Analysis of Bridge Pier Scour by Using Numerical Simulations with FLUENT

May Than Zaw

***Abstract*—**In this paper, the scour depth around the pier is compared between the numerical simulations and field observation results. The case study, Bo Myat Tun Bridge, is located on Yangon-Pathein highway crossing upon Ayeyarwaddy River near Kyeinbinse village, Nyaung Done Township. Firstly, the scour depths are observed by using field observation data. Using the Fluent and Gambit software, scour depths are estimated by simulating to flow field around bridge piers and scour patterns around the piers are developed. Finally, it is observed that a satisfactory agreement is achieved between the results of numerical simulation and the observation.

***Keywords*—** Scour depth, bridge pier, numerical simulation, field observation, scour pattern.

I. Introduction

Scour is the removal of sediment around or near structures located in flowing water. Scour also occurs at the coastal regions as a result of the passage of waves. It is the removal of sediment around or near structures located in flowing water. The most common cause of bridge failures is from floods scouring bed material from around bridge foundations. Scour is the engineering term for the erosion caused by water of the soil surrounding a bridge foundation (piers and abutments). Flowing water can excavate and carry away materials from the river bed and from around the piers and abutments of bridges, thus resulting in scour hole. Formation of scour may weaken bridge foundation and in extreme cases bridge failures occur. The construction of bridges in alluvial channels causes a contraction in the waterway at the bridge site and hence gives rise to significant scour at that site [1].

There are generally three types of scours that affect the performance and safety of bridges, namely, local scour, contraction scour and degradation scour. Local scour is the removal of sediment from around bridge piers or abutments. When water is flowing through the piers or abutments, it may evacuate sediment around the pier and creates scour holes. This removal of sediment is caused mainly due to the formation of both horse vortex and down flow in front of the piers.

Wake vortices are formed as the flow, which is separated by the pier, converges at the downstream of the pier. With the increase in scour depth, the horseshoe vortex strength reduces, which automatically leads to a reduction in the sediment transport rate from the base of the pier [2]. Improving the understanding of the local scour phenomena is therefore vital to the engineer responsible for the design of pier foundations. The knowledge of the maximum possible scour around a bridge pier is of paramount importance in safe and economic design of foundations of bridge piers. Prediction of local scour holes plays an important role in bridge foundation design. Underestimation of the scour depth results in too shallow design of a foundation, which may consequently become exposed to the flow endangering the safety of the bridge. Overestimation of the scour depth results in too deep design of a foundation, which is not economical design. Excessive local scour can progressively undermine the foundation of the structure. Because complete protection against scour is too expensive, generally, the maximum scour has to be predicted to minimize the risk of failure [3].

FLUENT is the world leading CFD code for a wide range of flow modeling applications. With its long-standing reputation of being user-friendly, FLUENT makes it easy for new users to come up to productive speed. Its unique capabilities in an unstructured, finite volume based solver are near ideal in parallel performance[4].

GAMBIT is used as a tool to generate or import geometry so that it can be used as a basis for simulations run in FLUENT[4].

II. Literature Review

A. CFD Numerical Simulation

Computational Fluid Dynamic (CFD) is the analysis of systems by means of computer- based simulation. The advent of high-speed and large memory computers has enables CFD to solve many flow problems (including those are compressible or incompressible, laminar or turbulent, chemically reacting and multiphase) in a reasonable time. Computational Fluid Dynamic (CFD) codes cover the numerical solution of the Navier-Stokes equations in three dimensional and offer several properties for their extension. The methods of discretization in CFD are Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and the Boundary Element Method (BEM).

There are many advantages to use CFD including low cost, quick solutions, scale up, comprehensive solutions and safe options. The standard equations of continuity, momentum, and energy conservation are solved on a non-staggered grid by a finite volume approach. The k- ε model provides closure for

May Than Zaw, Department of Civil Engineering, Yangon Technological University, Myanmar

turbulence. One advantage of this solver is continuity and momentum equation [5].

B. Governing Equations

The continuity and the momentum equations are as given below:

Continuity equation

$$\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}+\frac{\partial u}{\partial z}\right)=0 \tag{1}$$

Momentum equations

x-direction:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \tag{2}$$

y-direction:

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{3}$$

z-direction:

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{4}$$

C. Governing Equations of Modeling Multiphase Flows

Eulerian Model

The Eulerian model is the most complex of the multiphase models in FLUENT. It solves a set of momentum and continuity equations for each phase. Coupling is achieved through the pressure and inter-phase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than non-granular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modeled [6].

D. Volume Fractions

The description of multiphase flow as interpenetrating continua incorporates the concept of phase volume fractions, denoted here by α q. Volume fractions represent the space occupied by each phase, and the laws of conservation of mass and momentum are satisfied by each phase individually. The volume of phase q, Vq, is defined by

Conservation Equations

The continuity equation for phase q is

$$\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right)+\nabla\left(\alpha_q\rho_q V_q\right)=\sum_{p=1}^{m} m_{pq} \tag{5}$$

where, V_q is the velocity of phase *q* and m_{pq} characterizes the mass transfer from the p^{th} to q^{th} phase.

Conservation of Momentum

The momentum balance for phase q yields

$$\frac{\partial}{\partial t}(\partial_q\rho_q V_q)+\nabla(\partial_q\rho_q V_q V_q)=-\alpha_q\nabla p+\nabla\tau_q+\sum_{p=1}^{n}(R_{pq}+m_{pq}V_{pq})+\alpha_q\rho_q\,(F_q+F_{lift,q}+F_{vm,q}) \tag{6}$$

where, *q* is the q^{th} phase stress-strain tensor [7].

E. Turbulence Models

The laws of mass, momentum and energy conservation govern Laminar and turbulent flows. Modeling of turbulent flows requires appropriate modeling procedures to describe the effects of turbulent fluctuations velocity and scalar quantities on the basic conservation equations for laminar flow. FLUENT uses the standard Reynolds' averaging of the conservation and momentum equations.

In comparison to single-phase flows, the number of terms to be modeled in the momentum equations in multiphase flows is large, and this makes the modeling of turbulence in multiphase simulations extremely complex. FLUENT provides mixture turbulence model in multiphase flows within the context of the k- ε models.

The mixture turbulence model is default multiphase turbulence model. It represents the first extension of the single-phase k- $\boldsymbol{\varepsilon}$ model, and it is applicable when phases separate, for stratified (or nearly stratified) multiphase flows, and when the density ratio between phases is close to one. In these cases, using mixture properties and mixture velocities is sufficient to capture important features of the turbulent flow.

k- $\boldsymbol{\varepsilon}$ models

Two-equation *k- ε* models, turbulent kinetic energy *k* and turbulent dissipation *ε*, are the simplest and the most widely used models among all turbulence models that aim to study the effect of turbulence in the flow. Two-equation model signifies that it includes two extra transport equations to represent turbulence properties of the flow. There are three different models that are derived from *k- ε* model standard *k- ε* model, Realizable *k- ε* model and Renormalization Group model (RNG). Despite of having the two general equations, these turbulence models use the different ways to calculate the principle form of the eddy viscosity equation

The turbulence kinetic energy k and its rate of dissipation, ε are obtained from the following transport equations

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\mathcal{E}-Y_M+S_k \tag{7}$$

$$\frac{\partial}{\partial t}(\rho\mathcal{E})+\frac{\partial}{\partial x_i}(\rho\mathcal{E}u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\mathcal{E}}\frac{\varepsilon}{k}(G_k+C_{3\mathcal{E}}G_b)-C_{2\mathcal{E}}\frac{\varepsilon^2}{k}+S_{\mathcal{E}} \tag{8}$$

G_k = the generation of turbulence kinetic energy due to the mean velocity gradient

G_b = the generation of turbulence kinetic energy due to buoyancy

Y_M = the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate

$C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_{\mu} = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$ [8]

F. Physical Models in ANSYS Fluent

ANSYS Fluent provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can be performed. In ANSYS Fluent, a broad range of mathematical models for transport phenomena (like heat transfer and chemical reactions) is combined with the ability to model complex geometries.

Another very useful group of models in ANSYS Fluent is the set of free surface and multiphase flow models. These can be used for analysis of gas-liquid, gas-solid, liquid-solid, and gas-liquid-solid flows. For these types of problems, ANSYS Fluent provides the volume-of-fluid (VOF), mixture, and Eulerian models, as well as the discrete phase model (DPM).

For all flows, ANSYS Fluent solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. For flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent.

G.Procedure of FLUENT Simulation

The basic steps of FLUENT are shown below;

1) Create the model geometry and grid.
2) Start the appropriate solver for 2D or 3D modeling.
3) Import the grid.
4) Check the grid.
5) Select the solver formulation.
6) Choose the basic equations to be solved.
7) Specify material properties.
8) Specify the boundary condition.
9) Adjust the solution control parameters.
10) Initialize the flow field.
11) Calculate a solution.
12) Examine the result.
13) Save the results. [9]

H.Empirical Scour Equations for Local Scour

There are many predictive empirical equations for local scour depth in the literature. These equations differ significantly in their form and in the magnitude of their predictions. The recommended equation developed at Colorado State University in its Hydraulic Engineering Circular No.18 Report, HEC-18

$$d_{se} = 2.0K_1K_2K_3K_4\,b^{0.65}\,y^{0.35}Fr^{0.43} \tag{9}$$

where,

d_{se} = equilibrium scour depth,
y = flow depth upstream of structure,
K_1 = correction factor for pier nose shape,
K_2 = correction factor for angle of attack flow,
K_3 = correction factor for bed condition,
K_4 = correction factor for armoring by bed material size,
b = pier width,
Fr = $V_0/(gy_o)^{1/2}$, Froude Number,
V_0 = Mean velocity of flow directly upstream of the pier
g = acceleration of gravity ($9.8 m/s^2$) ($32.2 ft/s^2$) [9]

III. Study Area

The study area, Bo Myat Tun Bridge, is located on the Yangon-Pathein highway crossing upon the Ayeyarwaddy River near Kyeinbinse village, Nyaung DoneTownship, Ayeyarwaddy Region ,95° 33’ 26.208” E and 17° 1’ 55.776” N (in terms of projected UTM, East 771,000 to 776,000 and North 1,884,000 to 1,886,000). It was opened on 15 November 1999. The overall length of Bo Myat Tun bridge is 2604.8 m including main length 1879.2 m and the approach length is 725.6 m. The river reach length of model area is about 3.8 km upstream of the bridge and 0.9 km downstream, totally 4.7 km. In the delta of the Ayeyarwaddy, river can be seen as meanderings or braided. According to the bridge in the deltaic area, there is a complex network of various types of channels. Bo Myat Tun Bridge was already constructed across the channel and pier number 17 existing on the west bank was in danger since construction period. Between pier number 16 and 17, the channel is constricted to 96m (span width) and helical flow generated by the bend becomes stronger due to the constriction.

Fig.1 Study area map

IV. Field Observation

A. Observed Bed Level Data

The observed bed level data are taken from the Directorate of Water Resources and Improvement of River System (DWIR). Safe scour line that is the limitation line for skin friction effect for pile design is defined by the Ministry of Construction (M.O.C). The safety factor is usually taken as 2.0 for pile foundation design in M.O.C. It means that the estimated maximum scour depth is taken as about 15.8m by M.O.C.

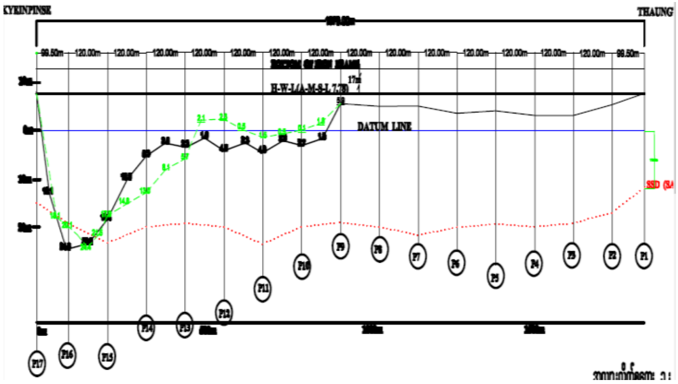

Fig.2 Cross sectional profile of design foundation of Bo Myat Tun Bridge

B. Scour around Piers of Bo Myat Tun Bridge

Scour around piers, P-9 to P-17 are shown in Table. Bo Myat Tun Bridge was already constructed across the channel and pier number 17 existing on the west bank was in danger since construction period. Between pier number 16 and 17, the channel is constricted to 96 m (span width) and helical flow generated by the bend becomes stronger due to the constriction. There are some years when catastrophic floods occurred and some of the structures are damaged due to the extreme events, weakness of the construction conditions and budget allowance. From 1996 to the present, adequacy and inadequacy of bank protection works are visible in reviewing the bed level changes.

TABLE I

SCOUR DEPTH AROUND A BRIDGE PIER

Date	P17	P16	P15	P14	P13	P12	P11	P10	P9
G.L(before construction)	1	-4.57	-4.9	-3	-4	-4.6	-4.9	-3	5
1998(December)	8	-13.4	-11.4	-8.4	-7.8	-5.6	-4.6	-2	5
1999(September)	8	-9.2	-8.4	-7.2	-6.8	-6.4	-5	-4	5
2000(February)	1.5	-15.2	-15.2	-11.2	-8.2	-8	-5.2	-4.2	-2.8
2001(July)	8	-16	-14	-11	-9.2	-6.5	-5	-4	-2
2002(November)	8	-22.4	-20.7	-16	-12.2	-0.5	-0.1	1	5
2003(October)	8	-21.5	-22.9	-14.2	-6.3	-0.8	-1.1	1.4	5
2003(November)	8	-23	-23	-19.8	-5.6	-5.6	-1	-1	-1
2004(January)	1.6	-16.75	-21.1	-13	-2	-0.5	-0.4	0	0
2004(September)	8	-24.2	-17.7	-11.3	-11	-8.75	-5.3	0.4	3.12
2005(August)	8	-16.31	-14.76	-11.8	-12.74	-12.2	-9	-3	5
2006(August)	8	-15.49	-15.15	-11.74	-12.25	-11.58	-10	-4	5
2008(January)	1.5	-13.4	-7.5	-10.7	-14	-14.9	-9.8	-3.8	0
2009(January)	1.5	-16.1	-11.6	-11.2	-12.4	-10.7	-11.9	-4.6	0
2009(September)	8	-16.2	-10.6	-10.6	-10.4	-8.5	-8.5	-4.5	5.9
2009(October)	8	-16	-12	-11	-9.3	-8	-8	-4.4	5.9
2010(January)	1.5	-14	-9	-8.6	-7.3	-6	-5.5	-4	-3.5
2010(September)	8	-23.2	-13.1	-11.13	-10.3	-9.1	-8.92	-5.44	5.9
2012(February)	8.2	-18	-12.7	-11.3	-11	-9.8	-8.5	-5.6	5.9
2013(January)	8.2	-18.1	-15.8	-11.5	-10.1	-9.6	-5.8	6	5.9
2013(December)	8.2	-17.1	-15.5	-13.7	-8.3	-5.3	0	2.2	5.9
2014(December)	8.2	-18.1	-17.9	-13	-5.7	2.3	-1.5	0.1	5.9
2015(November)	8.2	-24.6	-18.4	-5.3	-3.3	-4.2	-4.3	-2.7	5.9
2017(March)	8.2	-16.7	-16.9	-7.6	-6.5	-5.5	-2.5	-2.1	5.9

C. Bed level changes

The bed level from high water period in 2002, 2003, and 2004 have the changes reached to about 24 m at P-16. It can be seen that bed level changes in these year are very significant because most of the river training structures were destroyed by the catastrophic floods and unstable of channel conditions. The bed level changes reached about 16.3 m in 2005 and 2006 in high water period. These changes occurred because of the effect of renewed training structure in this waterway.

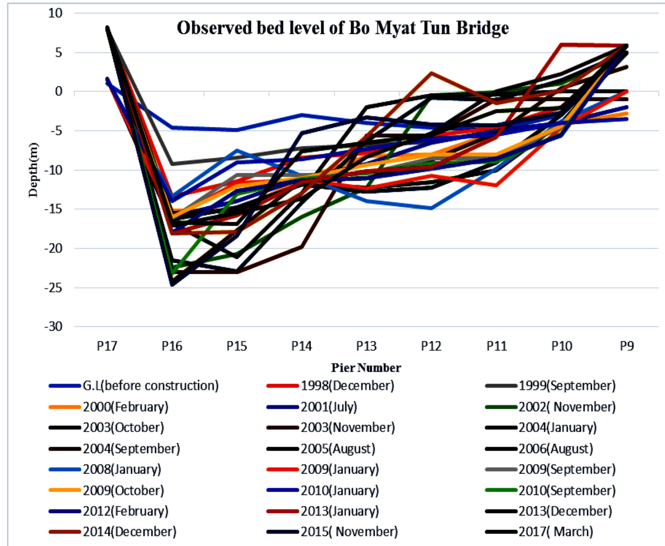

Fig.3 Bed level changes

V. NUMERICAL METHOD

A. Gambit and Mesh Generation

FLUENT is a two part program consisting of a pre-processor, GAMBIT, and a main module, FLUENT. GAMBIT was used to define the geometry and a structured grid of the problem to be modeled. The grid information was then imported from GAMBIT to FLUENT.

B. Geometric Setup

In the finite difference solution, the absolute size of cells within the grid is an important issue, since if the cells are too large, the solution obtained can be dependent on the cell size rather than purely on the physical constraints of the solution domain and the input conditions. For the present simulation, a two-dimensional grid system was generated by GAMBIT for the FLUENT simulation.

After several trial runs, the present grid arrangement was found satisfactory. The geometry of the flow zone for numerical simulation is generated by GAMBIT. The whole zone of the flow field is cuboids. The numbers of mesh are considered in 3 dimensions of length, width and height are 300x311x30 for model. A cylinder is placed at 150m from the center of the cuboids and its diameter is 5m for model. The grids of the simulation generated by GAMPIT are shown in Fig.2.

The grid close to the pier has much higher density than those away from it. Grid spacing increases gradually in the radial direction from the pier center. The grids near the cylinder are generated more densely because the flow in the region is more complex. The grid spacing use an interval size of 3 (the default).The mesh volume is 117230.

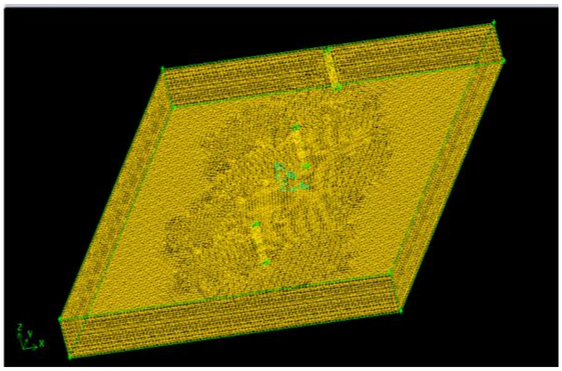

Fig.4 Mesh File by using GAMBIT

C. Boundary Condition

The diameter of cylindrical pier is 5m for numerical simulation and the material used for the bed are sand with specific weights of 2600 kg/m^3, mean particle sizes of 0.0028m.The inlet boundary is placed at 150m from the center of the pier. At the inlet boundaries, the type of boundary was set as velocity-inlet. In simulation, water velocity is 0.69 ms^{-1} and the inlet boundary is placed at 150m from the center of the pier. The type of boundary conditions is set as outflow. At the top surface, wall boundary conditions are used. The wall boundary is set at the bottom of sand. Pier is set as a slip wall boundary.

Two phases (water and sand) were set up. The top layer is set up water phase and bottom layer sand phase respectively. The two phase (water and sand) Eulerian model is used in order to simulate the local scour. For the Eulerian model, FLUENT provides Laminar and k-ε turbulence model. Here, k-ε turbulence model is employed to simulate.

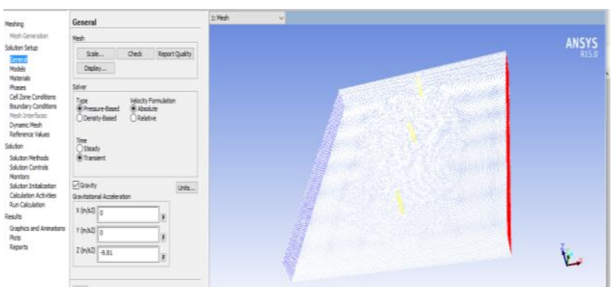

Fig.5 Using FLUENT to calculate pier scour

VI. RESULTS

A. Numerical Simulation Results

The numerical modeling procedure of the multiphase flow is made up of two steps. The first step is geometry preparing and mesh generating. In the second step the flow field is calculated, after defining the general and multiphase model, phases and their interactions, viscous and turbulence model, boundary conditions, accuracy of numerical discretization, and initialization of the flow field. After obtaining solution convergence, post-processing and analysis of results are made.

The numerical solution predicts velocities in the x, y and z directions. The velocity contour and the velocity vector around the cylindrical structure is analysed using ANSY Fluent. In this study, simulation time step is 1(seconds) and number of time steps is 7200. Maximum iteration per time step is 5. The variation of different flow parameters are presented in the Fig. 6 and 7.

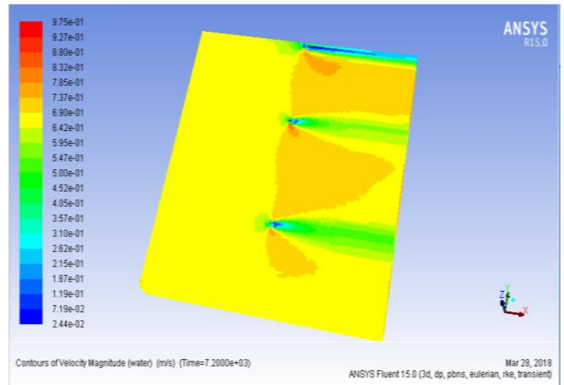

Fig.6 Contours of velocity magnitude by FLUENT

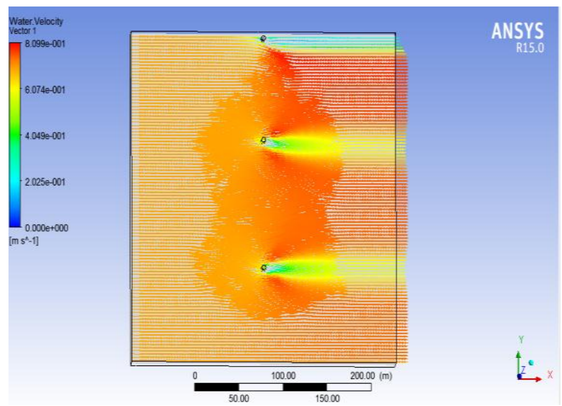

Fig.7 Contours of velocity vector by FLUENT

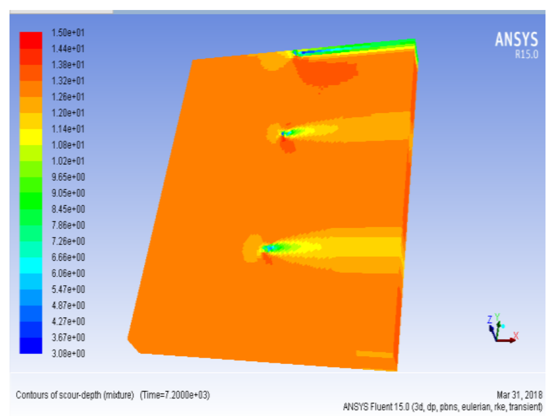

Fig.8 Contours of scour depth by FLUENT

B. Comparison of scour depth around bridge pier between numerical simulation and field observation in 2012

In 3-D simulation, change of velocity magnitudes and development of scour around pier were found reasonably. The comparison of scour depth between numerical simulation and observation are described in Fig. 9. The computed maximum scour depth for February 2012 is 15.6m at P-16 for Bo Myat Tun Bridge. The bed level for that is 18m. It can be compared with the estimated scour depth of Ministry of Construction that is 15.8m.

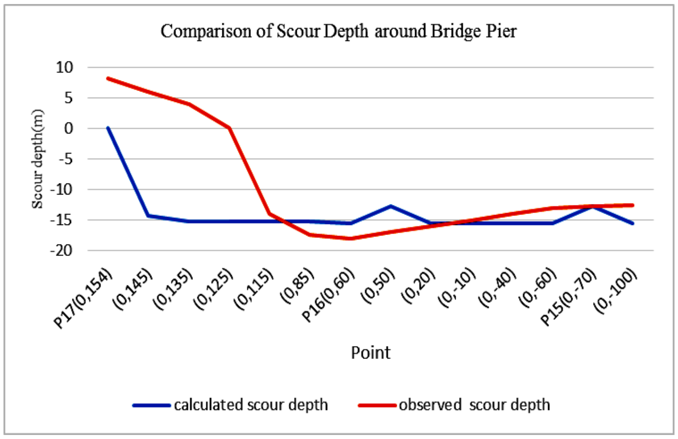

Fig.9 Comparison of scour depth around bridge pier

VII. DISCUSSION AND CONCLUSION

The numerical prediction scheme provided by CFD and experiment are used to predict the flow patterns of local scour around pier. The research has an important application in formulating numerical model of complex piers that include river bathymetry and bridge geometry so that scour can be predicted. Local scour is a direct consequence of the flow obstruction caused by the bridge. It is a time dependent process. The basic mechanism causing local scour at piers is the formation of horseshoe vortex.

During the Numerical solution, it was observed that the scour depths deeper and deeper at the midpoint of the upstream face of cylindrical pier. Simulated results show reasonable agreements with observation results. The comparisons between observation results and numerical results show that the mathematical model can simulate the process of local scour around pier and can obtain equilibrium profiles which are similar to observation results.

ACKNOWLEDGMENT

The author would like to express special thanks to all of my teachers at Yangon Technological University for their encouragement, true-line guidance, support, and suggestions.

The author is deeply grateful to my family and friends who kindly help throughout this study.

REFERENCES

[1] L.A. Arneson, L.W. Zevenbergen, P.F. Lagasse, P.E. Clopper, "Evaluating Scour at Bridges. Fifth Edition" U.S. Department of Transportation, Federal Highway Administration.

[2] Sabita Madhvi Singh & P. R. Maiti, “Flow Field and Scouring around Cylindrical Structure in Channel Bed”, Proc. of the Second Intl. Conf. on Advances In Civil, Structural and Environmental Engineering-ACSEE 2014.

[3] Padmini Khwairakpam, Dr. Asis Mazumdar. “Local Scour Around Hydraulic Structures”, International Journal of Recent Trends in Engineering, Vol. 1, No. 6, May 2009.

[4] Z. Zhang and B. Shi, “Numerical Simulation of Local Scour around Underwater Pipeline based on FLUENT Software”, Journal of Applied Fluid Mechanics, Vol. 9, No. 2, pp. 711-718, 2016. https://doi.org/10.18869/acadpub.jafm.68.225.22810

[5] QIPING YANG, “Numerical Investigations of Scale Effects on Local Scour around a Bridge Pier”, The Florida State University, FAMU-FSU College of engineering, October 28, 2005.

[6] eddy simulations and analyses for bridge scour development” E-proceedings of the 36th IAHR World Congress, The Hague, the Netherlands, 28 June – 3 July, 2015.

[7] B. D. ADHIKARY, P. MAJUMDAR, and M. KOSTIC, “CFD Simulation of Open Channel Flooding Flows and Scouring Around Bridge Structures”, Proceedings of the 6th WSEAS International Conference on FLUID MECHANICS (FLUIDS'09).

[8] Nazila KARDAN, Yousef HASSANZADEH, Habib HAKIMZADEH, “Comparison of Dynamic Bed Shear Stress Distribution around a Bridge Pier Using Various Turbulence Models”, ICSE6-283, August 27-31, 2012.

[9] ANSYS, Inc. Southpointe 275 Technology Drive,Canonsburg, PA 15317 “ANSYS Fluent Theory Guide”, November 2013.

[10] Vernon R. Bonner, Gary W. Brunner, “Bridge Hydraulic Analysis with HEC-RAS”, US Army Crops of Engineers, Institute for Water Resources, Hydrologic Engineering Center, April 1996.

About Author (s):

[The relationships among field and numerical studies can be investigated and existing bridges for scour vulnerability can be calculated.]