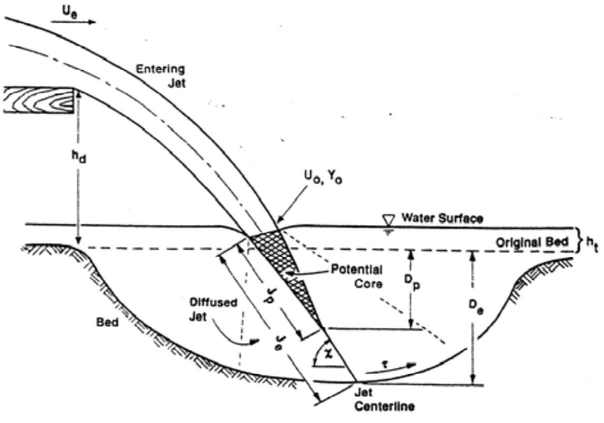


Figure 1. Plunge pool and free falling jet [3]

Turbulence flow in plunge pool to cause shear stress in sediment fluid interface, when shear stress excesses of critical shields parameter scouring is beginning

3. EXPERIMENTAL REASEARCH

In laboratory study a sediment pool is built by 2*1.5*0.75 dimension and discharged pipe with square section by 3*3 centimeter. Water is clear and sediment has uniformly graded with d_{50}=1.27mm, d_{84}=1.35mm, d_{16}=1.1mm, $\sigma=1.4<1.3$ properties. Falling height between middle of pipe and sediment bed level is Hc=35 cm and tailwater is Tw=6cm. scour process after 300min entered to equilibrium phase in interval 5, 10, 30, 60, 180, 300 min, dimension of hole scour (depth, width, length) and hill of downstream in middle axial of sediment pool is determined. Based on above mentioned experimented scour equation for prediction of dimension of hole scour has been presented which this equation are based on reaction offered by US department of transportation (DOT).

This equation is as fallowing Equation (3):

$$\begin{aligned}
\frac{h_s}{R_c} &= 1.05\left[\frac{2.27}{\sigma^{1/3}}\right]\left[\frac{Q}{\sqrt{g}\,(R_c^{\;2.5})}\right]^{0.39}\left[\frac{t}{316}\right]^{0.06} \\
\frac{W_s}{R_c} &= 0.88\left[\frac{6.94}{\sigma^{1/3}}\right]\left[\frac{Q}{\sqrt{g}\,(R_c^{\;2.5})}\right]^{0.53}\left[\frac{t}{316}\right]^{0.08} \\
\frac{L_s}{R_c} &= 0.48\left[\frac{17.1}{\sigma^{1/3}}\right]\left[\frac{Q}{\sqrt{g}\,(R_c^{\;2.5})}\right]^{0.47}\left[\frac{t}{316}\right]^{0.1}
\end{aligned} \tag{3}$$

In above equation h_s, W_s, L_s is depth, width and length of scour hole respectively and R_c is hydraulic radios and t is time of scouring per minuet.

Numerical simulation will computed for t=300s from total equilibrium scour time (18000s).

Regarding to Equation (2), dimensional of scour's hole (length, depth, width) is calculated with chronicle distance of 30 second (Table 1).

t(s)	hs(m)	Ws(m)	Ls(m)
0	0	0	0
30	0.06571	0.2753	0.24931
60	0.0685	0.291	0.2672
90	0.07019	0.30059	0.27826
120	0.07141	0.30759	0.28638
150	0.07237	0.31313	0.29284
180	0.07317	0.31773	0.29823
210	0.07385	0.32168	0.30286
240	0.07444	0.32513	0.30693
270	0.07497	0.32821	0.31057
300	0.07544	0.33099	0.31386

Table1.dimension of hole scour calculated by Eq.(3)

3 NUMERICAL SIMULATION

3.1 Governing equations

For simulation, FLOW3D-V9.0 software has been applied which is a pack for computation fluid dynamics. Method for solution governing equation is finite volume .as we know in turbulent flow we use time average equation for considered turbulence's effects. Therefore, continuity and momentum equation (Reynolds equation) is applied.

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{4}$$

$$u_j \frac{\partial u_i}{\partial x_i} = -\frac{1}{\rho}\frac{\partial}{\partial x_i}\left(P\delta ij + \rho\overline{u_i u_j}\right) \tag{5}$$

Where ρ is the fluid density, u_i are the components of the local time-averaged flow velocity, P is dynamic pressure and $\overline{\rho u_i u_j}$ are the turbulence stresses. In this paper, the turbulence stress is modeled with the aid of the Boussinesq relation:

$$-\rho\overline{u_i u_j} = \rho v_t + \left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}k\delta ij \tag{6}$$

The standard k-ε turbulencec model is adopted, the eddy viscosity ν_t being given by :

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{7}$$

Where k is the turbulent kinetic energy and ε is the dissipation rate of k. the governing equations for k and ε can be expressed:

$$\frac{\partial k}{\partial t} + \overline{U}_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[(\nu + \frac{\nu_t}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + P - \varepsilon \tag{8}$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{U}_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[(\nu + \frac{\nu_t}{\sigma_k}) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P - C_{2\varepsilon} \frac{\varepsilon^2}{k} \tag{9}$$

Where P that is the production of k, is defined by:

$$P = \nu_t g = \nu_t \left[\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right] \frac{\partial \overline{u_i}}{\partial x_j} \tag{10}$$

The following values of the model coefficients are used Cμ=0.09, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, σ_k=1,σ=1.26.

3.2 Computational Filed

Two-meshed block is used for computational domain in which 14 cm in floor for sediment and 6 cm for height of tailwater is considered.
Block 2 is defined for the case of increase in accuracy of simulation of free falling jet .Cell size in block 2 is much smaller than cell size in block 1 specification of each block is presented in Table2.

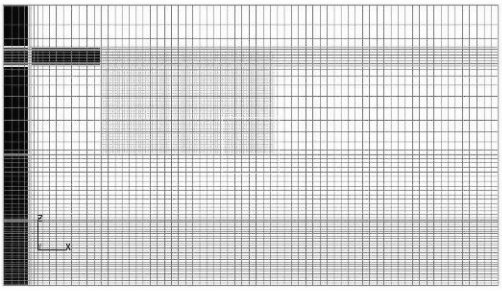


Figure 2.two meshed block in computational filed
xz direction

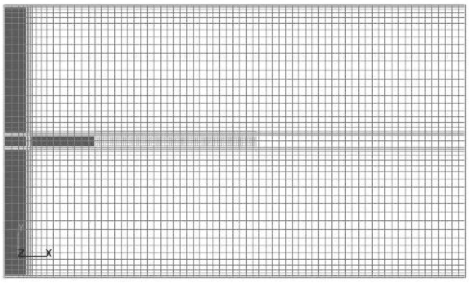


Figure 3. Two meshed block in xy direction

	x	y	z	Nx	Ny	Nz	Nt
B1	136	60	60	97	40	60	232800
B2	37	5	22	74	10	62	45880

Table 2. Specification of each block used in computational domain

	BLOCK 1	BLOCK 2
X	1.4	1
Y	3	1
Z	3.5	1

Table 3 . maximum adjusted cell size

DIRECTION	Block 1	Block 2
X-Y	4.42	1
Y-Z	8.5	1
Z-X	8.5	1

Table 4. maximum aspect ratio

In Table 2 x, y, z is dimension of computational domain, Nx, Ny, Nz is number of cells in each direction, Nt is total number of cells in each block.

The sediment scour model uses two concentration fields: the suspended sediment and packed sediment. the suspended sediment advents and drifts with the fluid due to the influence of the local pressure gradient. suspended sediment origination from inflow boundaries or from erosion of packed sediment the packed sediment, which does not advent, represents sediment that is bound by neighboring sediment particles. Its value divided by the critical packing concentration is the volume fraction of the cell that is occupied by packed sediment. Packed sediment can only move if it becomes eroded into suspended sediment at the packed sediment -fluid interface. Suspended sediment can become packed sediment if the fluid conditions are such that the sediment drifts towards the packed bed more quickly than it is eroded away.

The sediment concentration are stored as units of mass/volume (e.g.g/cm^3) .the solid volume fraction, fs, is a measure of the fraction of the total volume that is occupied by the sediment and the liquid fraction ,fl, is a measure of the fraction of the total volume occupied by liquid therefore:

$$f_L+f_s=1 \tag{11}$$

The suspended sediment enhances the mean fluid viscosity. This enhancement rises with sediment concentration until the solid volume fraction reaches the cohesive solid fraction $f_{s,co}$. Further rises in sediment concentration beyond solid volume fraction $f_{s,co}$ do not cause the viscosity to rise, rather the particles begin to interact with one another to cause solid-like behavior. In this situation, the average viscosity of fluid obtains by:

$$\mu^* = \mu_f \left[1 - \frac{\min(f_s, f_{s,co})}{f_{s,CR}}\right]^{-1.55} \tag{12}$$

Where μ_f is the molecular viscosity of the liquid , μ^* is the average viscosity of the mixture and $f_{S,CR}$ is the critical solid fraction.

The macroscopic density $\bar{\rho}$ is assumed a linear function of the sediment volume fraction

$$\bar{\rho} = \rho_L + f_s(\rho_s - \rho_L) \tag{13}$$

Where ρ_s and ρ_L are the microscopic densities of the sediment particles and the liquid, respectively.

The settling component of the model presumes the sediment particle to be spherical and their velocity to be small (so that viscous effects predominate in the flow around each sediment particle) the settling (drift) coefficient is automatically calculated as:

$$D_f = \frac{{d_{50}}^2 \times (\rho_s - \rho_L)}{18\mu} \tag{14}$$

therefore the settling velocity is:

$$u_{drift} = D_f \times f_L \frac{\nabla P}{\overline{\rho}} = \frac{f_L \times {d_{50}}^2}{18\mu} \frac{\nabla P}{\overline{\rho}} (\rho_s - \rho_L) \tag{15}$$

Where $\nabla P / \overline{\rho}$ is the mechanical potential gradient or body acceleration. $\nabla P / \overline{\rho}$ is limited to 10 time the magnitude of gravity to eliminate the effect of possible numerical oscillation in pressure. In the vicinity of the liquid free surface $\nabla P / \overline{\rho}$ is replaced by g. f_L is included in Equation (15) because drifting is limited by the presence of solid; in regions that are full as solid (f_L=0), u_{drift} falls to zero.

At the surface of the packed bed of sediment, the fluid shear stress acts to remove sediment, the amount eroded from the surface is a function of the fluid shear stress, the critical shear stress and the fluid and solid densities. A parameter familiar to hydraulic engineers is the critical shields parameter. This parameter correlates the minimum shear stress required to lift a sediment particle away from the packed bed and fluid interface:

$$\theta_{crit} = \frac{\tau_{crit}}{g(\rho_L - \rho_s)d} \tag{16}$$

Where τ_{crit} is the minimum shear stress along the bed surface. The goal of the model being developed is to predict the local flux of sediment being eroded everywhere on a packed bed interface a characteristic measure of velocity in the boundary layer is the shear velocity ($\sqrt{\tau / \rho}$) The resulting formula for scour lift is:

$$U_{lift} = \alpha n_s \sqrt{\frac{\tau - \tau_{crit}}{\overline{\rho}}} \tag{17}$$

Where n_s is the vector normal to the packed bed surface and α is a dimensionless parameter that represents the probability that the particle is lifted away from the packed surface its value is typically 1 or smaller.

The angle of repose controls how steep a slope can be supported by the packed sediment in a quiescent flow region. A high angle of repose allows for a steep (e.g. clay), while sediment with a low angle of repose slides downward much more easily (e.g. sand). In the FLOW3D sediment scour model, the angle of repose ζ is a parameter entered in degrees; the actual angle φ is computed by Equation (18):

$$\varphi = \frac{n_{\mathrm{int}erface} \cdot g}{|g|} \tag{18}$$

Where $n_{\mathrm{interface}}$ is the surface normal vector and g is the gravity vector .the model works by altering the critical shear stress τ_{crit} at sloping interfaces. Where φ is zero the effective critical shear stress τ_{crit}^{0} on sloping interfaces the critical shear stress is:

$$\tau_{crit} = \tau_{crit}^{0} \sqrt{1 - \frac{Sin^2 \varphi}{Sin^2 \zeta}} \tag{19}$$

Notice that when the local interface slope φ is equal to the angle of repose ζ , τ_{crit} is zero; sediment erodes from the interface due to any shearing action .Additionally the model predicts negative values of τ_{crit} where φ is greater than ζ; in such regions sediment spontaneously erodes even in the absent of shearing action of the fluid.

The motion of the suspended sediment in the system is descried by the advection - diffusion equation with the addition of the effect of drifting and lifting of the sediment:

$$\left(\frac{\partial c_s}{\partial t} \right)_x + u.\nabla c_s = \Gamma \nabla^2 c_s - u_{lift}.\nabla c_s - u_{drift}.\nabla c_s \tag{20}$$

Here u is the local fluid velocity, and u_{lift} and u_{drift} are the local lifting and drifting velocities .Γ is a diffusion coefficient that in FLOW3D is the turbulence diffusion of coefficient equal 1.

4 RESULTS OF NUMERICAL SIMULATION

Conclusion by simulation in time of 300 second indicates that shape of scour hole completely adapts with hole created in laboratory, which shows success in numerical model in solution of problem. The figure 4 indicates the final bed profile and situation of scour hole. The figure 5 also shows the depth variation in 100,200,300 second after starting simulation. The figure 6 indicates the variation of hill in downstream of scour hole in time.

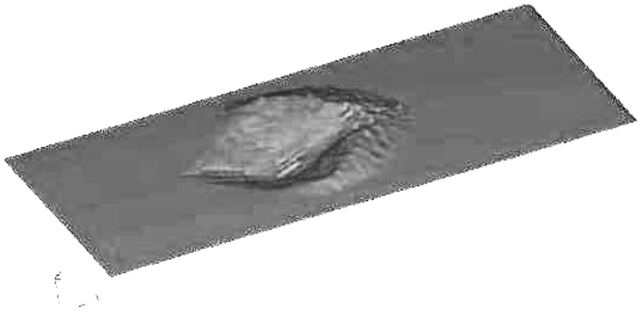
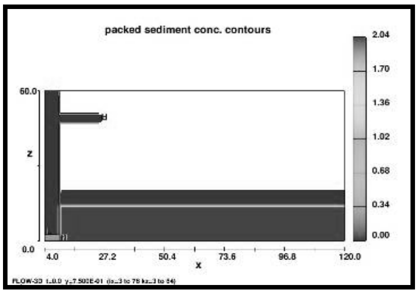
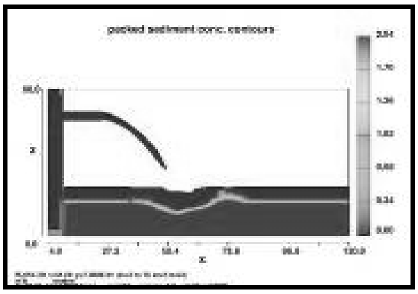


Figure 4. The hole scour t=300s

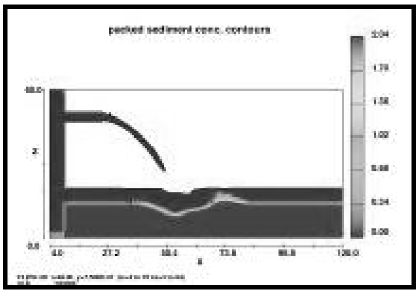
The computed maximum scour depth, width, length and experimental result that obtained by Eq.(3) is plotted versus time in figure 7,8and 9 respectively .



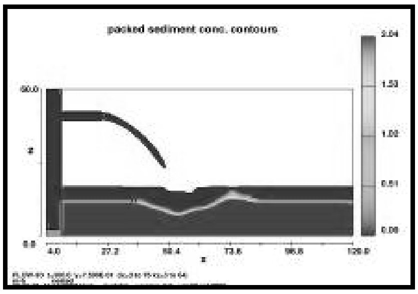
t= 0



t=100 s

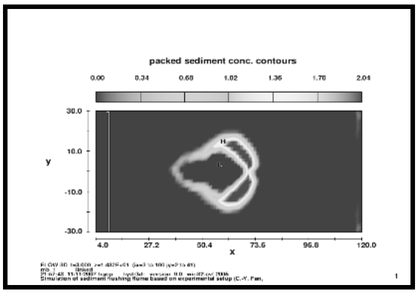


t=200 s

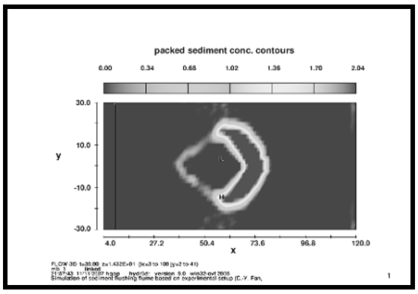


t=300 s

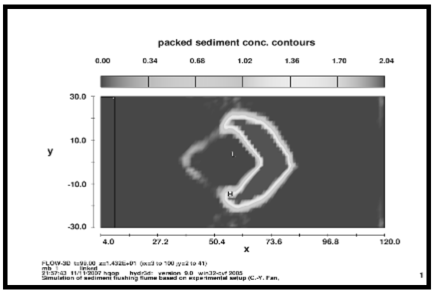
Figure 5. Variation of hole scour in time



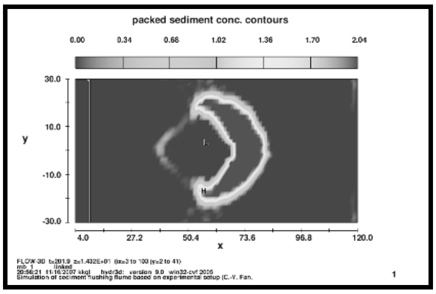
t=3 s



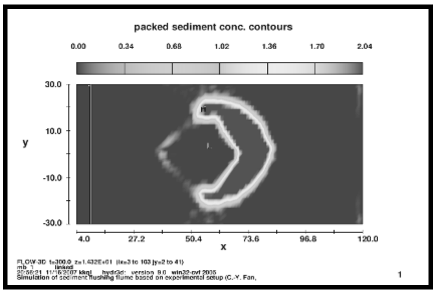
t=30 s



t=100 s



t=200 s



t=300 s

Figure 6. variation of hill in downstream in time

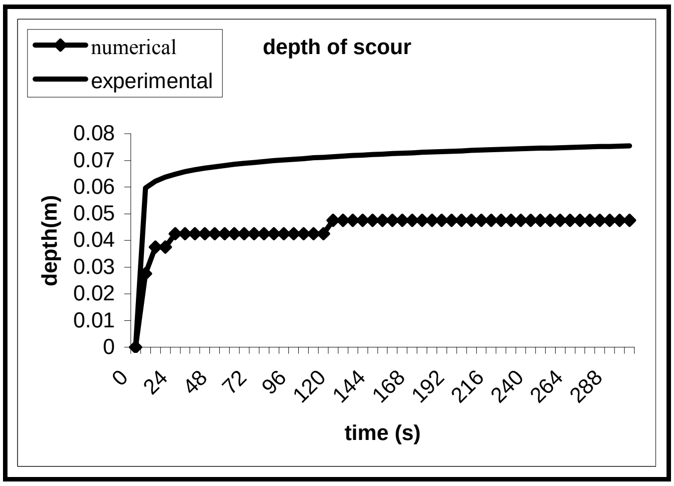


Figure7. Depth of scour hole in time

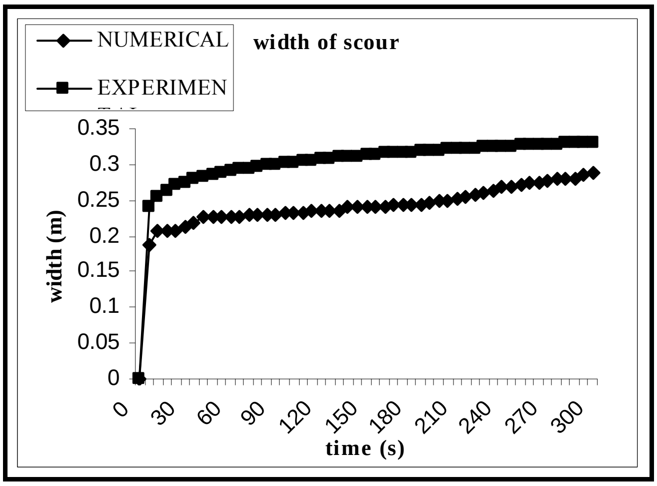


Figure 8. Width of scour hole in time

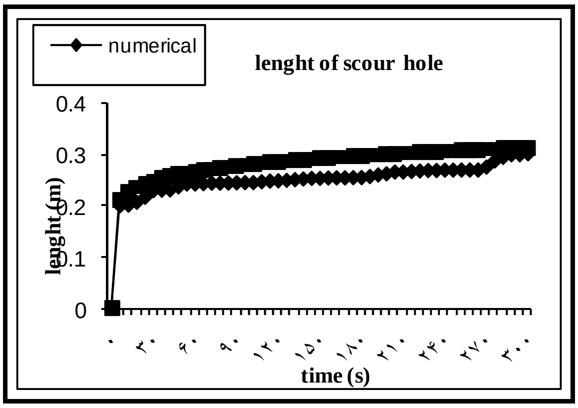


Figure 9. Length of scour hole in time

Table 4 indicate the final result of scour hole dimension in t=300s for numerical simulation, calculated with modified DOT equation (Eq.(3)) and actual experimental result .

	hs(cm)	Ws(cm)	Ls(cm)
Num	0.0475	0.28	0.30
EXP	0.05	0.29	0.31
DOT(Eq)	0.075	0.33	0.31

Table 4 .final result

Regards to results that presented for depth, length, width of scour hole numerical simulation calculated could offer acceptable conclusion for predicating of dimension scour hole.

5 CONCLUSION

Result shown in the firs 10 sound of scour process variation of dimension of scour hole is very fast and then variation continuity slower .In the case of increase sensitivity of model by use of blocks with more number of cell and smaller size, better results could be gained. Variation of dimension in time shows the scour process developed in vertical direction in the initial time and reach to constant value .although raises hole dimension in width and length continues.

6 REFFERENCES

[1] O.R.Stein and P.Y.Julien ,(1994),Sediment Concentration Below Free OverFall . journal of Hydraulic Engineering ,Vol .120 ,No.9 ,September ,1994

[2] A.A. Salehi Neyshabouri et al (2003) .Numerical Simulation of Scour by a Free Falling Jet. *Journal of Hydraulic Research* Vol 41 No.5, pp533-539

[3] Gijs J.C.M.Hoffmans (1998) .Jet Scour in Equilibrium Phase. *Journal of Hydraulic Engineering*, PP 430-437

[4] James Brethour .Modeling Sediment Scour .FLOW Science, Inc. 03-TN62

[5] Stefano Pagliara: Willi H.Hager , (2006). Hydraulic of plane plunge pool scour, *Journal of Hydraulic Engineering* ASCE , PP 450-461.

[6] U.S Department of Transportation, (2006). Hydraulic Design of Energy Dissipaters for Culverts channels, Chapter V, Hydraulic Engineering Circular No.14, Office of Engineering Federal Highway Administration, Washington, D.C., Third Edition.

[7] Blaisdell F.W and Anderson, C.L., (1991). pipe plunge pool Energy Dissipater, Journal of Hydraulic Engineering, ASCE, Vol .117 ,No .3. pp 303-323