Al-Azhar University Civil Engineering Research Magazine (CERM)
Vol. (43) No. (1) January 2021

3D-MODELING OF SCOUR AROUND BRIDGE SUPPORTS USING COMPUTATIONAL FLUID DYNAMICS

Sama M. Abdelalim[1*], Mahmoud Samy[2], Mohamed A. Gad[3],

1 *Teaching Assistant, Irrigation & Hydraulics Dept., Ain Shams University, Egypt
Email: sama.mohammad@eng.asu.edu.eg Tel: +201150872034

2&3 Professor, Irrigation & Hydraulics Dept., Ain Shams University, Egypt
Email:hydroshams@yahoo.ca

الملخص:

تاريخيا يعتبر النحر احد أهم الأسباب الرئيسية المؤدية لانهيار الكباري المقامة على الأنهار ومسارات الفيضانات. لأخذ هذا النحر في الحسبان عند تصميم الكباري فى الماضى تم استخدام المعادلات التجريبية لتقدير عمق هذا النحر لاعتباره في التصميم . في هذه الدراسة تم تطوير نموذج حسابي مبسط باستخدام ديناميكا الموائع الحسابية ثلاثية الأبعاد لتقدير قيمة عمق النحر بالقرب من دعامات الكبارى. النموذج تم تسميته SCDFS (ديناميكا الموائع الحسابية المبسطة للنحر) وتم برمجة هذا النموذج باستخدام ++C ولغة التكويد الخاصة ببرنامج ANSYS workbench. الفكرة العامة للنموذج هو خفض مستوى قاع النهر بالقرب من منطقة اعاقة تدفق المياه إلى الحد الذي يجعل اجهادات القص على قاع النهر تقل بالقرب من هذه العوائق الى قيم مستهدفة مستقرة يتم تحديدها. هذه الطريقة تم مقارنتها وتحقيقها بواسطة بيانات تجريبية للنحر. أثبتت النتائج ان هذا النموذج المطور يتسطيع محاكاة النحر بالقرب من دعامات الكباري. وتعتبر هذه الطريقة مشجعة للتطبيق خاصة في حالات الكباري ذات القواعد المركبة حيث لا توجد اي معادلات تجريبية متاحة يمكن الوثوق في نتائجها.

ABSTRACT

Historically, scour under bridge supports is considered one of the main reasons causing bridge failures across flooding routes and rivers. To account for scour in bridge design, empirical and experimental formulas have always been used worldwide to estimate anticipated scour depths in order to be taken into the design. The empirical approach has been preferred on the engineering level due to the difficulty in simulating the 3D hydrodynamic and sediment transport in the vicinity of bridge supports in a numerical and deterministic approach. In this study a simplified numerical model using 3D computational fluid dynamics is developed to estimate scour depths in the vicinity of bridge supports. The model is called SCDFS (Simplified Computational Fluid Dynamics Scour) and is written using both ANSYS workbench scripting and C++. The general idea of the model is to lower riverbed in the vicinity of the flow obstruction until the shear stresses in the vicinity of the obstruction is reduced to certain stable target values. The results show that the developed model can

simulate the scour in the vicinity of bridge supports. The developed approach is an attractive approach especially in cases of complex bridge foundations where no trusted empirical formulas are available.

KEYWORDS: Bridge; Scour; Numerical; CDF; Modeling; ANSYS; Fluent; Programming

1. INTRODUCTION

Scour is defined as the removal and excavation of bed and bank material of waterways due to the erosion effect of water flow. Scour holes formed around the supports of hydraulic structures have dangerous consequences. Scouring is considered the major cause of bridge failures [1]. Bed scouring causes riverbed to be lowered so that bridge foundations get exposed which can lead to structural failure. Statistics in the US show that 40% of failure cases of bridges is due to flood scouring while in New Zealand, this percentage reaches 62%. Similarly, scour is a main cause of bridge failures in different other countries [2].

Scour is classified into three types: long-term degradation, contraction scour and local scour. The three types contribute by different percentages in the final scour pattern around bridge foundations. Degradation occurs naturally over a long reach of stream. This type does not have a direct impact on hydraulic structures but the structural design must allow for long term degradation if it exists [3]. Contraction scour is the general decrease in bed elevations at the structure vicinity due to narrowing the stream cross section and the subsequent increase of average water velocity beneath the structure. Local scour occurs locally around structure foundations as a result of the significant increase in water velocities around foundations due to flow obstruction and the resulting vortices. Local scour is responsible for forming scour holes around bridge supports and it is the main scour component that can lead to bridge failure. The total scouring depth can be estimated by adding the three scour components.

Scour can be estimated using three main approaches: physical simulations, numerical modeling, and empirical formulation. Physical simulations work by estimating scour directly using a laboratory prototype of the structure and river under consideration. The physical simulations are not widely used on the engineering level due to the difficulty to precisely represent the real case, the typical scaling errors involved in the prototype similarity, in addition to its high cost. On the other side, the numerical modeling approach tries to calculate scour through numerical simulations of the underlying hydrodynamic processes. The numerical simulation approach is also not widely used on the engineering level due to the very complex nature of the underlying processes (hydrodynamic as well as sediment transport) in addition to the high sensitivity of the parameters which require significant effort for calibration. The empirical formulation approach tries to use the available trusted results made in the previous two approaches as well as the available historical field observations in order to relate scour to its predictors (i.e., the hydraulic parameters, geometric configurations, and sediment properties). Due to the simplicity of this approach (i.e., empirical formulas), it is widely used worldwide on the engineering level. However, in cases of complex geometric configurations (e.g., complex piers foundations) the third approach (i.e., the empirical approach) can lack precision due to the complex 3D flow pattern and the phenomenal vortices [3]. For example, HEC-18 [4] scour formulas are proved through small-scale models using Froude number similarities ignoring Reynolds number since Froude number highly affects free surface flow patterns. In complex hydrodynamic situations, turbulence parameters or Reynolds number cannot be neglected in the vicinity of bridge due to the significant contribution of turbulence in forming the scour hole [5]. It has been repeatedly reported in the literature that most of the empirical formulas show relatively an inaccurate representation of the scouring process and scour depth estimation when applied to complex conditions of water flow [6]. Hence, in complex hydrodynamic configurations, the first and/or second simulation approaches described above (the physical or numerical approaches) should be used to simulate scour.

Olsen & Melaaen (1993) investigated the use of the finite volume method to solve non-transient Navier Stokes equations aiming at expecting the scour hole around a cylindrical obstacle concluding that greater progression in computers capabilities will enable engineers to make better expectations for scour hole shapes and dimensions. Encouraged by this promise, J. E. Richardson & Panchang, (1998) constructed an approach to estimate the ultimate scour depth using repeated numerical simulations of flow in the vicinity of bridge foundations with different sizes of scour holes. However, this approach was limited to the cases where the shape of the scour hole can be known and expected. Recently, the significant advances in computing capabilities induced more profound investigations of the numerical approach capacity to depict the scouring process (a review of this recent progression of scour studies can be found in Thanh et al, 2014). However, to the best of the author knowledge, the numerical approach is still limited to simple configuration and little work using the numerical approach is available in cases of complex foundations.

This paper develops a numerical modelling approach using Computational Fluid Dynamics (CFD) that can be used for simulating complex situations in a trial to fill the above-mentioned gap. The modelling approach is developed using ANSYS Fluent.

2- PROBLEM FORMULATION

The exact deterministic numerical simulation of sediments transport in the vicinity of flow obstructions is a very complicated process especially if considered in a 3D sense. This is because the difficulty in accurately specifying the settling velocity and suspension/re-suspension velocities. This is in addition to the need to model the advection/dispersion process in a numerically stable space-time discretization. In other words, such simulations require a hydrodynamic simulation followed by sediment transport simulation in every time step. Hence, such solution can be impractical for engineering applications. However, a blended approach between hydrodynamics and a simplified sediment transport is looked upon in this study. The idea is to replace the sediment transport component with a simplified method that relates scour depths to shear stresses developed around obstructions. Hence, the study considers preforming accurate steady-state 3D hydrodynamics using CFD then uses the calculated shear stresses to modify (i.e., lower/raise) bed levels (i.e., form scour holes). Lowering of bed levels at high shear stresses regions leads to reducing shear stresses at those regions. The best solution should produce shear stresses as close as possible to certain stable target shear stresses (which will be described in detail in the following section). To achieve this simulation, a time-independent model was developed. The model is called SCFDS (Simplified Computational Fluid Scour).

3- THE TARGET SHEAR STRESSES

The core idea of our approach is to use shear stresses values calculated via CFD simulations to develop the local scour hole around flow obstructions. This is generally accomplished by the comparison between the shear stresses in the natural (before flow obstruction exists) and the obstructed cases (after obstruction). First, in order to simplify the description of our approach, it is important to define four main shear stress values:

τ_n — The natural shear stress (shear stress calculated using CFD for the natural un-obstructed flow).

τ_{cr-s} — The critical shear stress for scouring the bed material (at which bed particles begin to mobilize). This value is constant for a certain bed material.

τ_{cr-d} — The critical shear stress for deposition below which deposition is triggered. This value is constant for a certain bed material.

τ_O — The obstructed local scour shear stress (after obstruction exists). This value starts at initial high value called the initial obstructed shear stress τ_{InitO} (i.e., just before any scour occurs) and is subsequently modified by the flow and the hole formation in every step until it reaches a stable target value.

As mentioned in previous section, the best solution (i.e., bed bathymetry) is that one having shear stresses as close as possible to certain target stresses. Those target stresses are specified differently depending on different cases as follows:

- *Degraded natural condition*. In this condition, the natural bed is already undergoing long term scour (degradation) before obstruction exists. In other words, the CFD calculated natural bed shear (τ_n) is already exceeding the critical value for scouring (τ_{cr-s}). In such case, and since the focus is only to calculate local scour, we assume that the scour hole in the obstructed case progresses until the obstructed shear stress

(τ_O) reaches the natural value (τ_n). In other words, τ_o will start at its initial value (τ_{InitO}) just after the obstruction is made and decreases with the progression of the scour hole. The scour hole is assumed to be fully developed when $\tau_O = \tau_n$ or the positive difference between them ($\Delta\tau = \tau_O - \tau_n$) is minimum. . Accordingly, the target shear stress for scour in this case is $\boldsymbol{\tau_{Target-s} = \tau_n}$**.** While the target shear stresses for deposition is the critical shear stress for deposition $\boldsymbol{\tau_{Target-d} = \tau_{cr-d}}$.

- *A stable natural or accreted condition*. The natural bed here is stable or undergoing long term deposition. The natural stream power here is not sufficient to mobilize the bed ($\tau_n < \tau_{cr-s}$). If the initial value of the obstructed shear stress (i.e., after obstruction) is also less than the critical shear stress, then no local scour occurs ($\tau_O < \tau_{cr}$). Otherwise, the scour hole progresses and is said to be fully developed once $\tau_O = \tau_{cr-s}$ or the positive difference between them ($\Delta\tau_O = \tau_O - \tau_{cr-s}$) is minimum. Accordingly, the target shear stress for scour in this case is $\boldsymbol{\tau_{Target-s} = \tau_{cr-s}}$**.** While the target shear stresses for deposition is the least of the natural shear stress and the critical shear for deposition $\boldsymbol{\tau_{Target-d} = min\ of}$ **(**$\boldsymbol{\tau_n}$ & $\boldsymbol{\tau_{cr-d}}$**).**
- The range between the target shear stress for scour and target shear stress for deposition is called the stable range at which no local bed change occur.

4- SCFDS DESCRIPTION

<u>4-1 THE HYDRODYNAMIC COMPONENT</u>

3D fluent-based systems within ANSYS workbench is used to model hydrodynamics. Fluent is used in the developed SCFDS to model the 3D hydrodynamics and develop bed shear stresses of the natural case τ_n (i.e., before obstruction), the initial obstructed case τ_{InitO} (i.e., just after obstruction), and for all subsequent bed modifications τ_O. Hence, Fluent is used repeatedly (called repeatedly from SCFDS steering module) to calculate bed shear stresses corresponding to any assumed scour $S_{i,j}$.

Fluent requires different inputs (geometry, materials, boundary condition, turbulence parameters, and solution controls). Fluent then automatically generates a computational mesh on which flow equations are solved to determine 3D hydrodynamics including shear stresses on any face of the domain. For full details on using ANSYS-Fluent, refer to (ANSYS Inc. (US), 2013). For theoretical background, flow equation, numerical schemes, refer to (ANSYS Inc. (US), 2013). Finally, the required shear stress distribution on the stream bed is displayed is exported to an un-structured ASCII encoded point file (representing the mesh nodes existing on the bed) with entities of: x, y ,z and τ.

It is important to note here that an irregular (unstructured) mesh is used in Fluent in order to allow details of flow obstruction without scarifying the speed of computations. Accordingly, locations of the ASCII bed points exported from Fluent differ along the simulation steps according the mesh generated in each step (e.g., natural, initial obstructed, or the modified obstructed solution steps). And since both scour calculations and the bathymetry entered to Fluent requires the use of structured grid, it is necessary to interpolate (refer to the interpolation component in next section) shear stresses calculated in each case as attribute to a fixed regular scatter point set on the bed (i.e., structured bed scatter) on which scour calculations are performed (refer to the scour component in section 3-3).

4-2 INTERPOLATION FROM UN-STRUCTURED TO STRUCTURED

In order to perform the spatial interpolation from Fluent un-structured bed points to a fixed location structured scatter set, the Inverse Distance Weighted (IDW) spatial interpolation technique is implemented. The idea of the technique it to use shear stresses values at the nearest FLUENT unstructured points to determine an interpolated value to any structured point. To illustrate the technique, Let τ_m denotes shear stress at unstructured point *m*. The interpolated shear stress at structured point *(i,j)* can be calculated from:

$$\tau_{i.j} = \frac{\sum_{m=1}^{M} \frac{\tau_m}{l_{m\to(i,j)}{}^2}}{\sum_{m=1}^{M} \frac{1}{l_{m\to(i,j)}{}^2}} \to \text{ Equation 1}$$

Where $l_{m\to(i,j)}$ is the distance between points *(i,j)* and *m*, and *M* is the number of nearest unstructured points (nearest to structured point a) that will be considered in interpolating a value at point a. A relatively small number of points *M* is used in the interpolation here (n = 4) to avoid unnecessary smoothing of the shear stress field. C++ is used to program this interpolation technique as a sub-module to be repeatedly called from SCFDS steering module.

4-3 THE BED CHANGE COMPONENT

This component can also be called the scour hole developer. It is also written using C++ and is repeatedly called from the steering module. The objective of this component is to calculate the bed change field $S_{i,j}$ (i.e., scour at each structured bed point) corresponding to a certain given shear stress field. For that reason, a stream power formula is used to relate the scour at any location to the shear stress at this location. The stream power formula is given by:

$$\Delta S_{(i.j)t} = C(\Delta\tau_{O_{(i,j)t}})^K \to \text{ Equation 2}$$

Where:

$$\Delta\tau_{O_{(i,j)t}} = \begin{cases} \left|\tau_{target-s_{i,j}} - \tau_{O_{(i,j)t}}\right| & \left(if \quad \tau_{O_{(i,j)t}} > \tau_{target-s_{i,j}}\right) \\ \left|\tau_{target-d_{i,j}} - \tau_{O_{(i,j)t}}\right| & \left(if \quad \tau_{O_{(i,j)t}} < \tau_{target-d_{i,j}}\right) \\ zero \ \left(esle, i.e., \tau_{target-d_{i,j}} < \tau_{O_{(i,j)t}} < \tau_{target-s_{i,j}}\right) \end{cases} \quad \to \quad \text{Equation 3}$$

Where:

C & K are the stream power coefficients

Note that *t* denotes trial number here and not time

Now, the bathymetric field (i.e., bed levels) to be used in the next hydrodynamic step can be calculated from:

$$BedL_{(i,j)t+1} = BedL_{(i,j)t} \pm \Delta S_{(i.j)t} \rightarrow \quad \text{Equation 4}$$

Where:

$BedL_{(i,j)t+1}$	The modified bathymetric surface for trial $t+1$
$BedL_{(i,j)t}$	The bathymetric surface in trail t
$\pm$	The minus is for scour and plus is for accretion

4-4 THE STEERING MODULE

SCFDS automates the whole process through a steering module written on ANSYS workbench scripting language (Refer to Figure 1). The steering module first runs Fluent to obtain the natural shear stresses field. The steering module then proceeds the solution trials starting from the initial obstructed case (that has zero scour) until it reaches to a converged (i.e., stable) solution. At every trial, the steering module performs the following:

1- Reads the bathymetry (i.e., bed levels) of the previous trial.
2- Generates a new mesh and runs Fluent to obtain the shear stresses corresponding to this bathymetry.
3- Calls the interpolation module to transform the shear stresses from Fluent un-structured bed points to a fixed structured grid.
4- Calculates the bed change field (Equation 2) then add the bed changes to the bathymetric levels to reach a new bathymetric surface to be used for next trial (Equation 3).
5- Evaluate the change in bed levels in order to stop iterations if there is no significant change in the bed levels field (i.e., a converged solution).

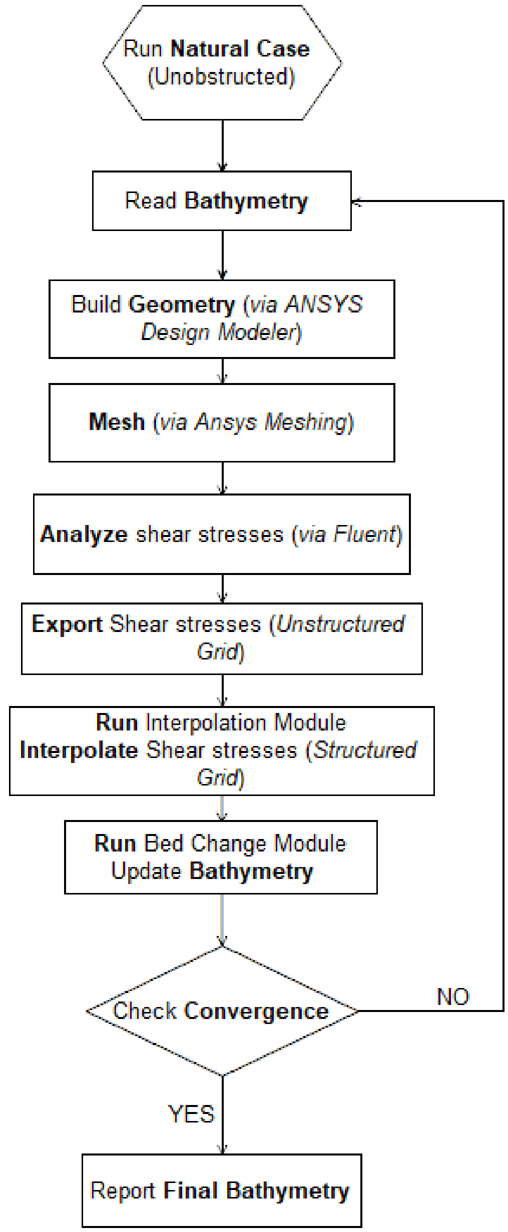

Figure 14 – Flowchart of SCFDS

Note that in each trial, there are different geometry and different 3D finite element mesh (since the bed levels are changed). In addition, and as described above, the steering module stops where there is no significant changes in the bathymetric levels between a trial and the previous trial (i.e., $\Delta S_{(i.j)t} \approx 0$). Note that this solution also satisfies a shear stress field as close as possible to the target shear stresses. In order to check for convergence, the model checks both the change in the maximum scour depth as well as the averaged root mean square change in the bed as following:

$$\Delta S_{(max)t} \leq err_{Smax} \quad \rightarrow \quad \text{Equation 5}$$

$$RMS_S = \frac{1}{n}\sqrt{\sum_{(i,j)=1}^{n} \Delta S_{(i.j)t}{}^2} \quad \leq \quad err_{RMS} \quad \rightarrow \quad \text{Equation 6}$$

Where n is the total number of scatter bed points and the allowed err values are predefined threshold values depending on the required accuracy. Low values of the err values requires more run time. It is up to the user to define the accuracy of the solution. For engineering practice and efficiency, it is recommended for both allowed error values to be 1-2% of their corresponding initial values at the starting trial (i.e., the initial obstructed case).

5- SCFDS VALIDATION

The same experimental data in [12] is tested here to validate the methodology. A discharge of $0.035\,m^3/s$ at velocity equals $0.3\,m/s$ flows in 1.2m wide flume developing water depth of 0.1 m obstructed by a 0.1 m diameter pier (refer to Figure 2). Bed is formed by layers of sand with d_{50} equals 1.3 mm and geometric standard deviation of grain size distribution of 1.27 and the maximum scour depth reported in the experiment is 0.04 m.

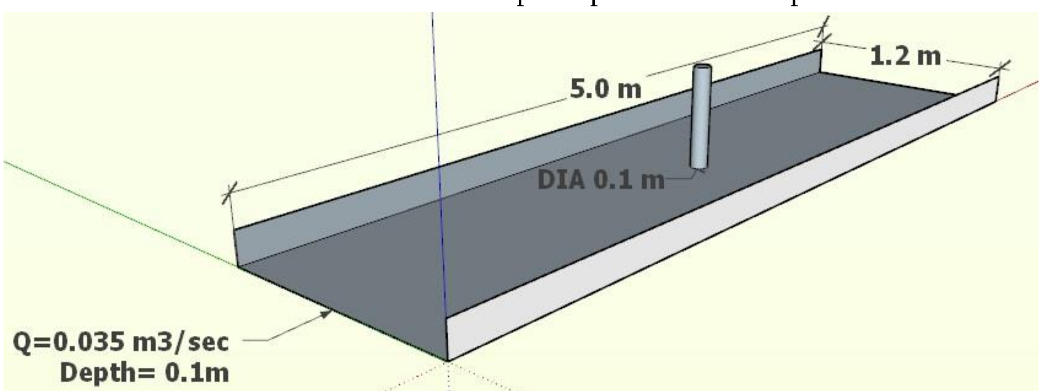

Figure 15 – Flume Dimensions

Geometries of the trials are built on ANSYS design modeler and then processed automatically via ANSYS meshing module creating different tetrahedron meshes. Sample screenshots are displayed in Figure (3). Single-phase (water) Fluent simulations before and after obstruction with boundary conditions described in Figure (3) are solved. A standard k-epsilon turbulence model is used in the simulations. Roughness height of bed is calculated in terms of d_{50} according to [13], $k_s = 0.002269\,m$.

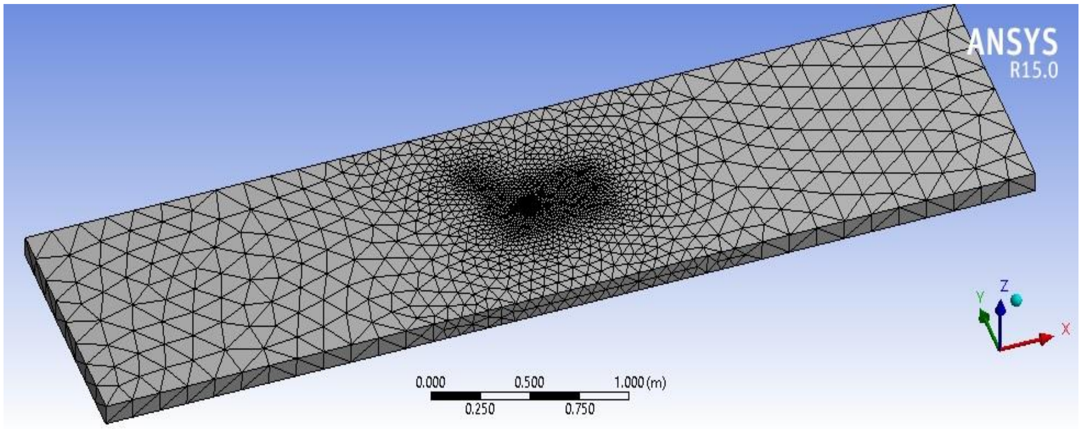

Figure 16(a) – A sample of meshing in 3D (at the 7th trial)

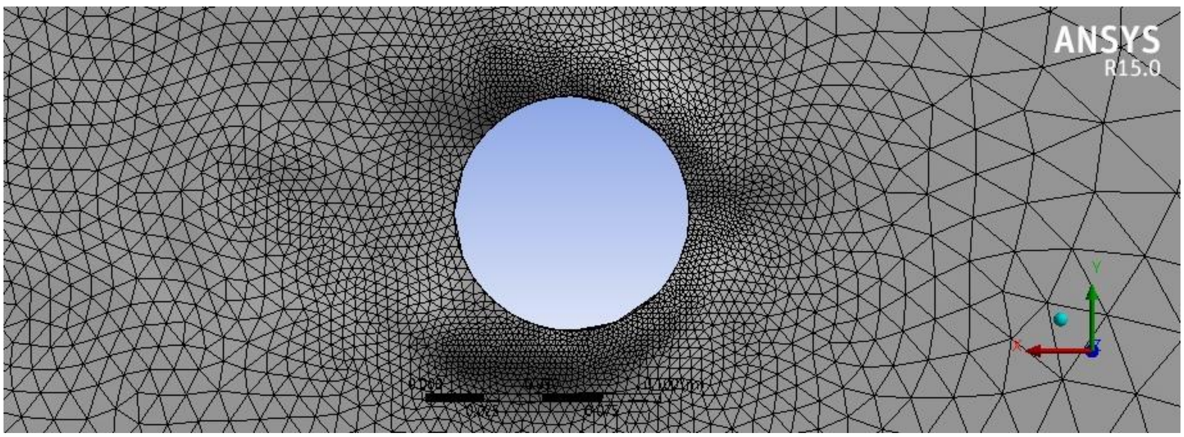

Figure 3(b) – A close zoom to the region of the pier at the sample mesh shown in Figure 3(a)

Bed	Outlet	Inlet	Side walls	Pier
No slip wall (k_s = 0.00269)	Pressure outlet	Velocity inlet	Slip walls	No slip wall

Figure 3(c) – Boundary conditions used in SCFDS

Shear stress contours at the stream bed before obstruction show an average value $\approx 0.25 * 10^{-1}\ pa$ indicating the natural shear stress, while in the initial obstructed case, the values significantly increase to reach a maximum value of $1.4\ pa$ around the pier, as appearing in Figure (4). The initial obstructed velocity distribution on a horizontal surface at mid-depth is also shown in Figure (5). The critical shear stress τ_{cr-s} is calculated according to [14] and it was found to be $0.708\ Nm^{-2}$ – which means *a stable natural condition* , so the targeted shear stress for scour equals $0.708\ pa$ while that target shear stress for deposition was taken as the natural shear stress for simplicity since the focus here is to look at scour only.

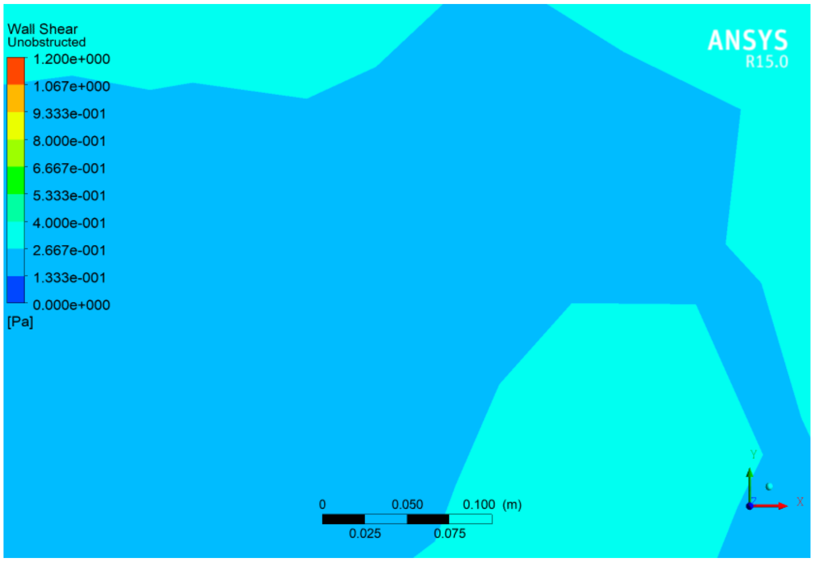

Figure 17(a) – Natural shear stress distribution (Unobstructed case)
As shown in the figure: The natural shear stress ranges from 0.2 to 0.3 pa

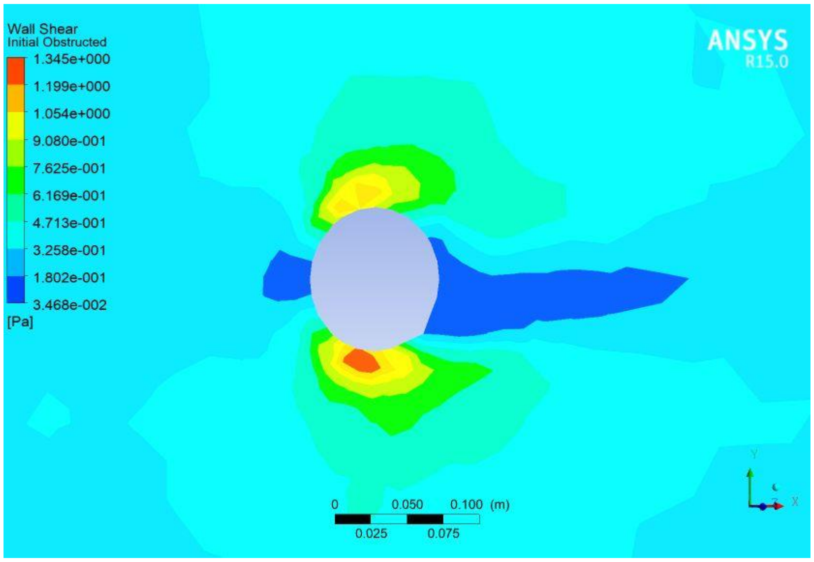

Figure 4(b) – Initial obstructed shear stress distribution
As shown in the figure: Shear stresses increase to a maximum value of 1.4 pa

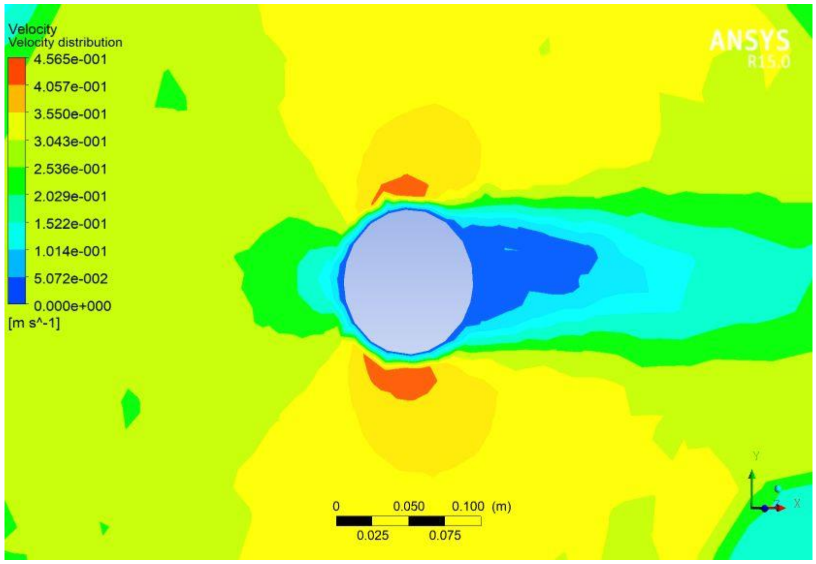

Figure18 – Velocity distribution on a horizontal surface taken at mid-depth

The model is run using C =0.0068 , K = 0.72, %err$_{Smax}$ = %err$_{RMS}$ = 2%. Figure (6) presents the spatial distribution of scour on the bed for the final solution (the calculated maximum scour depth here is 4.1 cm which agrees with the observed one which is 4.6 cm). Figure (7) shows the shear stresses corresponding to this final solution.

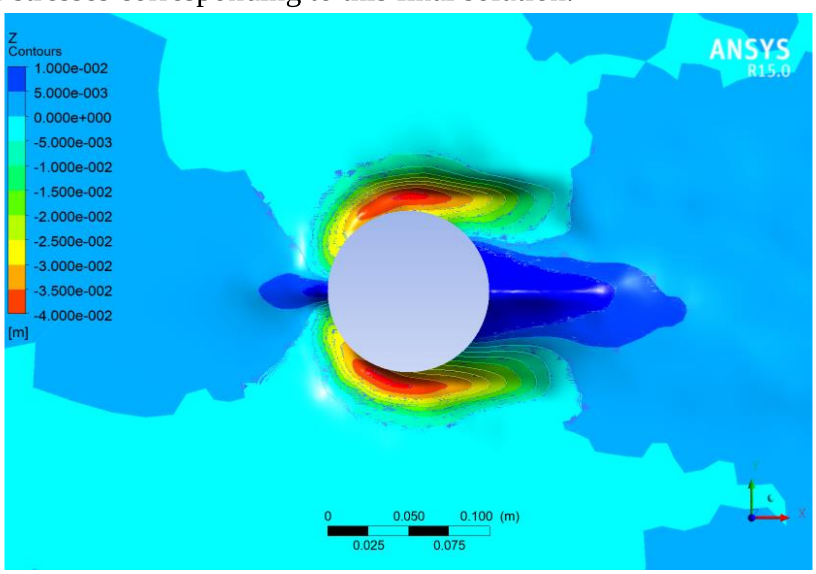

Figure 19 – Contours of scour distribution (Max. scour = 4.1 cm)

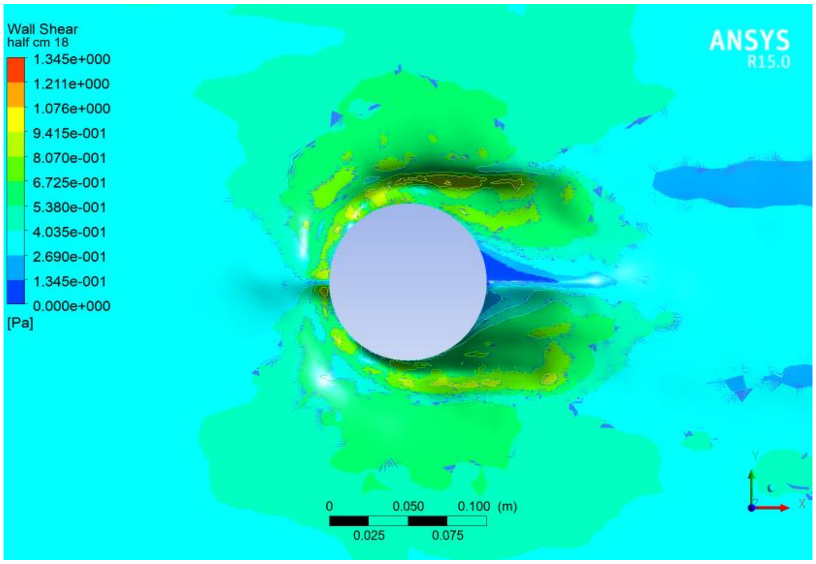

Figure20 – Shear stress distribution after scouring (Final Solution)
As shown on the figure the shear stresses dropped due to the formation of the scour hole and reached the stable range

6- SELECTION OF THE VALUES OF C & K

The K coefficient is used in the model to represent the non-linearity between the scouring process and the causing shear stresses changes. In other words, it represents how scour depths in the different places with lower values of shear stresses are related to the maximum scour depth. A value of 1 represents a linear relationship between scour and the causing changes of shear stresses. The K value basically affects the shape of the scour hole while its effect on the value of the maximum shear stress is less. A value of K in the range of 0.6-0.9 showed good performance. However, a future (coming) study shall inspect in details the sensitivity of this coefficient.

On the other side, the value of C actually can be thought to represent the accumulation period corresponding to a trial. Higher values for this coefficient can lead to numerical instability since it may cause the model to alternate between scour and deposition and eventually diverge. Very small C values, although ensuring stability, can cause unnecessary long number of trials and consequently long run-time to reach the solution. Hence, a compromise is to be made to find the optimum (i.e., a stable and yet efficient) value of C. Our experience with the model showed that a good value of C should corresponds to one tenth of the final expected maximum scour depth in the first iteration (i.e., the initial obstructed shear stresses). Accordingly:

$$C = (0.1 \rightarrow 0.3)\, \frac{S_{\max\_guess}}{(\Delta\tau_{Init0_{max}})^{K}} \qquad \rightarrow \qquad \text{Equation 7}$$

Where:

$$\Delta\tau_{Init0_{max}} = \tau_{Init0_{max}} - \tau_{target-s_{at\ maximm}}$$

$S_{\max\_guess}$ = A guess for the anticipated final maximum scour depth.

And since engineering practice over the past showed that the anticipated maximum scour depth is very roughly in the order of 0.5-0.8 the specific energy of the flow (as a rough guess), then this can be used also to guess a good value for $S_{\max\_guess}$. Substituting this guess in equation (7) yields:

$$C = (0.1 \rightarrow 0.3) \frac{E_s}{\left(\Delta\tau_{Inito_{max}}\right)^K} \quad \rightarrow \quad \text{Equation 8}$$

Where E_s is the natural specific energy of the flow (before obstruction exists). This value of C was found to ensure model stability with acceptable number of trials to reach the final solution.

7- CONCLUSIONS

A model called SCDFS (Simplified Computational Fluid Dynamics Scour) is developed in this study to be used for modeling local scour in the vicinity of bridge supports. The main idea of the model is to systematically modify bed levels until shear stresses reaches the target stable shear stresses. To achieve this, the model starts from the initial obstructed case (before scour occurs) and proceeds through a number of iterations of systematically changing the bed levels until shear stresses reaches the target stresses. The model performs 3D hydrodynamics and scour simulations in each iteration. This is because changing bed levels affects the 3D hydrodynamic field. The hydrodynamic simulation is performed in the model using ANSYS-Fluent while scour calculations is performed using two coefficients (C & K) stream power formula. The C coefficient affects the stability of the model as well as its run time to reach the solution while the K coefficient affects the internal curvatures of the scour hole. Both coefficients do not significantly affect the value of the maximum scour depth since it is mainly controlled by the target shear stress value. For local scour, the target shear stress for scour is taken in the model equal to the maximum of both the natural shear stress (before obstruction exists) and the critical shear stress for scour. In case the natural stress exceeds the critical stress (i.e. the target stress for scour is the natural) then the model shall produce very accurate estimation of the maximum scour. On the other side, if the critical shear stresses exceed the natural stresses (the target shear stress of scour is the critical), then the maximum scour mainly depends on the value of the specified critical shear stress which should be specified accurately with detailed care. Our initial assessment of the model indicates that the model is a good tool to evaluate scour around bridge supports especially for the complex configurations where no empirical formulation is available. Additional quantification of the sensitivity of the model parameters and applications for other cases that have detailed observations are highly recommended for future studies of the model.

8- REFERENCES

[1] E. V Richardson and S. R. Davis, “Evaluating scour at bridges (HEC-18),” *US Dep. Transp. Fed. Highw. Adm. Color. 4th edn. FHWA NHI*, p. 1, 2001.

[2] G. W. Annandale, *Scour technology: mechanics and engineering practice*. McGraw-Hill New York, 2006.

[3] L. Hamill, “Bridge hydraulics.” E & FN Spon, 1999.

[4] L. A. Arneson, L. W. Zevenbergen, P. F. Lagasse, and P. E. Clopper, “Evaluating scour at bridges,” 2012.

[5] Qiping Yang, “Numerical Investigations of Scale Effects on Local Scour Around a Bridge Pier,” THE FLORIDA STATE UNIVERSITY, 2005.

[6] M. N. Landers and D. S. Mueller, “Channel scour at bridges in the United States,” 1996.

[7] N. R. B. Olsen and M. C. Melaaen, “Three-dimensional calculation of scour around cylinders,” *J. Hydraul. Eng.*, vol. 119, no. 9, pp. 1048–1054, 1993.

[8] J. E. Richardson and V. G. Panchang, “Three-dimensional simulation of scour-inducing flow at bridge piers,” *J. Hydraul. Eng.*, vol. 124, no. 5, pp. 530–540, 1998.

[9] N. V. Thanh, D. H. Chung, and T. D. Nghien, “Prediction of the local scour at the bridge square pier using a 3D numerical model,” *Open J. Appl. Sci.*, vol. 4, no. 02, p. 34, 2014.

[10] ANSYS Inc. (US), “Ansys Users’ Guide,” *Aging*, vol. 7, no. 11. pp. 956–963, 2015.

[11] ANSYS Fluent Theory Guide, “ANSYS Fluent Theory Guide,” *ANSYS Inc., USA*, vol. 15317, no. November, pp. 724–746, 2013.

[12] S. F. Saghravani and A. Azhari, “Simulation of clear water local scour around a group of bridge piers using an Eulerian 3D , two-phase model,” vol. 12, no. 5, 2012.

[13] N.-S. Cheng, “Representative grain size and equivalent roughness height of a sediment bed,” *J. Hydraul. Eng.*, vol. 142, no. 1, p. 6015016, 2016.

[14] R. Soulsby, *Dynamics of marine sands: a manual for practical applications*. Thomas Telford, 1997.