NATIONAL INSTITUTE OF TECHNOLOGY PATNA
(An Institute under Ministry of HRD, Govt. of India, New Delhi)
Ashok Rajpath, Patna – 800005

Dr. Ramakar Jha
Chair Professor & Head
Department of Civil Engineering

Email: rj@nitp.ac.in, rjha43@gmail.com
Tel.: +91 0612 269130 (Ext) 122(O)
Web site: www.nitp.ac.in

To,

Dear S.Behera, K.Rout, and Dr. Bandita Naik,

Subject: (a) Acceptance of Research Paper for ORAL Presentation
(b) Submission of Registration Fee for its publication in Proceedings

Ref.: Paper No. 445 (Loss coefficient for Contraction in Converging Channel)

It is my pleasure to inform you that your above referred research paper has been accepted for ORAL PRESENTATION in the very prestigious International Conference HYDRO-2018-INTERNATIONAL.

You are requested to kindly make online payment of Registration Fee for at least one Author and Co-authors attending the Conference and send the duly filled Registration Form by 2oth November 2018. **The research papers are likely to be published in the Proceedings (Springer) only after the receipt of registration fee from the author.** E-poster Template is attached too.

Looking forward to see you in HYDRO 2018 International at NIT Patna.

With kind regards,

Ramakar Jha

(PROF. (DR.) RAMAKAR JHA)

Chair, HYDRO-2018-INTERNATIONAL

hydro2018international@gmail.com

Loss coefficient for Contraction in Converging Channel

S.Behera[1], K.Rout[2], B.Naik[3]

[1]*M.Tech Student, Department of Civil Engineering, CAPGS, BPUT, Rourkela, India*
[2] *M.Tech Student, Department of Civil Engineering, CAPGS, BPUT,Rourkela, India*
[2]*Resource Person, Department of Civil Engineering, CAPGS,BPUT, Rourkela, India*
Email: beheraswagatika001@gmail.com
Telephone/Mobile No.: +919438139100

Abstract

In this present work, a hydraulic study was done on the contraction section of Kaskaskia River Reach-1, Illinois (USA), and Nani.G. Bhowmik. The data set of flow variables was collected from the Kaskaskia River-1. Investigation on loss coefficient were done on the contraction sections of Kaskaskia River of two 14 – 16 and 16- 17 and converging angle of 1.24° and 0.63° respectively. This study was done to understand the effect of flood on non-prismatic converging channels. The ANSYS Fluent software was used for four different turbulence model like *k-w, k-e, LES, RANS* on the contraction section of Kaskaskia River. Among all the models LES was given the best results. Predicted velocity was found nearly equal to the observed velocity.

Keywords: Turbulence model, ANSY-fluent, Contraction section Kaskaskia River Nani G. Bhowmilk, k-e, k-w, LES, RANS models

1. Introduction

Likely water is one the most fundamental and important resources available to the mankind. It archives the surface earth through methods for precipitation and after that is conveyed that around waterway channel to join the ocean. This phenomenon is more complex in non-prismatic compound channels with converging floodplains due to change in geometry. In converging compound channel the flow is forced to leave the flood plains and enter the main channel resulting in increased interactions and momentum exchange (Bousemer and Zech (1999), Bousemer et al. (2004), Proust et al. (2006), Rezaei (2006),Naik & Khatua(2014)). This extra momentum exchange should also be taken into account in the flow modelling. Today more than half of the world's population lives within 65km of a sea cost, and most of the major cities are also located on main river systems. So whenever flood occurs, this has lead to increase in the loss of life and economic cost (Knight and Shamseldin (2005).If the liquid particle appear to movie in definite smooth paths and the flow appears to be as a movement of thin layers on top of each other is known as a movement of thin layers on top of each other is known as laminar flow. The liquid particle move in irregular paths which are not fixed with respect to either time or space is known as turbulent flow. The river is the pillar of all progress inhabitants in ancient times, all the major progress and cities grows in the bank of the rivers. The river is the necessary part of human beings at the past and continued in the present because it provides the fertile land and sufficient water production. The river system can be divided as seasonal and year-continued flowing water .The importance was to understand the flow attributes of rivers in both the flow and over flow condition. Here the aim is to find out efficiency of LES (Large eddy simulation), RANS (Reynolds average Navier- Stoke equation), k-ϵ (k- epsilon), k-ω (k- omega) turbulence models to determine the flow conditions in the contraction section of Kaskaskia River, Illinois (USA) .

2. Study Area

The data set was collected from the Kaskaskia River, Illinois (USA) shown in Figure 1. It consists of two reaches namely Reach-1 and Reach-2.The total drainage area is 5801 square miles. The drainage area at reach-1 is 1330 square miles. We have worked on the Reach-1 River using its data sets. We found four diverging section namely (1-2 and 13-15) and angles (0.50° and 0.17°) respectively. After getting these data these were analyzed using ANSYS (FLUENT) and we get the velocity contours at inlet and outlet of the section. For the validation of the contours, actual contours were needed for which we used the software named SURFER for creating the actual contours which has been founded in the above named river. We have also used the four turbulence models such as *LES* (Large eddy simulation), *RANS* (Reynolds average Navier- stoke equation), *k-ω, k-ϵ* and finding which was better turbulence model among them.

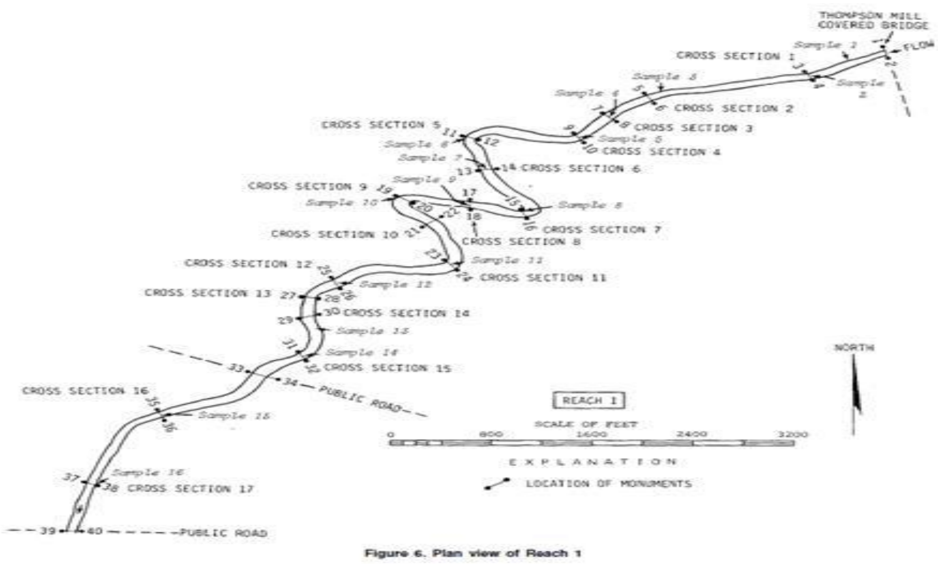

Figure1. Showing Kaskaskia River, Illinois (USA), Reach-1 (Nani G. Bhowmik)

3. Numerical modelling

A number of CFD packages (Fluent, CFX, and Star-CD, amongst others) were available and have been used for research in water flows. In recent past, a good number of researchers have used these software packages for prediction of different aspects of 3D flow fields e.g. Sahu et. al (2011). They detected that flow features in channels were dependent on topography of the channel, surface roughness etc. However, the flow behavior changes are still an unresolved phenomenon and attempts are underway to address this problem. These researchers attempted to predict the flow behavior using different numerical models as it is difficult to capture all flow features experimentally but still a lot of work is to be done. This is due to various problems which are encountered in numerical modelling such as grid generation, choice of turbulence model, discretization scheme, specifying the boundary and initial conditions etc. In this work, an attempt has been made to apply to diverging channel a

3D numerical code FLUENT has been used to test for its suitability for simulation of flood flows. The models tested here were *k-ε, LES, k-ω* and *RANS*, used for all simulation works.

3.1.Geometry

The first step in CFD analysis is the explanation and creation of computational geometry of the fluid flow region. A consistent frame of reference for coordinate axis was adopted for creation of geometry. Here in coordinate system, x-axis corresponded the lateral direction which indicates the width of channel bed. Y-axis aligned stream-wise direction of fluid flow and Z-axis represented the vertical component or aligned with depth of water in the channel. The origin was placed at the upstream boundary and coincided with the base of the centre line of the channel. The water flowed along the positive direction of the y-axis. The simulation was done on a non-prismatic compound channel with a converging flood plain. The setup of the compound channel is shown in Figure 2.

- Inlet
- Outlet
- Surface Geometry
- Channel Bottom
- Side Walls

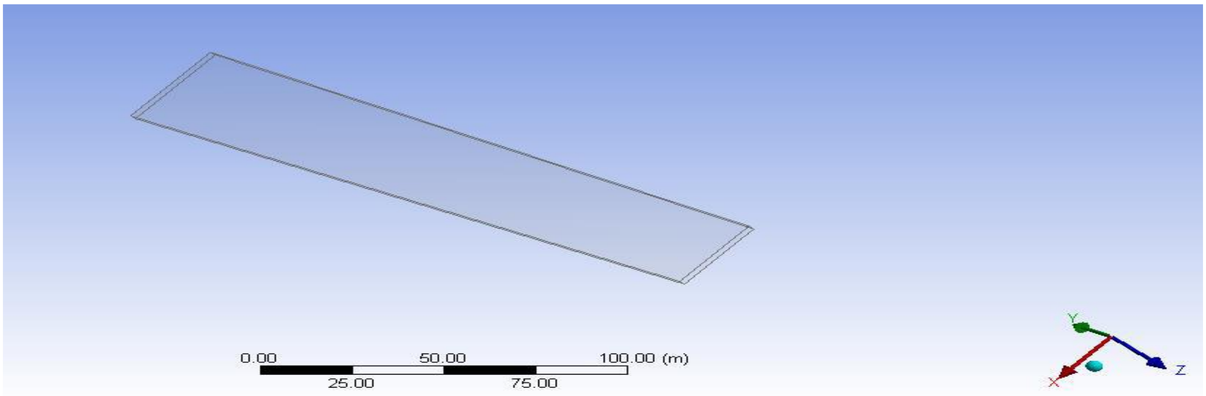

Figure. 2 Showing the geometry of converging channel

3.2 Mesh generation

The second and very important step in numerical analysis is setting up the discretized grid associated with the geometry. Construction of the mesh involves discretizing or subdividing the geometry into the cells or elements at which the variables will be computed numerically. Using the Cartesian co-ordinate system, the fluid flow governing equations i.e. momentum equation, continuity equation are solved based on the discretization of domain. The meshing divides the continuum into a finite number of nodes. The discretization of the computational domain is done through mesh generation, which can be identified later through control volume constructions. However, a very dense mesh of nodes causes excess computational time and memory. For CFD analysis, more nodes are required in some areas of interest, such as near wall and wake regions, in order to capture the large variation of fluid properties. Thus, the structure of grid lines causes further unnecessary use of computer storage due to further refinement of mesh. In this study, the flow domain is discretized using an unstructured grid and body-fitted coordinates. Unstructured grid is used so that intricacies can be covered under the grid which is left over in structured one.

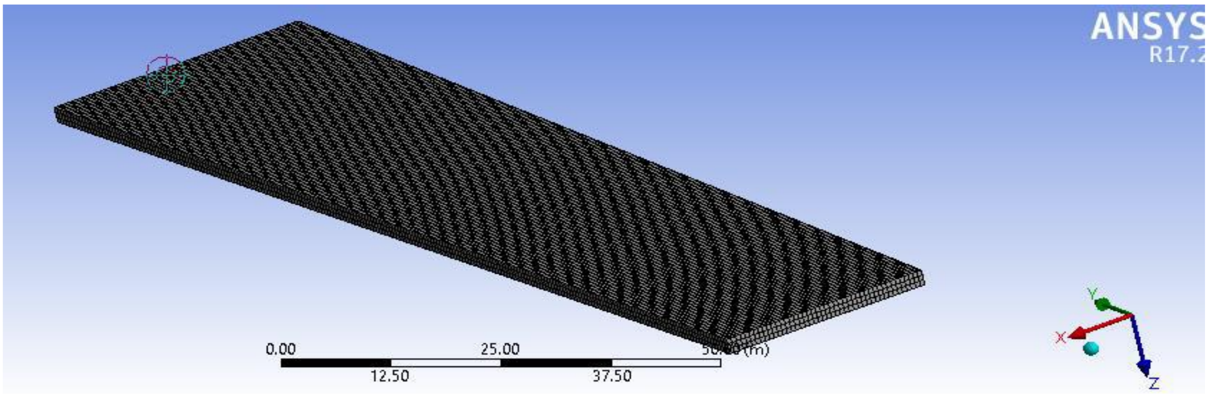

Figure. 3 Showing the mesh generation of simple converging channel

3.3 Solver Setting

3.3.1. Setup

After the meshing part was finished, various data were given in the Setup section. VOF (volume of fluid) model was the only design prepared for open channel flow simulation because this numerical analysis method concern with free surface flow. VOF was able to calculate time-dependent solutions. Flow in an open channel is mainly bound by channel from all directions especially for the rising upward free surface. To attain a free surface zero friction, an instructor called "surface symmetry" was given at the named selection in the computation. Velocity for inlet and pressure outlet for outlet is determined and the roughness coefficient was adjoining to the walls for "no slip" parameters. Temporary flow was chosen as the flow parameters were manifold in time in the experimental. Gravity was marked and the value for Z-axis was given as -9.81m/s² because gravity acts downward opposing to the z-direction vector. As mentioned earlier, the turbulence model was chosen as *k-ω, k-ϵ, RANS* (Reynolds Average Navier-Stoke equation) and *LES* (Large Eddy Simulation) models were used in our analysis. PISO was selected for solving the pressure equation as it was a pressure based algorithm used for short- lived flow conditions. It also allows large time step for precise calculation. Calculation was taken from inlet after the initial values of pressure and velocity were given and y-velocity value was given as water depth of the channel, then it is patch and close. Then it's time for calculation part in which the time step size was set to 0.001s and number of iteration given was 1000 for better accuracy and best results. After the calculation part is over the then we go to the result part where the velocity contours were found at the each section of the channels.

3.3.2. Governing equation

ANSYS Fluent uses the finite volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, viscous, laminar or turbulent etc. FLUENT is user-friendly and it is applied almost all branch of engineering science dealing with Numerical science. Initially, the closure problem of governing equations was considered as there is no universal closure model which is acceptable for all flow problems. Several models were studied to compare the effect of turbulent modelling in the converging compound channel, including the following: (1) *k-ϵ*, (2) *k-ω*, (3) Large Eddy Simulation (*LES*) model and (4) *RANS*.

(i) *k-ω* and *k-ϵ*

k-ω model is used for turbulence modelling. The *k-ω* model solves the k-transport equation and a transport equation for *ω*. The k-transport equation and the transport equation for *ω* can be written (Wilcox 1988) as:

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + P - \beta' k\omega \tag{1}$$

$$\frac{\partial \omega}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma_\omega}\frac{\partial k}{\partial x_i}\right) + \alpha\frac{\omega}{k}P - \beta\omega^2 \tag{2}$$

And the eddy viscosity is given by:

$$v_t = {}^{k}/_{\omega} \tag{3}$$

Where *k* is the turbulence energy, *ω* is the turbulence dissipation rate and *p* is the turbulence kinetic energy production term. The turbulence equation was suggested by Menter (1994) as:

$$P = min\ (P,\ 10\beta' k\omega) \tag{4}$$

The *k-ω* model involves five empirical constants *β', β, α,* α_k and α_ω. They have their universal constant values which have been derived on the basis of high quality data. Their values vary from one turbulence model to another. For any particular turbulence model, the values these constant remains same for all simulation purposes. For standard k-ω, their values are presented in Table 1.

Table 1 values of the constant in the k-ω model

β'	B	α	σ_k	σ_ω
0.09	0.075	5/9	2	2

(ii) Large eddy simulation (*LES*)

Large eddy simulation (*LES*) attempts to partially resolve turbulence. The fundamental idea was that the small scales of turbulence can be modelled by a sub grid model, while the larger scales are resolved by the governing equations. This equation (5) was no uses in the turbulence model since new unknown's correlation appear in the turbulent transport and dissipation terms. The governing equation was;

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial u_i} = -\frac{\partial}{\partial x_i}\overline{\left[u_i'\left(\frac{u_j u_j'}{2} + \frac{p}{p}\right)\right]} - \overline{u_i u_j}\frac{\partial u_i}{\partial x_j} - v\overline{\frac{\partial u_i'}{\partial x_j}\frac{\partial u_i'}{\partial x_j}} \tag{5}$$

(iii)*RANS* (Reynolds Average Navier-Stoke Equation)

The Reynolds stress models (RSM) were more complicated than the eddy viscosity model. They provide a more accurate representation of the turbulence and valid over a wide range of flows. *RANS* equation was the time averaged equations of motion for fluid flow and primarily used to describe turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximation time- averaged solutions to the Navier- Stoke equation. The governing equation was;

$$\frac{D\overline{u_i' u_j'}}{Dt} + \frac{\partial}{\partial x_k}T_{kij} = P_{ij} + R_{ij} - \epsilon_{ij} \tag{6}$$

4. Result

4.1. Comparison of Actual Velocity Contours and ANSYS Velocity Contours

The non-prismatic converging sections were collected from Kaskaskia River reach-1 and the velocity contours of each cross section are plotted using software called SURFER (15.0). Similarly, velocity contour are plotted using ANSYS Fluent, by taking four turbulence models. Four turbulence model were taken such as *k-ε, k-ω, RANS*, *LES*. The figure 4 show the actual velocity contour for section-1 and the figure 5 (a, b, c, d) show the ANSYS velocity contour for section-1 using four turbulence model such as *k-ε, k-ω, RANS*, *LES*. The figure 6 shows the actual velocity contour for section-2 and the figure 7(a, b, c, d) show the ANSYS velocity contour for section-2 using four turbulence models. The figure 8 shows the actual velocity contour for section-17 and the figure 9(a, b, c, d) show the ANSYS velocity contour for section-17 using four turbulence models.

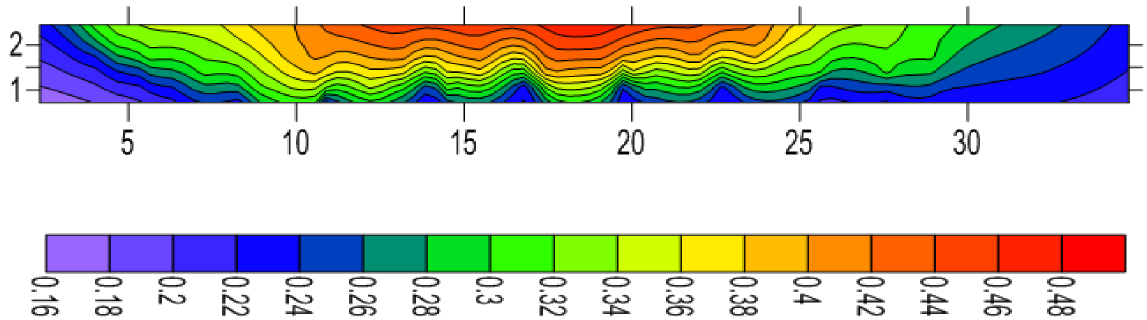

Figure 4. Showing the velocity contour section -14

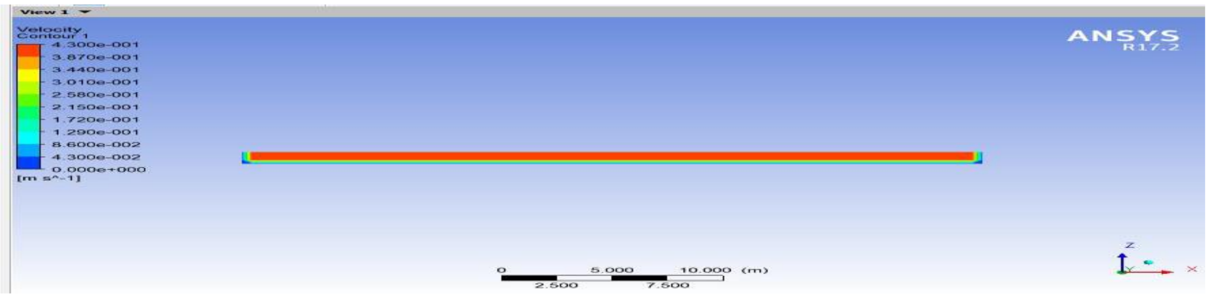

Figure 5(a) Showing the velocity contour for section-14 *k-ϵ* model for converging angle-1.24°

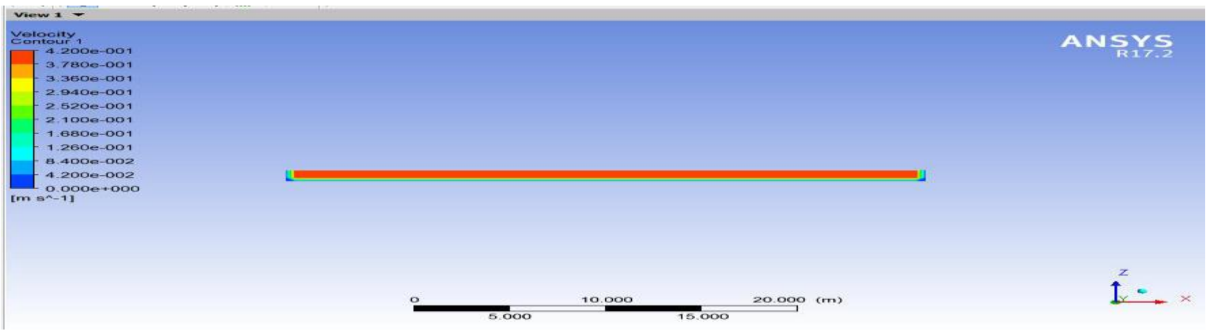

Figure 5(b) Showing the velocity contour for section-14 *k-ω* model for converging angle-1.24°

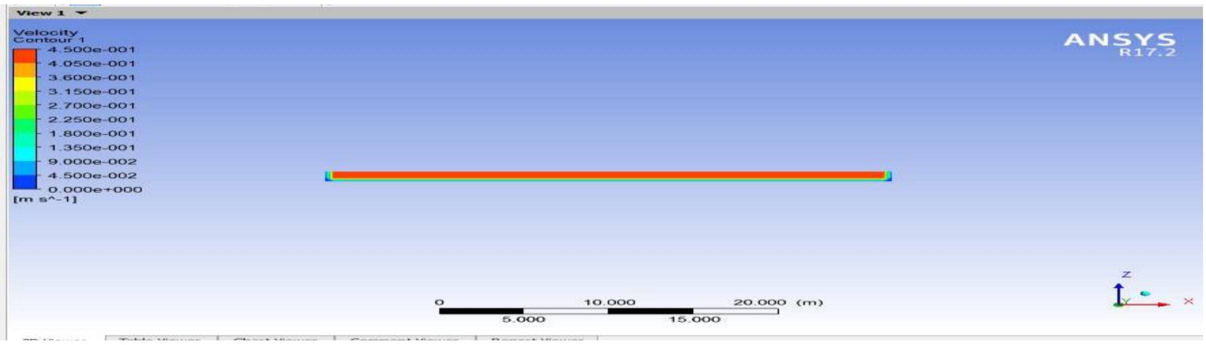

Figure 5(c). Showing the velocity contour for section-14 *RANS* model for converging angle-1.24°

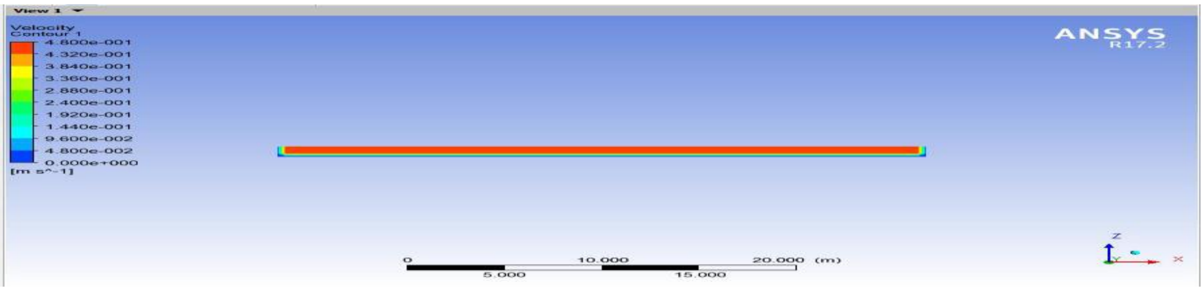

Figure 5(d). Showing the velocity contour for section-14 *LES* model for converging angle-1.24°

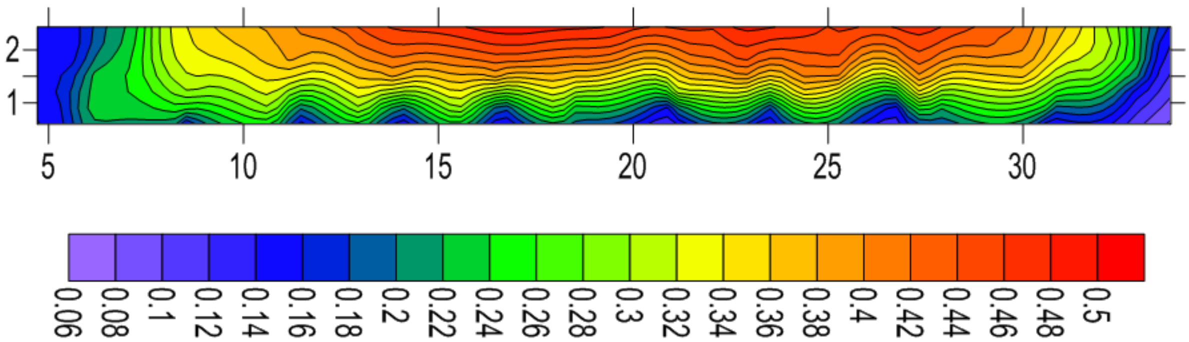

Figure 6. Showing the Velocity contour of section-16

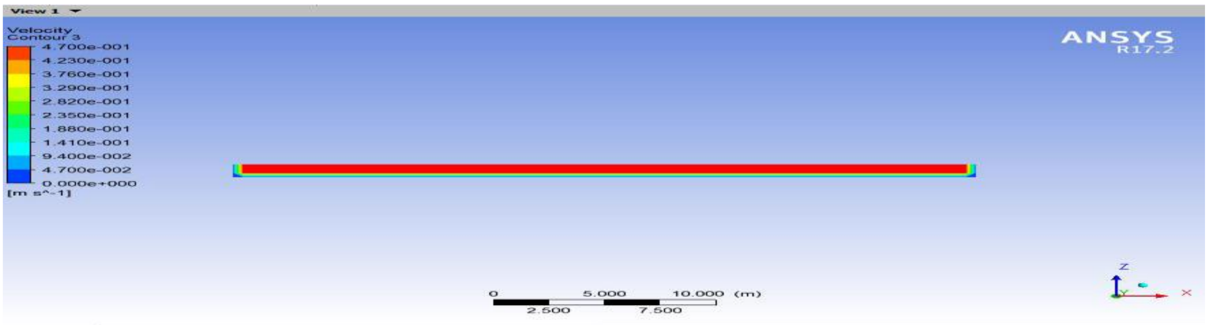

Figure 7(a) showing the velocity contour for section -16 *k-ϵ* model for converging angle-0.63°

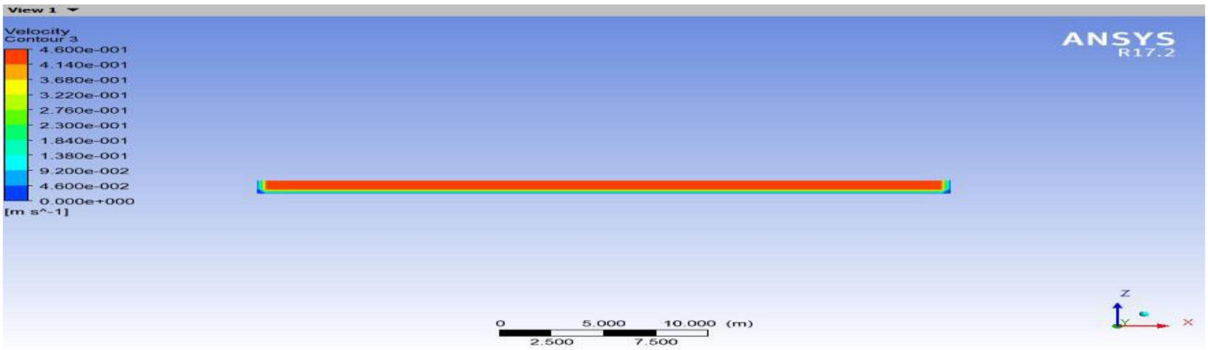

Figure 7(b) showing the velocity contour for section-16 *k-ω* model for converging angle-0.63°

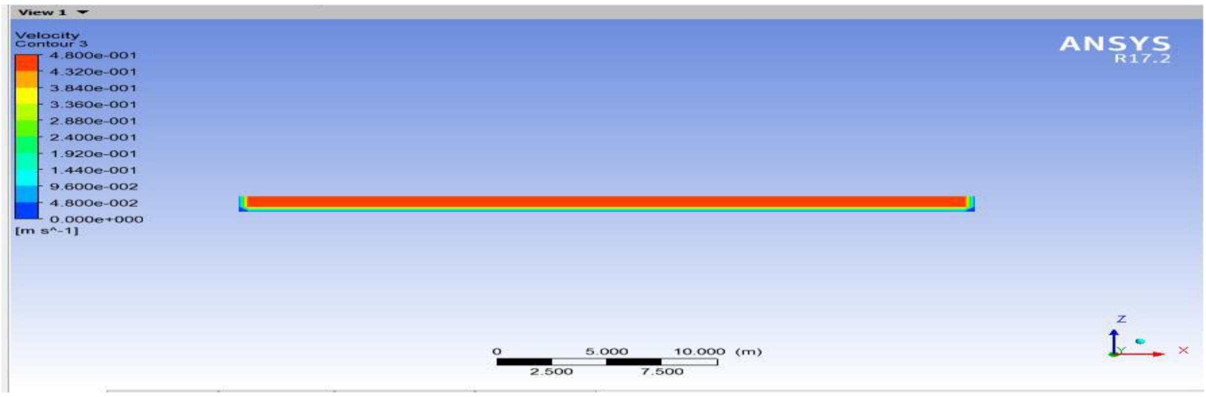

Figure 7(c) showing the velocity contour for section-16 *RANS* model for converging angle-0.63°

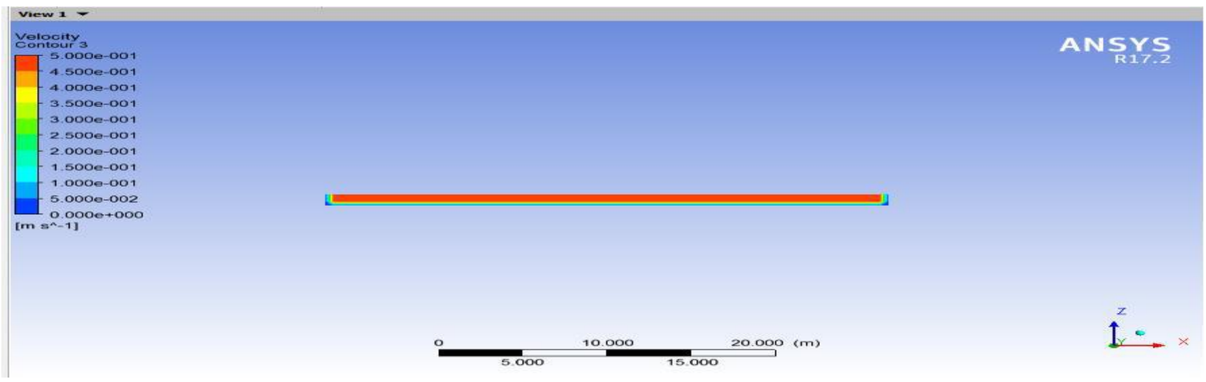

Figure 7(d) showing the velocity contour for section-16 *LES* model for converging angle-0.63°

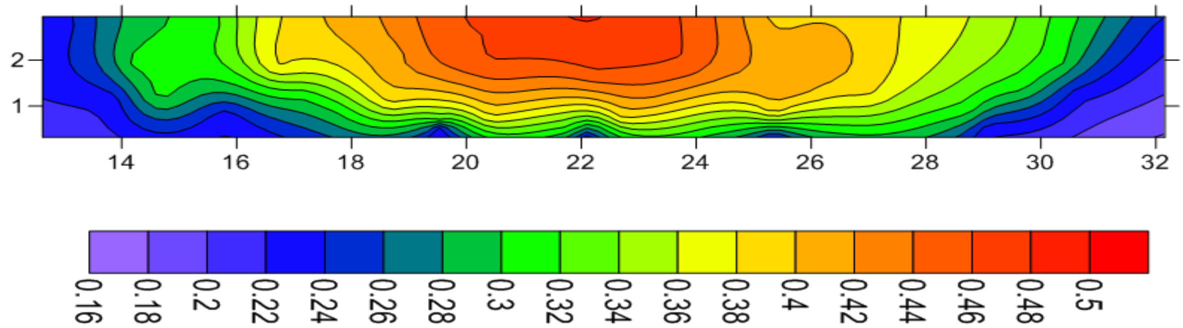

Figure 8. Showing the Velocity contour of section-17

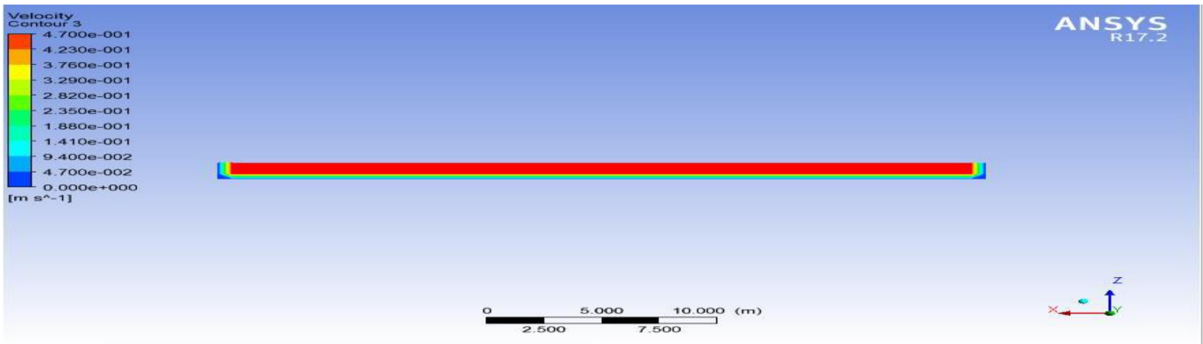

Figure 9(a). Showing the velocity contour for section -17 *k-ϵ* model for converging angle-0.63°

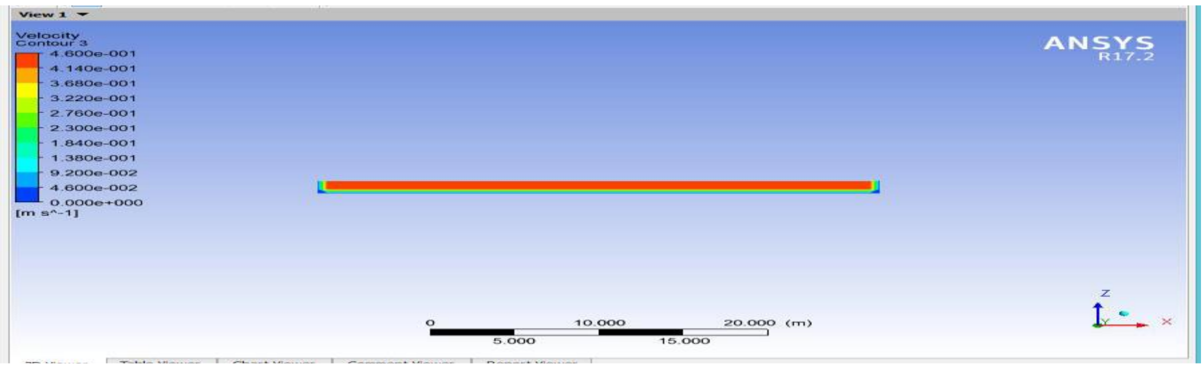

Figure 9(b). Showing the velocity contour for section -17 *k-ω* model for converging angle-0.63°

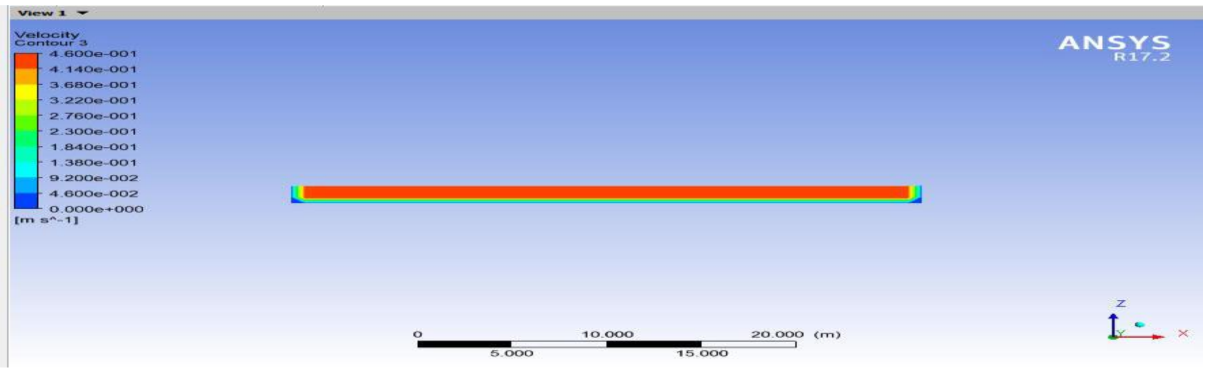

Figure 9(c). Showing the velocity contour for section -17 *RANS* model for converging angle-0.63°

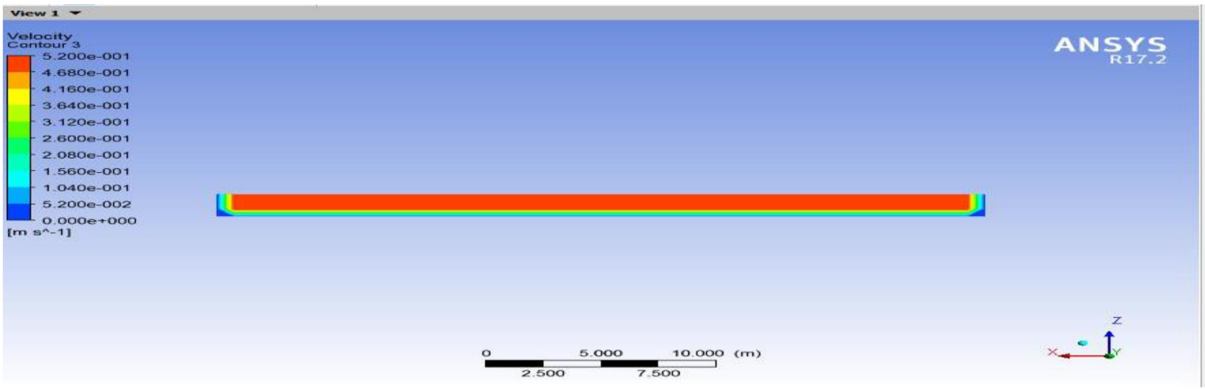

Figure 9(d). Showing the velocity contour for section -17 *LES* model for converging angle-0.17°

5. Conclusions

The analysis was performed in Kaskaskia River, Illinois (USA) for finding out the expansion section of reach-1 from it. It was found two expansion sections of (14, 16 and 17) with the expansion angle of 1.24° and 0.63° respectively. The real contours were drawn from software called SURFER and validate these contours with the ANSYS contour. It was concluded that the velocity increased in the converging sections and increased with the increase of flow. The study of observed velocity and predicted velocity was done using ANSYS FLUENT. *LES* was found to be best model among k-ϵ, k-ω and *RANS* for predicting velocity in diverging channel because it giving more accurate result than other models.

6. References

Alexandrov, S., & Barlat, F. (1999). Modelling axisymmetric flow through a converging channel with an arbitrary yield condition. Acta mechanica, 133 (1-4), 57-68.

Gandhi, B. K., Verma, H. K., & Abraham, B. (2010). Investigation of flow profile in open channels using CFD. *Roorkee, India.*

Knight, D., & Shamseldin, A. (Eds.). (2005). River basin modelling for flood risk mitigation. CRC Press

Khatua, K.K., and Patra, K.C. (2008). “Boundary shear stress distribution in compound open channel flow.” *J. Hydraul. Eng. ISH*, 12(3), 39–55.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, *32*(8), 1598-1605.

Sahu, M., Khatua, K.K., and Mahapatra, S.S. (2011). “A neural network approach for prediction of discharge in straight compound open channel flow.” *Flow Measure. Instrument.* 22, 438–446

Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA journal*, *26*(11), 1299-131.