CFD MODELING OF FLOW FIELD AROUND BRIDGE PIERS USING ANSYS FLUENT

A Thesis submitted in partial fulfilment of the requirement for the award of degree of

MASTER OF TECHNOLOGY
IN
HYDRAULICS & WATER RESOURCES ENGINEERING

BY

GURLEEN KAUR
(ROLL NO. 2K14/HFE/08)

Under the Guidance of
Dr. RAKESH MEHROTRA
Associate Professor
Department of Civil Engineering
Delhi Technological University
Delhi

DELHI TECHNOLOGICAL UNIVERSITY
(FORMERLY DELHI COLLEGE OF ENGINEERING)
DELHI – 110042
June-2016

CANDIDATE'S DECLARATION

I do hereby certify that the work presented is the report entitled "**CFD MODELING OF FLOW FIELD AROUND BRIDGE PIERS USING ANSYS FLUENT"** in the partial fulfillment of the requirements for the award of the degree of "Master of Technology" in Hydraulics & Water Ressources engineering submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of our own work carried out from January 2016 to July 2016 under the supervision of Dr. Rakesh Mehrotra (Associate Professor), Department of Civil Engineering. I have not submitted the matter embodied in the report for the award of any other degree or diploma.

Gurleen Kaur

Date: 5/7/2016 (2K14/HFE/08)

CERTIFICATE

This is to certify that above statement made by the candidate is correct to best of my knowledge.

Dr. Rakesh Mehrotra
(Associate Professor)
Department of Civil Engineering
Delhi Technological University

ACKNOWLEDGEMENT

I take this opportunity to express my profound gratitude and deep regards to **Dr. Rakesh Mehrotra** (Associate Professor, Civil Engineering Department, DTU) for his exemplary guidance, monitoring and constant encouragement throughout the course of this project work. The blessing, help and guidance given by him from time to time shall carry me a long way in life on which I am going to embark.

I would also like to thank **Dr. Nirendra Dev** (Head of Department, Civil Engineering Department, DTU) for extending his support and guidance.

Professors and faculties of the Department of Civil Engineering, DTU, have always extended their full co-operation and help. They have been kind enough to give their opinions on the project matter; I am deeply obliged to them. They have been a source of encouragement and have continuously been supporting me with their knowledge base, during the study. Several of well-wishers extended their help to me directly or indirectly and we are grateful to all of them without whom it would have been impossible for me to carry on my work.

1. ABSTRACT

Piers are the one of the most important parts of a bridge. They are the substructures which support all the weight and vertical load of the bridge and moreover, they resist all kind of forces, be it horizontal or transverse, acting on them. They transfer the live and dead load to the footings or the foundation of the bridge. There are various materials used for their construction these days. They are located in the middle points between the bridge abutments and at the span ends. Inappropriate design of the bridge piers can result in the failure of a bridge. Most of the times, the bridges built on the rivers are damaged and hence, they collapse due to defects in their hydraulic designing. One of the main reason of collapsing of a bridge is its inability to bear the shear stresses, hydraulic pressure and torque imposed by the flowing water on its piers. Other reason is the local scour that occurs around the base of the piers due the flowing water, caused by the shear impact and pressure at the bed of the channel, which creates a cavity at the base, making the piers unstable. The bridges built on water channels should be designed keeping in mind both these factors of collapsing and damage to the bridges. The design codes have not covered this area of hydraulics and water resources in deep and so, the traditional methods often lead to the construction of either under-secure bridges (prone to easy damage) or over-secure bridges (lead to over-costing). For more accurate results, one can design the bridges with the use of high efficiency software.

The present study is mainly focussed on the bridge piers in an open channel. In this study, a CFD analysis of the flow field around bridge piers has been done with the help of software, CFD ANSYS FLUENT, version 14.5. Various parameters like pressure acting on upstream face of the piers and on the channel bed surrounding the piers, shear stress acting on the piers and the channel bed surrounding the piers, velocity field around the piers and torque acting on the piers have been studied to predict the extent of damage and local scouring that can happen around the piers for various cases. The prediction of the local scour is then cross-checked by calculating the exact values of local scour for each case by using the software MARYLAND SHA BRIDGE SCOUR PROGRAM based on HEC-18 by FWHA. Various comparisons are done on how the above parameters change with the shape of the piers depth of the flow, velocity of the flow (Froude No.) and width of the piers.

The results show that the shape, depth, velocity and width of piers have a noticable effect on all the above parameters. It is noticed that curved shaped piers are a better option than the sharp edged piers in case of pressure, stresses, torque and less local scouring. The shape really influences the flow around a pier. The shear stress on the bed of the channel is a very effective parameter to predict the local scouring. It is noted that the shear stresses, hydraulic pressure, torque and hence, local scouring all increase with the increase in Froude No., depth of flow and increase in the pier width.

LIST OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

2. INTRODUCTION

A bridge is referred to any structure which is built over any obstacle like a river or a road, etc. to connect the two opposite sides of the obstacle. It is also called overpass, flyover, etc. A general bridge consists of various parts such as : abutments, bearings, deck, footings, parapets, wing walls, joints, parapets and piers.

Piers are the one of the most important part of a bridge. They are the substructures which support all the weight and vertical load of the bridge and moreover, they resist all kind of forces be it horizontal or transverse, acting on them. They transfer the vertical load to the footings or the foundation of the bridge. They are located in the middle points between the bridge abutments and at the span ends.

The bridge piers built in the rivers or water bodies come across various hydraulic forces due to the flowing water. These forces are maximum during the flood periods. If a bridge is unable to bear these forces, destabilization of the structure occurs. This can lead to huge life loss, property loss, money loss etc. There are methods in design codes present to evaluate the forces acting on the piers but they fail during the occurrence of a rain storm, a flash flood or any sudden rise in the water level in the channel. This happens because at the time of these kind of calamities, the downstream side of the pier is not submerged in the water. Only the upstream side is affected. But the methods in the design codes presume fully submersion of the piers in the water to calculate the forces. So, these design codes can't be used in case of for example big waves, large tidal currents etc. In the design codes, the water flow pressure in case of bridge engineering is not given so much attention and importance as it is given in case of pumps and tubes where this water flow pressure is given a name "water hammer". The main forces and its derivatives which act on the piers due to flowing water are the pressure, shear stresses and torque. Stronger the flow, stronger will be these forces. When we keep anything in the way of any moving fluid, be it air or water, the fluid enfolds around the obstruction. Either that wrap or enfold creates pressure or it creates vacuum around the obstruction. Due to this phenomenon, boundary layer separation occurs. Because of this a huge current formation occurs which slows down the approaching flow, and starts to reverse the direction of approaching flow. This steers the generation of vortices. These vortices and eddies lead to development water flow velocities in all the three directions i.e. x, y and z axis, giving rise to the shear stresses on all the three direction on the piers. Vortices lead to increase in pressure, velocities, stresses around the piers. It also gives rise to torque which has the capability of twisting the piers.

Now, all these forces which act on the piers, also act on the channel bed at the base of piers and surrounding them. This leads to local scouring. The scouring of bridges occur when due to several causes the material around the bridge piers, abutments, on the channel bed etc starts to eradicate and displace from their original places due to forces acting on them. The scouring around the bridge abutments or piers depends upon the nature of flow in the channel such velocity of the flow, discharge of the flow, angle of the flow striking the piers, nature of the channel bed, shape of the channel etc. There are various types of scouring namely:

a) Degradation & aggradation

b) Contraction Scour

c) Local Scour

d) Migration of stream channels

The scour which occurs at the base of the piers because of accelerating water is the Local Scour. It occurs due to the formation of eddies and vortices around the bed and base of the piers. When the fast flowing water gets accumulated at the upstream end of the pier or an abutment, the acceleration of the water occurs at the upstream nose of the pier. Formation of vortices occur which remove the bed material or bed sand from around the pier. The amount of bed material getting scooped out due to the vortices is much more than the amount of the bed material that is actually getting deposited there. So, an empty hole formation takes place around the pier base in the bed, which is called a scour hole. As the holes deepens, the horse shoe vortex becomes light.

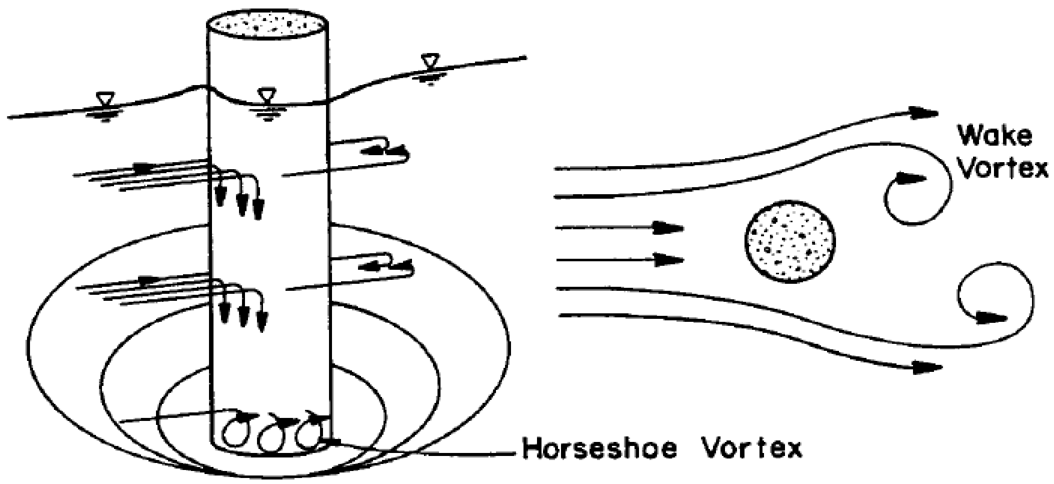


FIGURE 1: OCCURRENCE OF SCOUR AT A CIRCULAR PIER

There are various factors that affect the local scouring around the piers. They are :

a) Velocity and Discharge of the approaching flow: Higher is the velocity, more will be the scour. The local scour is different for different Froude Numbers, different foe a subcritical and a supercritical flow. Generally, only subcritical flows are taken into consideration that is the flows with less than one Froude Number.

b) Depth of the flow: Greater the depth, deeper is the local scour. This is in the case of piers. For abutments, the increase is not that much.

c) Width of the piers : The width of the piers has a straight impact on the depth of the local scour. Increase in the pier width leads to the deepening of the scour depth. But it happens until a certain limit only. After that limit, the scour depth stops to change.

d) Angle of attack and the length of the piers : For minimum scouring, according to the design codes, the piers should be aligned along the direction of the flow. When the piers are aligned along the direction of the flow i.e. at the angle of zero degree , the increase and decrease in the length of the piers does not have any effect on the nature and extent of the local scouring.

When the piers are inclined at some angles, the length plays an important role in deciding the scour the depth and its extent, depending upon the angle. With the increase in the length of the pier, the amount of scouring increases.

e) Bed material: The type of material the channel is made of, affects the nature, depth and extent of the local scour. The loose granules, the material with very cohesion, uniform size particles, light weight particles etc leads to high scouring because they will be easy to detach from the bed and carry from one place to another.

f) Shape of the pier nose: The shape of a pier nose directly influences the local scour. It can actually affect the local scour up to thirty percent. To reduce the effects of the horseshoe vortex at the nose of the piers, the streamlining of the upstream nose is important. The minimum scouring occurs at the round nose piers or circular piers, followed by the square nosed piers at the second and the maximum scouring occurs at the sharp nosed piers.

g) Nature of the bed and its configuration: The local scouring depends on the configuration of the channel bed for example if the bed has ripples, antidunes, dunes or just a plane smooth bed. These various configurations and structures of the bed depends upon various hydraulic parameters, the viscosity of the fluid flowing in the channel and the nature of the bed material. Sometimes it also depends on the temperature.

h) Ice and debris: The floating ice chunks and the debris can hinder the flow by blocking it and can change the dimensions of the piers. This can lead to increase in the local scouring.

In the present study, a Computational Fluid Dynamics analysis of the flow field around bridge piers will be done. The software used for this purpose will be CFD ANSYS FLUENT, version 14.5. A channel of a particular dimensions will be selected. The main parameters that are studied are:

a) Pressure on piers on upstream side by the flowing water i.e. flow impact.
b) Shear stresses acting on the piers in x, y and z direction due to the flowing water.
c) Pressure acting on the bed material at the base of the pier on the upstream side.
d) Shear stresses acting in x and y direction on the bed material at the base of the piers.
e) Torque acting on each pier due the forces acting on the piers because of the flowing water.
f) Velocity changes around the piers that lead to scouring, through velocity contours and velocity vectors.
g) Calculation of local scour depth for various cases using software MARYLAND SHA BRIDGE SCOUR PROGRAM.

Various comparisons are done on how the above parameters change with the:

a) Shape of the piers i.e. sharp nose and curved nose.
b) Depth of the flow.
c) Velocity of the flow.
d) Width of the piers.

3. ASSUMPTIONS MADE IN THE ANALYSIS

a) Two phase flow of fluid and gas is taken into consideration.
b) For the fluid, water is considered. For the gas, air is considered.
c) The case of clear water is considered.
d) The manual physical models take a lot of time and are expensive. Moreover, they can't give you the accurate results as compared to the well developed softwares. So, for this analysis, a section of river channel is built with piers in it and then it is analyzed by different boundary conditions. The software used for the flow is CFD ANSYS FLUENT (version 14.5).

4. ANSYS FLUENT

Computational Fluid Dynamics (CFD) is the field that helps in projecting the flow of any type of fluid in a particular domain, reactions taking place between various chemicals and similar occurrences by solving various kinds of equations like mass conservation equation, momentum conservation equation, species conservation equation, etc. The results which we obtain from the CFD help in the development of the product in a detailed way, troubleshooting any kind of problem, redesigning, etc. Computational Fluid Dynamics reduces the cost and moreover lessens the effort for an experiment.

The CFD works in the following manner :

a) The area on which we are working (domain), is divided into various particular no. of control volumes.
b) The particular equations for energy, mass etc, depending upon the chosen method for solving the problem are solved on the above stated control volumes.
c) The software discretizes the partial differential equations into the set of algebraic equations.
d) Solution is finally generated by numerically solving the previously stated algebraic equations.

4.1 STEPS OF SOLVING THE PROBLEM:

a) GEOMETRY : There are various ways of making the geometry. It can be made through the CAD software, through GAMBIT software, with the inbuilt geometry feature in the ANSYS FLUENT, etc. Don't include the features which you do not require, which won't be having any influence on the result and which can affect and complicate the nature of meshing. After making the geometry, the nature of domain is specified like if it's fluid, solid or gaseous etc

b) MESHING: The domain is divided into many small control volumes through meshing. One should keep in mind that the meshing should be such that it meets all the gradients and features of the concern like temperature, stress etc. Next the naming of the boundaries is done.

c) SETUP AND SOLVER: Once the meshing is done, the properties of the material are specified i.e. whether the material is a mixture of two phases or it is a solid or a fluid. The required physical model is chosen. Then the operating conditions are defined, followed by the defining of the boundary conditions. Initial values are defined.

d) POST PROCESSING: Results are calculated in the form of contours, streamlines, volume rendering, vectors, etc. The maximum and minimum values of every parameter are generated.

There are various plus points of the ANSYS software:

a) It is pocket friendly.

b) It saves a lot of time.

c) Gives better accuracy than the usual physical modelling.

d) Any kind of equipments are not required for the setup.

e) Any kind of boundary conditions can be applied and tested.

f) The speed of calculation can be very high for solving the problem.

5. OBJECTIVE OF THE STUDY

The objective of the present study is:

a) To find the exact pressure, shear stress in x, y, z directions, and torque acting on the upstream side of the faces of the bridge piers exerted by the flowing water using the multiphase k-e turbulence model in the ANSYS FLUENT.

b) To study the velocity field around the bridge piers through contours and vectors and analyse how it can lead to local scouring.

c) To study the pressure and shear stress acting in x and y direction at the base of the piers, on the channel bed by the approaching water.

d) To make comparison between sharp nose and curved nose piers on the basis of above parameters.

e) To compare the pressure, shear stress, velocity, torque for various depths, velocities, shapes and widths and find the most appropriate pier structure.

f) To calculate the values of local scour depth for each model of piers using the SHA BRIDGE SCOUR PROGRAM.

6. LITERATURE REVIEW

Sabita Madhvi Singh & P. R. Maiti [1]: "Flow Field and Scouring around Cylindrical Structure in Channel Bed"; Second Intl. Conference on Advances In Civil, Structural and Environmental Engineering- ACSEE 2014 : In this study, scour depth is estimated around circular piers by HEC-RAS for a straight channel. The flow characteristics around these structures are presented for different flow conditions. Mechanism of scouring phenomenon, formation of vortex and its consequent effect is discussed for straight channel. The present paper is concerned with the numerical simulations for the local scour around a bridge pier. The flow field around cylindrical pier is found by using ANSYS Fluent. The effect of pressure and turbulent kinetic energy is presented. It is stated that the flow characteristics influence the bed shear and shear along the slender structure. The erosion and deposition characteristic of soil particles in channel section depends on the flow characteristics. The scouring around slender structure depends on the discharge, velocity, and particle characteristics of the channel bed. The scour is determined using HEC RAS software. For the setup, the flow field around a circular cylinder of diameter 0.3m is determined for a free stream velocity 2 m/s. The slender structure is kept vertical in an impervious bed and there is no influence of boundary wall. The velocity vector, streamline and variation of pressure around the slender structure is analyzed.

Mohammad Vaghef , Hamed Dashtpeyma , Arash Adib , Javad Roohian [2]: "Numerical Analysis of Flow Pattern around square Bridge Piers by New Ansys Software"; 6 th National Congress on Civil Engineering, April 26-27, 2011, Semnan University, Semnan, Iran : The main aim of this this article was numerical analysis of flow pattern around square pier based on software simulating using Ansys-CFX (ver 12.1 2010). This simulation was based on are Froude numbers of flow in subcritical condition. Condition and specification of free surface and open channel had been mentioned directly. At the end generated data from software simulation was compared with laboratory researches. The locations of vortex systems and patterns of trailing wake-vortex systems was found.

Marieh Rajaie , Mohammad Reza Pirestani and Seyed Hossein Ghoreishi Najaf Abadi [3] : "Effects of Oval and Circular Piers on Velocity and Shear Stress Changes by using Open FOAM Software"; Special issue on Current World Environment, Volume 10, April 2015 : In this study, velocity and shear stress contour changes around the bridge piers of oval and circular shapes have been discussed and compared. 3-D modeling was done using Open FOAM software. Flow turbulence was considered using k-ε RNG turbulence model and governing equations were Navier-Stokes equations. The results showed that the bridge pier shape has a significant effect on the flow pattern and riverbed shear stress that is effective in prediction of the scour pattern. The results also showed that shear stress around the oval pier was lower than the circular one, and hence, less scour occurs around this type of pier.

I. Mistrová, D. Picka [4] : "Determine of velocity field with PIV and CFD during the flow around of bridge piers"; EPJ Web of Conferences- 45, EDP Sciences, 2013: The article described the research which dealt with physical and CFD of the velocity field during the flow around of bridge piers. Physical modelling has been carried out in Laboratory of water management research in Institute of Water Structures in Brno University of Technology - Faculty of Civil Engineering. To measure of the velocity field in profile of bridge piers were used laser measuring method PIV (Particle Image Velocimetry). The results of PIV served as a basis for comparing experimental data with CFD results of this type of flow in the commercial software ANSYS CFX. The project was divided into two parts, and physical and numerical modeling. The findings and outputs obtained from physical modeling served to show basic characteristics of the flow along the pillars, the pillars of the influence of the shape and influence the direction of deflection of the pillars on the flow in the space behind the pillars. The main objective of this research project was to obtain enough quality data for the initial calibration and subsequent verification of CFD model. The aim of the project was measured by PIV velocity field for three selected shapes gridiron pillars and three angles of deflection of the pillars of the longitudinal axis of flow.

Kamil H. M. Ali, Othman Karim [5] : "Simulation of flow around piers"; Journal of Hydraulic Research 40(2):161-174, March 2002: FLUENT CFD was used to predict the three-dimensional flow field around a circular cylinder. Solutions were obtained for rigid beds and for scour holes of different sizes resulting from different time-durations. The numerical results were used to obtain the variation of bed shear-stress around the cylinder. These results were used in the sediment continuity equation to obtain an expression for the variation of scour depth with time. The asymptotic scour depth was found to depend on three dimensionless numbers: the pile number, the sediment size number and the duration time number. The theoretical relationship was calibrated using various laboratory and field results.

Kassem Salah El-Alfy [6] : "Backwater Rise Due To Flow Constriction By Bridge Piers"; Thirteenth International Water Technology Conference, IWTC 13,Egypt, 2009 : the backwater rise due to bridge piers was experimentally investigated for extreme ratios of piers thickness to channel width (tPS/B) under a wide range of both subcritical and supercritical flow conditions between bridge piers. The present study illustrates that the backwater rise upstream bridge piers depends mainly on both flow type between piers and constriction ratio, while it is secondary depends on geometrical shape of pier end noses. Also, the experimental results illustrates that the backwater rise at supercritical flow conditions between piers agreed satisfactory with the corresponding computed values from the most widely relationships published by Yarnell than that for subcritical flow conditions between piers. The results of the backwater rise due to constriction of flow by bridge piers presented in this research paper are employed in development of two formulas, which could be used in computing backwater rise at both subcritical and supercritical flows between bridge piers.

Wenrui Huang, Qiping Yang, Hong Xiao [7] : "CFD modeling of scale effects on turbulence flow and scour around bridge piers"; 18th, January 2008: In this study, computational model simulations using a 3D CFD model were conducted to examine scale effects on turbulent flow and sediment scour. For the large-scale model, the physical scale and boundary velocity were set up from the small scale model based on the Froude similarity law. Results of flow and sediment scour were obtained from two different approaches: (a) Froude similarity which is commonly used in physical modeling and (b) full scale 3D CFD modeling. Unlike physical modeling in which the effect of turbulent Reynolds number is ignored, the CFD model employs a 2nd order turbulent model to calculate turbulent velocity and sediment scour. Effects of scale on turbulence flow and sediment scour were investigated by comparing different results obtained from a full scale numerical model to those derived from the Froude similarity method.

Sreedhara B M, Sanooj A, Manu , S Mandal [8] : "Simulation of Local Scour around Circular and Round Nosed Bridge Pier using REEF3D"; IJIRSET, volume 5, special issue 9, May 2016: In this study they stated that sediment transport and resulting local scour around the bridge pier can lead to the failure of structure. The knowledge and understanding the erosion and prediction of scour depth is difficult for the structural design. In this study the physics of local scour under current condition are modelled using CFD tool. The results obtained from numerical model are compared with experimental observations, showing good agreement. The study also concludes the effect of shape of pier on, velocity profile, mechanism and magnitude of local scour. The 3D CFD model REEF3D is used to calculate the detailed flow field and the resulting sediment transport pattern around the pier.

Mete Koken and George Constantinescu [9] : "An investigation of the flow and scour mechanisms around isolated spur dikes in a shallow open channel: 1. Conditions corresponding to the initiation of the erosion and deposition process"; 5th August, 2008: The present study investigated the flow physics and the role played by the main coherent structures in the scouring processes around a vertical spur dike in a straight channel at conditions corresponding to the start (flat bed) of the scouring process. Large eddy simulation (LES) was performed at a relatively low channel Reynolds number (Re = 18,000), in the range where most flume studies with clear water scour conditions are conducted. Similar to these studies, the incoming flow was fully turbulent and contained realistic turbulence fluctuations. Visualization experiments were conducted to better understand the nature of the interactions between the dominant coherent structures playing a role in the erosion process. It was found that the structure of the horseshoe vortex (HV) system at the base of the spur dike changes considerably in time and in vertical sections perpendicular to the trajectory defined by the axis of the main necklace vortex.

7. METHODOLOGY

A rectangular channel section is analysed using the software ANSYS FLUENT. For the analysis the following numerical steps are involved in the process:

a) **Pre-processing**

- Geometry
- Meshing
- Physics
- Problem Solving

a) GEOMETRY: There are various ways of making the geometry. It can be made through the CAD software, GAMBIT software, with the inbuilt geometry feature in the ANSYS FLUENT, etc. Don't include the features which you don't require, which won't be having any influence on the result and which can affect and complicate the nature of meshing. After making the geometry, the nature of domain is specified like if it's fluid, solid or gaseous etc

b) MESHING: The domain is divided into many small control volumes through meshing. One should keep in mind that the meshing should be such that it meets all the gradients and features of the concern like temperature, stress etc. Next the naming of the boundaries is done.

c) SETUP AND SOLVER: Once the meshing is done, the properties of the material are specified i.e. whether the material is a mixture of two phases or it is a solid or a fluid. The required physical model is chosen. Then the operating conditions are defined, followed by the defining of the boundary conditions. Initial values are defined.

b) POST PROCESSING: Results are calculated in the form of contours, streamlines, volume rendering, vectors, etc. The maximum and minimum values of every parameter are generated.

HOW TO MAKE A MODEL?

ANSYS FLUENT has an inbuilt design modular to create any kind of desired geometry with desired dimensions. Once a geometry is created, meshing is done, then boundary conditions and other required inputs are provided to obtain the result.

7.1.1. GEOMETRY SETUP

The geometry is made in the Design Modular feature of the ANSYS workbench software. Most of the precision and accuracy of the result will be dependent on the accuracy with which you are making the geometry. The dimensions should be accurate. For the present study, the dimensions of the channel were as following :

TABLE 1: NOMENCLATURE OF THE GEOMETRIES

DIMENSION	VALUE (m)
Length of the channel section	60
Width of the channel section	32

Depth- I of the flow	3.5
Depth- II of the flow	4
Depth- III of the flow	4.5
No. of piers	2
Length of the Rectangular pier	3
Width of the Rectangular pier	1
Distance between the Rectangular piers	10
Length of Curved edge piers (elongated piers)	3
Width- I of Curved edged piers	1
Width- II of Curved edged piers	1.5
Distance between Curved piers of width- I	10
Distance between Curved piers of width- II	9.67
Distance of piers from the Inlet	30

FIGURE 2: GEOMETRY OF MODEL A (RECTANGULAR PIER OF WIDTH 1m)

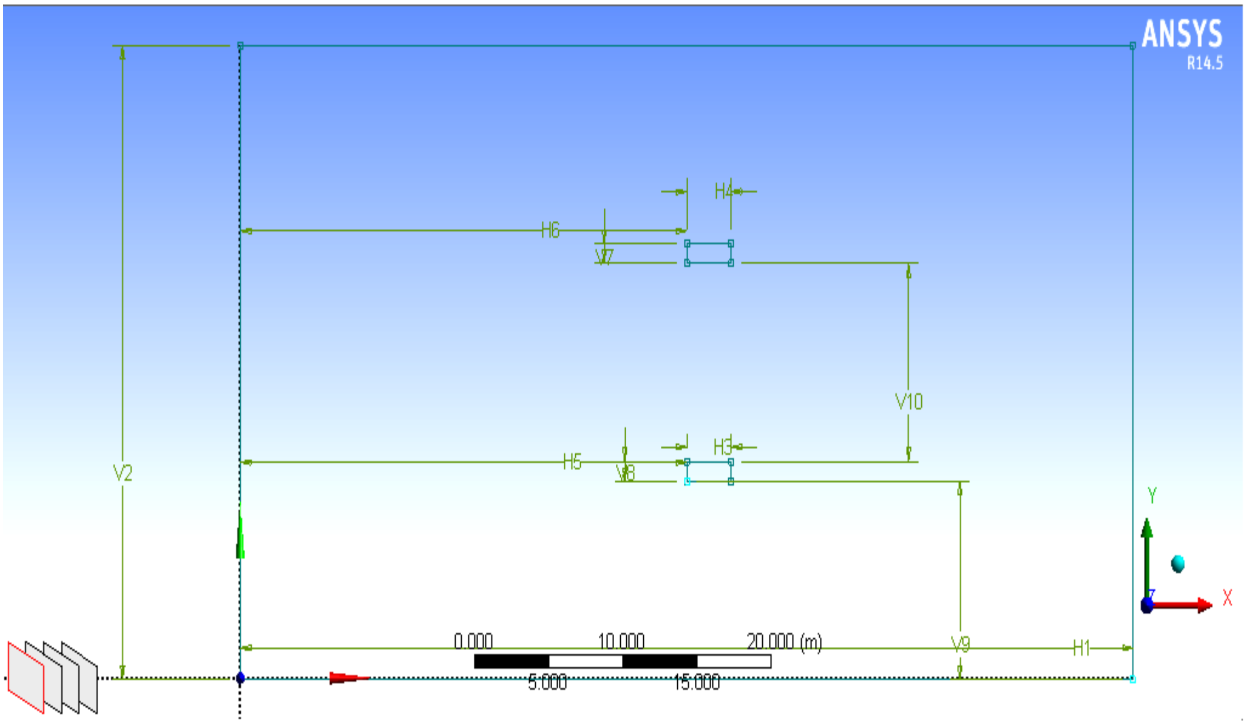
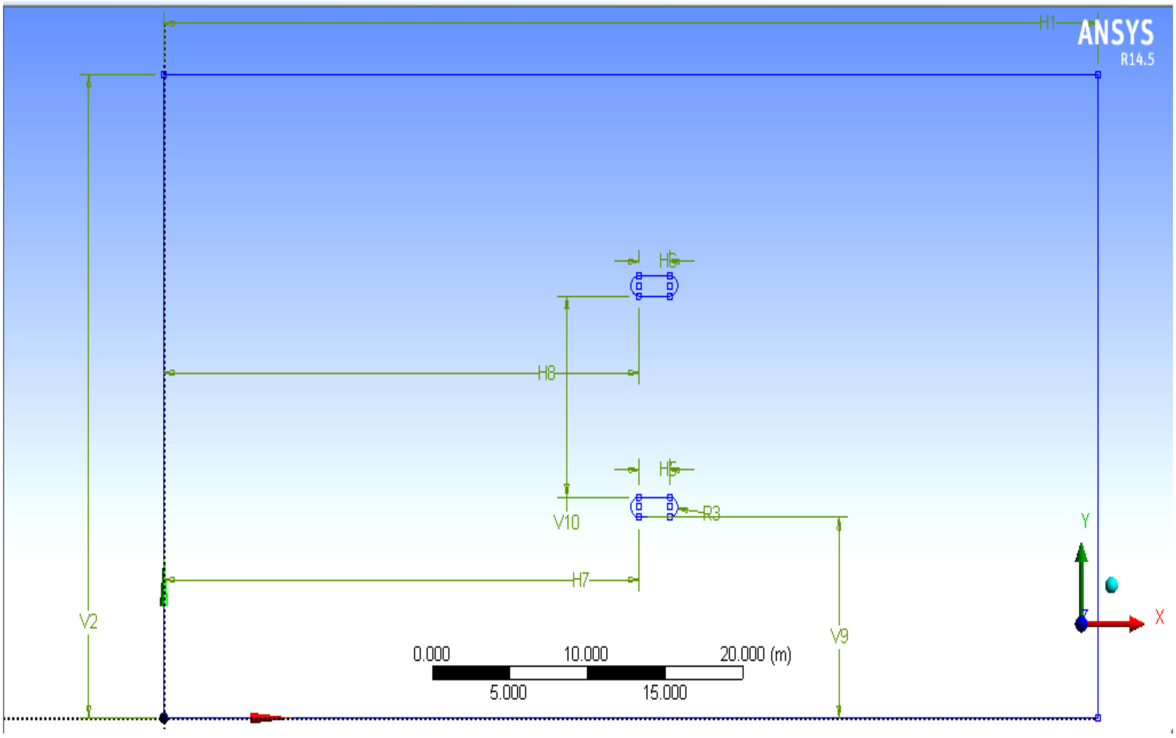
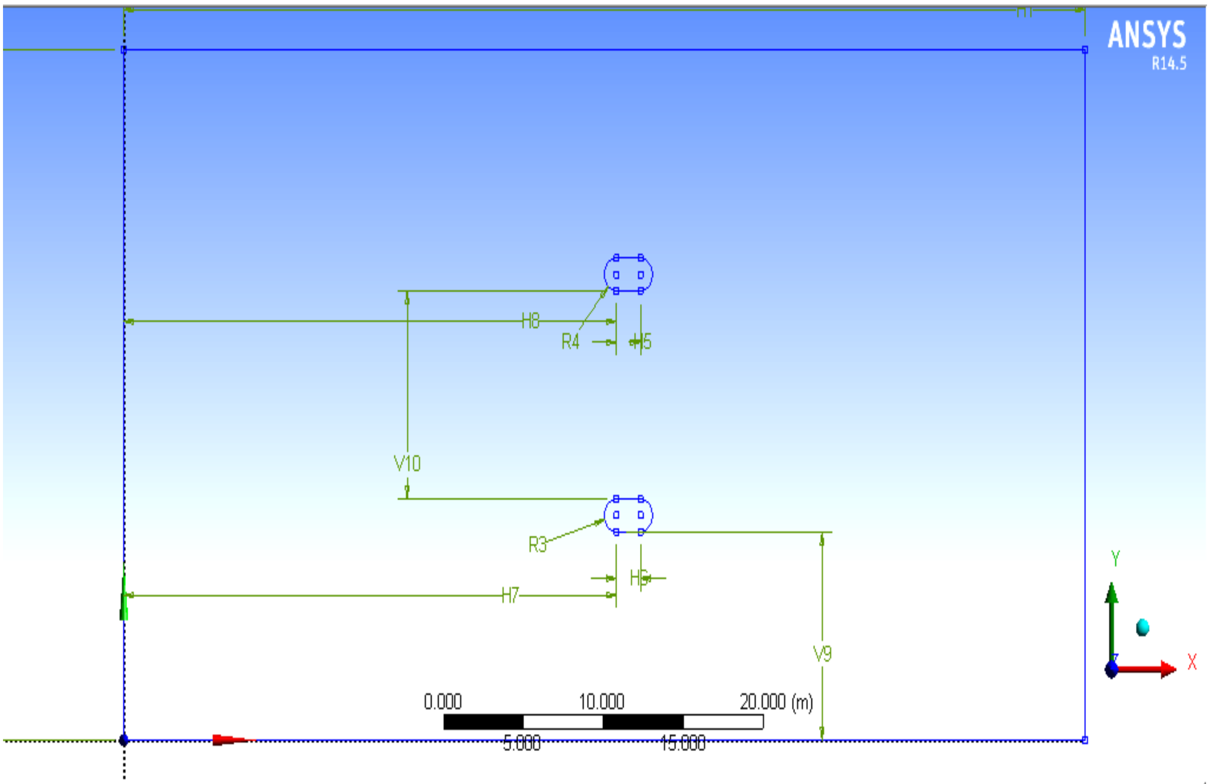


FIG 3 : GEOMETRY OF MODEL B (CURVED PIER OF WIDTH 1m)



FID 4 : GEOMETRY OF MODEL C (CURVED PIER OF WIDTH 1.5m)



7.1.2. MESHING

The three models mentioned above were meshed one by one, indistinctly. The maximum face size of cell was chosen to be 1.1. The meshing of all the models are:

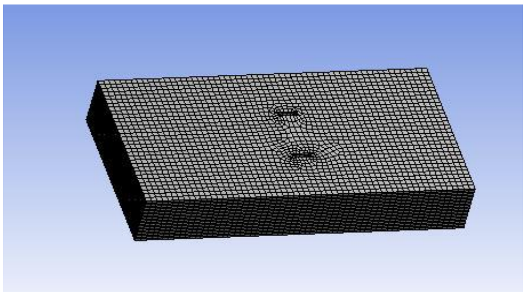


FIG 5: MESHING OF MODEL A

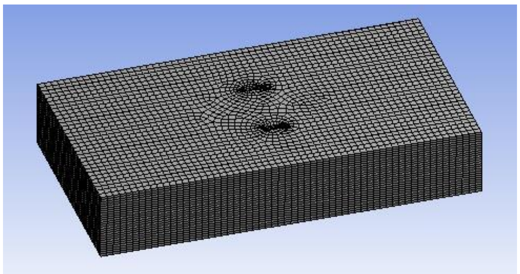


FIG 6: MESHING OF MODEL B

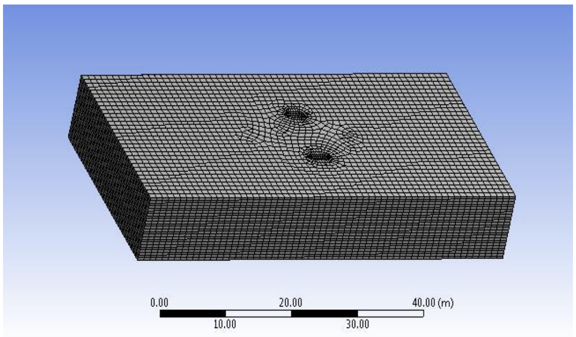


FIG 7: MESHING OF MODEL C

7.1.3. SETUP IN ANSYS FLUENT

The setup procedure is as followed:

- Under the operating conditions, the effect and value of gravity is defined along the negative y-axis.
- The flow of the water in the channel is along the positive x-axis.
- Under the type, "Pressure-Based" is chosen.
- Under the velocity formulation, "Absolute" is chosen.
- The time is chosen to be steady.
- There are various types of models. From those, we select "Multiphase- Volume of Fluid" model. This means the flow is going to have more than one phase. Under the multiphase flow, the open channel flow will be selected under the implicit forces.
- The viscous model chosen for this present study is Realizable K-E model with non equilibrium wall functions and curvature corrections.
- The phases or the materials considered in this study are water and air.
- For boundary conditions, inlet was assigned as pressure inlet and provided with the required velocity and free surface level. Outlet was assigned as pressure outlet and assigned neighbouring cell method. Walls, bottom and free surface were assigned no slip conditions. The free surface is assigned symmetry condition.
- The reference values are computed from the inlet.

- For solution, the scheme chosen is PISO and PRESTO!
- Standard initialization is chosen for the solution initialization. The solution is computed from the inlet.
- The result is calculated and the number of iterations chosen are 250.

The base of this study is the k-e model. K-e models are the turbulence models used to calculate and analyse the characteristics and parameters if the turbulent flow. There are three types of turbulence models present in the ANSYS FLUENT namely:

- Standard
- RNG
- Realizable

The model used in this study is the realizable k-e turbulence model. Features of this model are as following:

- It gives more accuracy in terms of turbulent viscosity.
- Vorticity fluctuations are well and more accurately considered and calculated.
- Realizable is its name because it highly satisfies the constraints on the stresses. The stresses here are Reynold's stresses. So, it is better than the other turbulent models.
- It helps in providing very accurate performance for the turbulent flows that involve boundary layers under high pressures, recirculation, vortices, formation of eddies, high powerful rotations, recirculations, etc.

The transport equations for the Realizable k-e model are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\varepsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}P_b + S_\varepsilon$$

$$C_1 = max\left[0.43, \frac{n}{n+5}\right],$$

$$n = S\frac{k}{\varepsilon},$$

$$S = \sqrt{2S_{ij}S_{ij}}$$

$P_k =$ *Turbulence Kinetic Energy Generation because of velocity gradients.*

$P_b =$ *Turbulence Kinetic Energy Generation because of buoyancy*

$u_i = Velocity$

$x_j = Position$

$P = Pressure$

$\rho = Density$

$v = Viscosity$

$t = Time$

Figures showing the iterations and convergence for the models involved in the study:

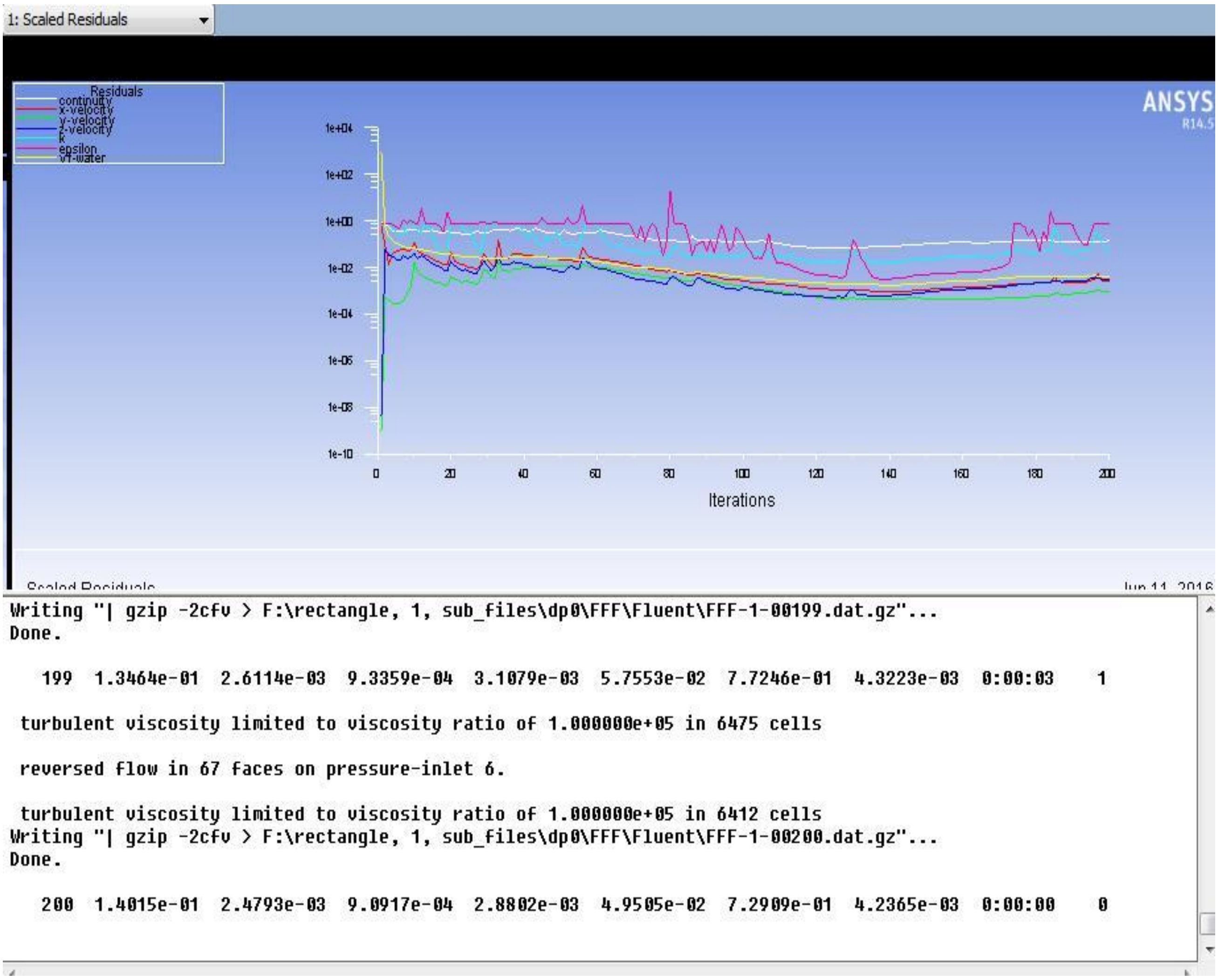


FIGURE 8: WINDOW IN FLUENT FOR THE CONVERGENCE FOR RECTANGULAR PIERS OF WIDTH 1m, VELOCITY 4m/s, DEPTH 3.5m

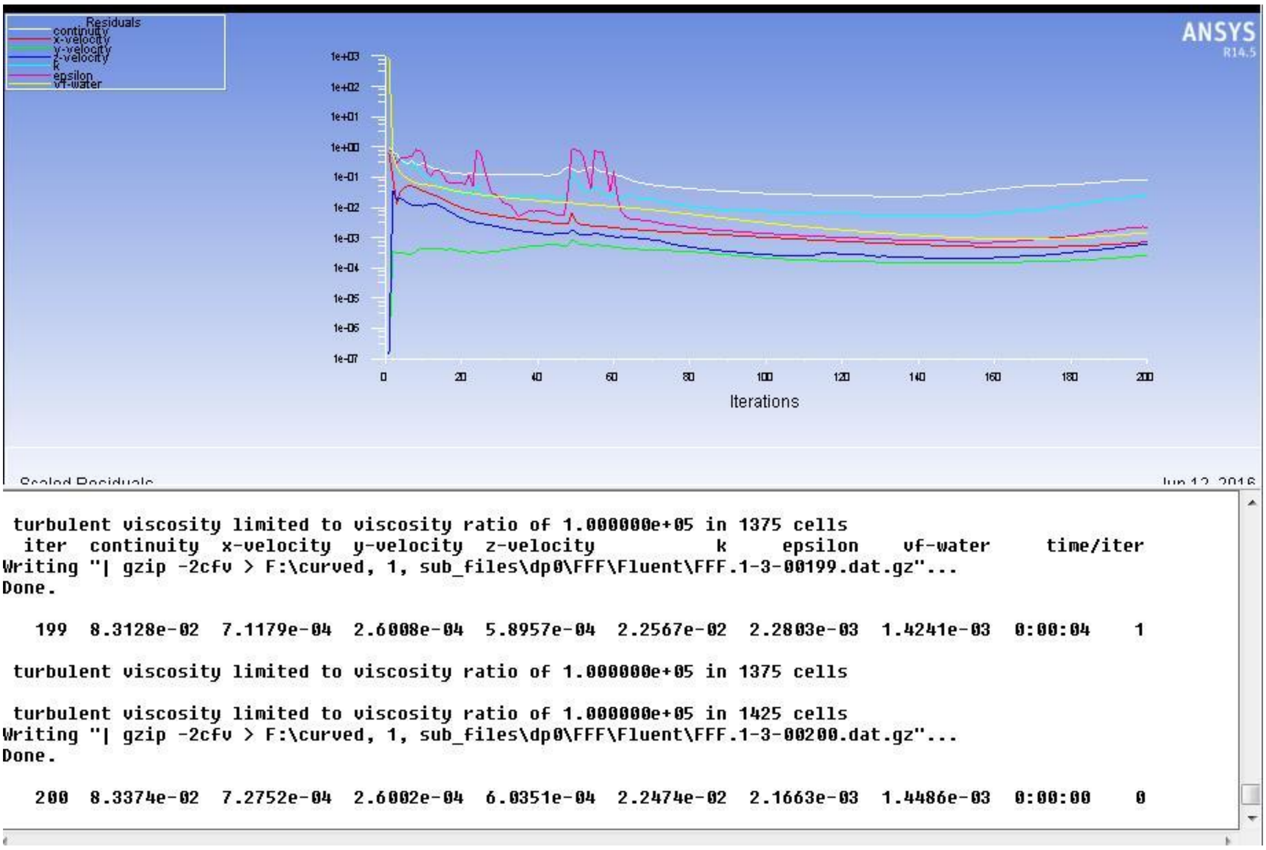


FIGURE 9: WINDOW IN FLUENT FOR THE CONVERGENCE FOR CURVED PIERS OF WIDTH 1m, VELOCITY 4m/s, DEPTH 3.5m

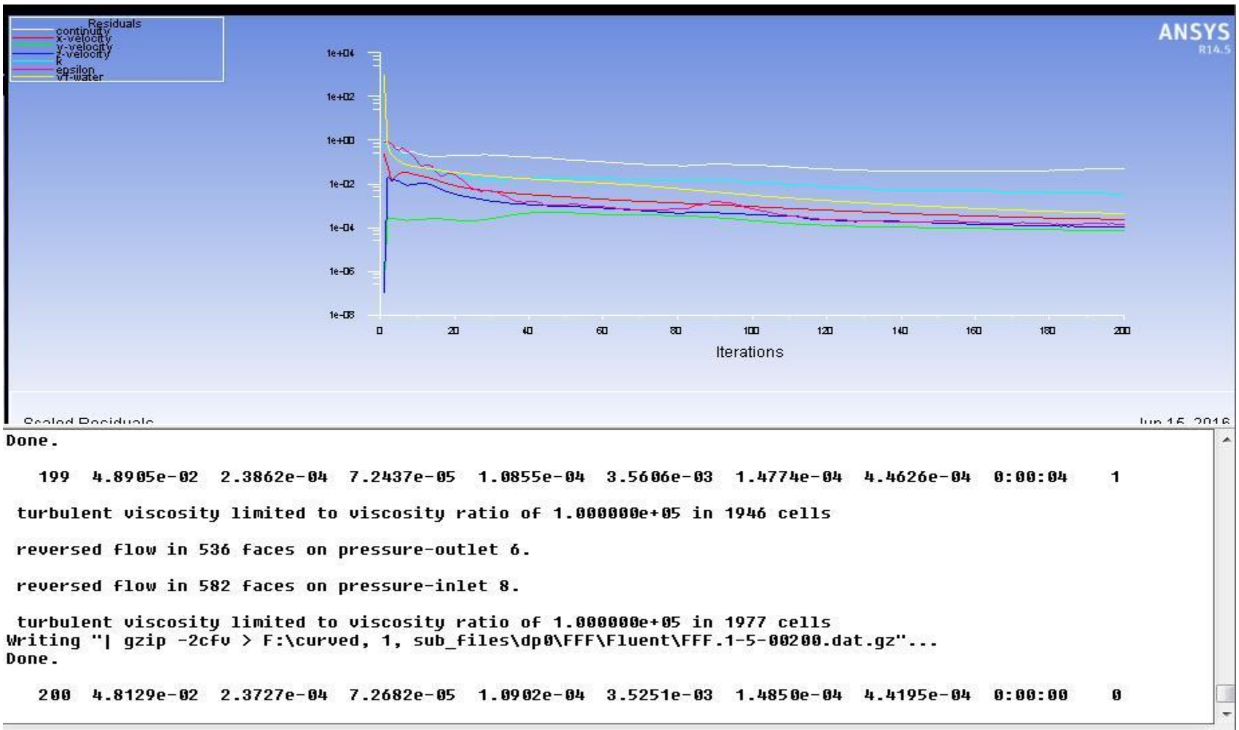


FIGURE 10: WINDOW IN FLUENT FOR THE CONVERGENCE FOR CURVED PIERS OF WIDTH 1m, VELOCITY 6.5m/s, DEPTH 3.5m

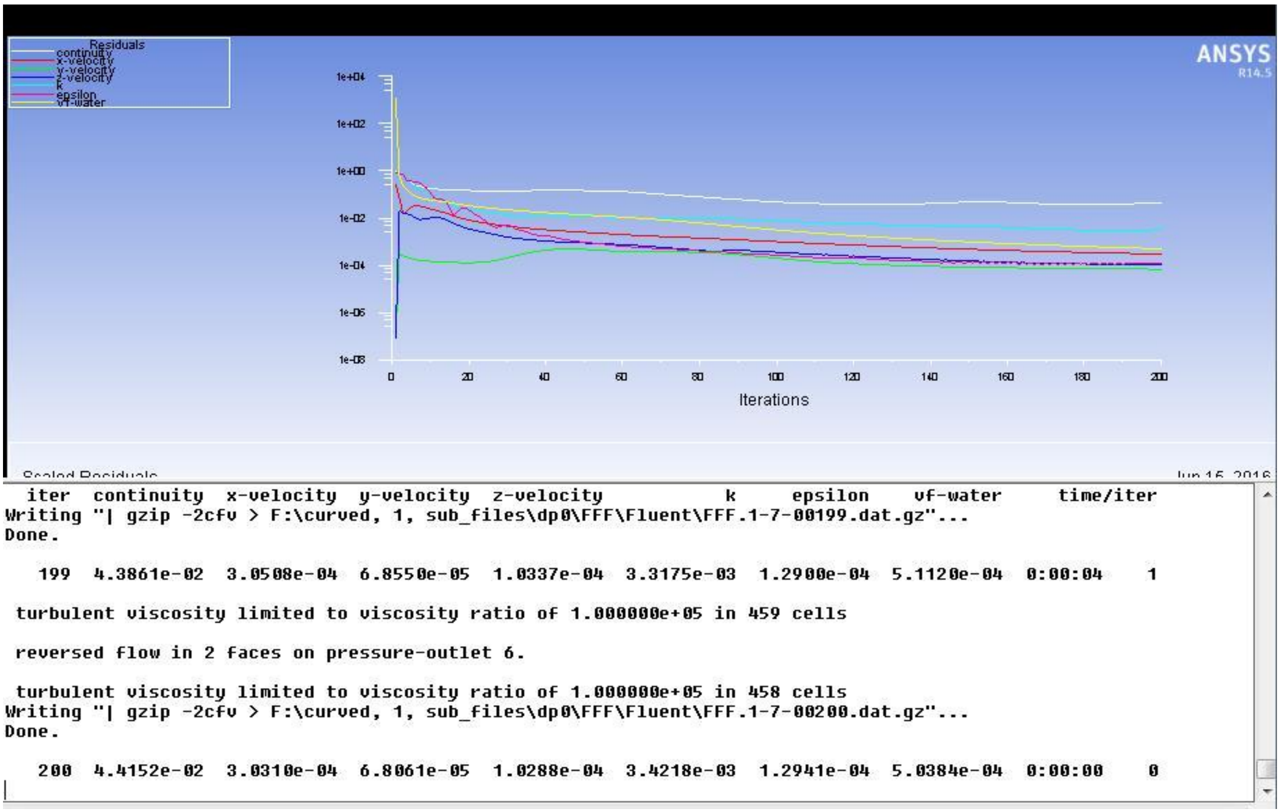


FIGURE 11: WINDOW IN FLUENT FOR THE CONVERGENCE FOR CURVED PIERS OF WIDTH 1m, VELOCITY 6.5m/s, DEPTH 4m

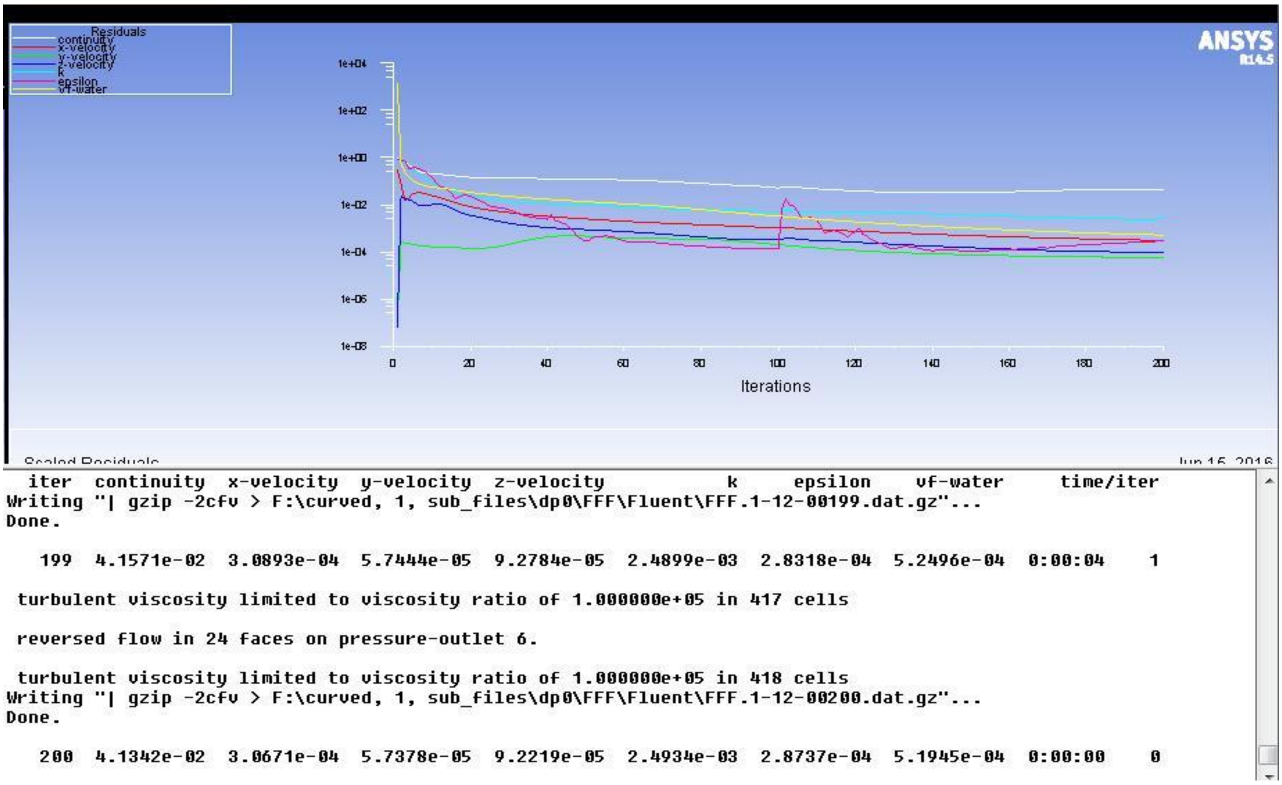


FIGURE 12: WINDOW IN FLUENT FOR THE CONVERGENCE FOR CURVED PIERS OF WIDTH 1m, VELOCITY 6.5m/s, DEPTH 4.5m

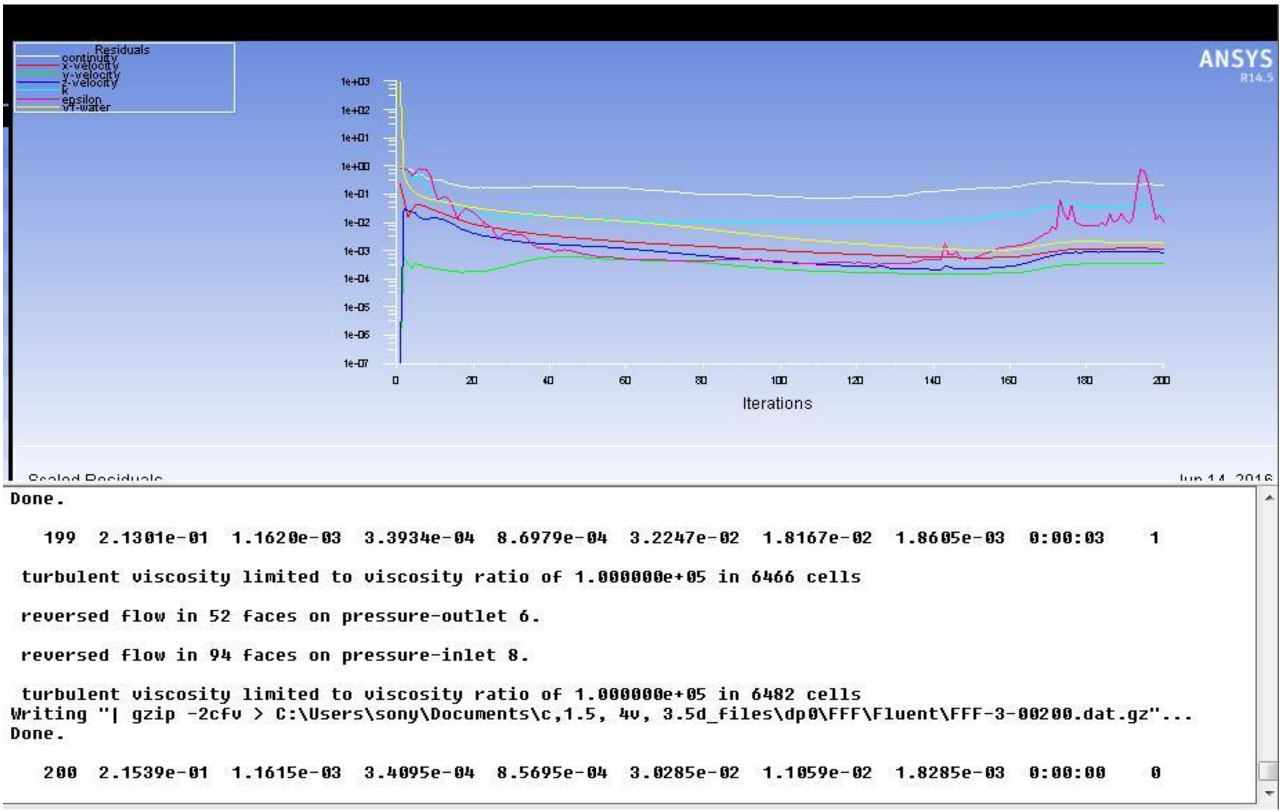


FIGURE 13: WINDOW IN FLUENT FOR THE CONVERGENCE FOR CURVED PIERS OF WIDTH 1.5m, VELOCITY 6.5m/s, DEPTH 3.5m

8. NUMERICAL DATA

There are total 9 number of cases in the present study for the analysis. They are:

- Flow around the Rectangular piers of width 1m with velocity 4m/s and depth 3.5m.
- Flow around the Curved piers of width 1m with velocity 4m/s and depth 3.5m.
- Flow around the Curved piers of width 1m with velocity 4m/s and depth 3.5m.
- Flow around the Curved piers of width 1m with velocity 6.5m/s and depth 3.5m.
- Flow around the Curved piers of width 1m with velocity 6.5m/s and depth 4m.
- Flow around the Curved piers of width 1m with velocity 6.5m/s and depth 4.5m.
- Flow around the Curved piers of width 1.5m with velocity 6.5m/s and depth 3.5m.

For all the cases mentioned above, the parameters studies are:

a) Pressure on piers on upstream side by the flowing water i.e. flow impact.
b) Shear stresses acting on the piers in x, y and z direction due to the flowing water.
c) Pressure acting on the bed material at the base of the pier on the upstream side.
d) Shear stresses acting in x and y direction on the bed material at the base of the piers.
e) Torque acting on each pier due the forces acting on the piers because of the flowing water.
f) Velocity changes around the piers through velocity contours and velocity vectors, which can lead to the formation of local scour.

Various comparisons are done on how the above parameters change with the:

a) Shape of the piers i.e. sharp edged and curved nose.
b) Depth of the flow.
c) Velocity of the flow.
d) Width of the piers.

For each of the cases, following values are calculated :

- Maximum pressure on the piers on the upstream side.
- Maximum shear on the piers in x-direction.
- Maximum shear on the piers in y-direction.
- Maximum shear on the piers in z-direction.
- Maximum shear on the bed around the piers in x-direction.
- Maximum shear on the bed around the piers in y-direction.
- Maximum pressure acting on the bed around the piers.
- Maximum torque acting on the piers.
- Maximum velocity around the pier near the bed.
- There are values calculated for pressure on its upstream face at near the bottom level, mid-depth and at the full depth level.
- There are values calculated for the pressure at the bed at near the inlet, near the upstream face tip of the pier and near the outlet.

The following values of input values are considered:

- Velocity I= 4m/s

- Velocity II= 6.5m/s
- Depth I= 3.5m
- Depth II= 4m
- Depth III= 4.5m
- Width I of the piers = 1m
- Width II of the piers = 1.5m
- Pier shape I= Sharp nose (Rectangle)
- Pier shape II= Round nose (Curved or elongated piers or oval piers)

TABLE 2: MODEL I-VALUES FOR RECTANGULAR PIERS WITH VELOCITY 4m/s, WIDTH OF PIERS 1m, DEPTH OF FLOW 3.5m

PARAMETER	VALUES
Maximum pressure on piers	42956.8 Pa
Maximum shear on piers in x-direction	162.249 Pa
Maximum shear on piers in y-direction	124.705 Pa
Maximum shear on piers in z-direction	69.523 Pa
Maximum pressure at the bottom	38051.5 Pa
Maximum shear at the bottom near the piers in x- direction	165.344 Pa
Maximum shear at the bottom near the piers in y- direction	102.86 Pa
Maximum Torque acting on the piers	26603.1 Nm

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	39944.2
	1.75	28388
	3.5	6491.77

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	39197.3
	30	38568.8
	60	20453.2

TABLE 3: MODEL II- VALUES FOR CURVED PIERS WITH VELOCITY 4m/s, WIDTH OF PIERS 1m, DEPTH OF FLOW 3.5m

PARAMETER	VALUES
Maximum pressure on piers	40706.9 Pa
Maximum shear on piers in x-direction	63.64 Pa
Maximum shear on piers in y-direction	124.365 Pa
Maximum shear on piers in z-direction	58.84 Pa
Maximum pressure at the bottom	37804.8 Pa
Maximum shear at the bottom near the piers in x- direction	147.964 Pa
Maximum shear at the bottom near the piers in y- direction	77.67 Pa
Maximum Torque acting on the piers	10389.3 Nm

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	38742
	1.75	26503.2

	3.5	5742.19

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	28924.7
	30	38139.5
	60	20300

TABLE 4: MODEL III- VALUES FOR CURVED PIERS WITH VELOCITY 6.5m/s, WIDTH OF PIERS 1m, DEPTH OF FLOW 3.5m

PARAMETER	VALUES
Maximum pressure on piers	53937.7 Pa
Maximum shear on piers in x-direction	195.88 Pa
Maximum shear on piers in y-direction	159.87 Pa
Maximum shear on piers in z-direction	79.799 Pa
Maximum pressure at the bottom	48358.8 Pa
Maximum shear at the bottom near the piers in x- direction	172.35 Pa
Maximum shear at the bottom near the piers in y- direction	120.98 Pa
Maximum Torque acting on the piers	74069.1 Nm

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	50633.2
	1.75	39803.3
	3.5	22396.6

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	33703.7
	30	49217.2
	60	22144

TABLE 5: MODEL IV-VALUES FOR CURVED PIERS WITH VELOCITY 6.5m/s, WIDTH OF PIERS 1.5m, DEPTH OF FLOW 3.5m

PARAMETER	VALUES
Maximum pressure on piers	54167.4 Pa
Maximum shear on piers in x-direction	213.136 Pa
Maximum shear on piers in y-direction	170.123 Pa
Maximum shear on piers in z-direction	81.348 Pa
Maximum pressure at the bottom	51335.3 Pa
Maximum shear at the bottom near the piers in x- direction	197.295 Pa
Maximum shear at the bottom near the piers in y- direction	139.781 Pa
Maximum Torque acting on the piers	-82569.5 Nm

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	52389
	1.75	41390.2
	3.5	22895.9

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	34804.2
	30	51315.4
	60	27969.7

TABLE 6: MODEL V- VALUES FOR CURVED PIERS WITH VELOCITY 6.5m/s, WIDTH OF PIERS 1m, DEPTH OF FLOW 4m

PARAMETER	VALUES
Maximum pressure on piers	59364.4 Pa
Maximum shear on piers in x-direction	257.552 Pa
Maximum shear on piers in y-direction	192.942 Pa
Maximum shear on piers in z-direction	81.1357 Pa
Maximum pressure at the bottom	52666.4 Pa
Maximum shear at the bottom near the piers in x- direction	232.62 Pa
Maximum shear at the bottom near the piers in y- direction	157.94 Pa
Maximum Torque acting on the piers	99184.3 Nm

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	55226.3 Pa
	2	42187.6 Pa
	4	212742.8

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	38573.4
	30	53747.5
	60	30837.8

TABLE 7: MODEL VI- VALUES FOR CURVED PIERS WITH VELOCITY 6.5m/s, WIDTH OF PIERS 1m, DEPTH OF FLOW 4.5m

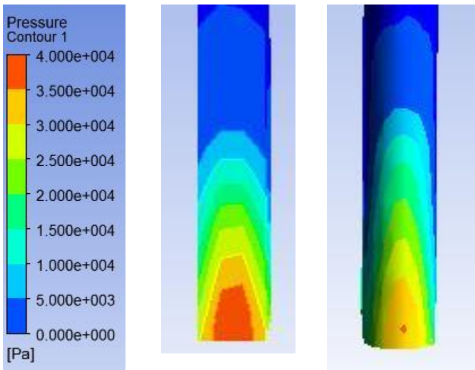
PARAMETER	VALUES
Maximum pressure on piers	64777.2 Pa
Maximum shear on piers in x-direction	213.959 Pa
Maximum shear on piers in y-direction	285.419 Pa
Maximum shear on piers in z-direction	83.21 Pa
Maximum pressure at the bottom	56877.5 Pa
Maximum shear at the bottom near the piers in x- direction	258.419 Pa
Maximum shear at the bottom near the piers in y- direction	174.83 Pa
Maximum Torque acting on the piers	126900

PARAMETER	DEPTH (m)	VALUE (Pa)
Pressure on piers at upstream side	0.15	59966.1
	2.25	44771.6
	4.5	20240.8

PARAMETER	DISTANCE FROM THE INLET (m)	VALUE (Pa)
Pressure at the bed	0	43319
	30	58196.1

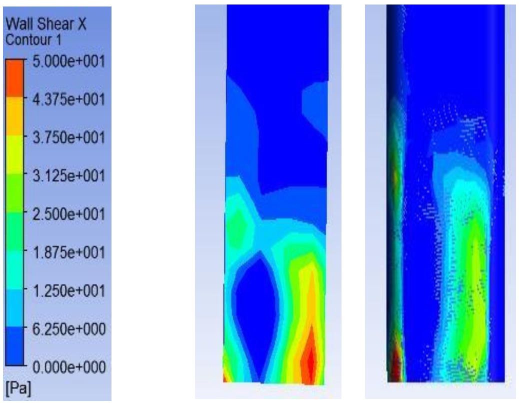
8.1 COMPARISON BETWEEN RECTANGULAR AND CURVED PIER:

Width of piers= 1m, Velocity of flow= 4m/s, Depth of flow= 3.5m



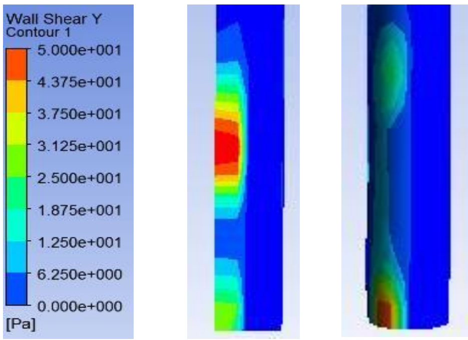
(a) **(b)**

FIG 14: PRESSURE CONTOUR ON (a) RECTANGULAR PIER AND (b) CURVED PIER



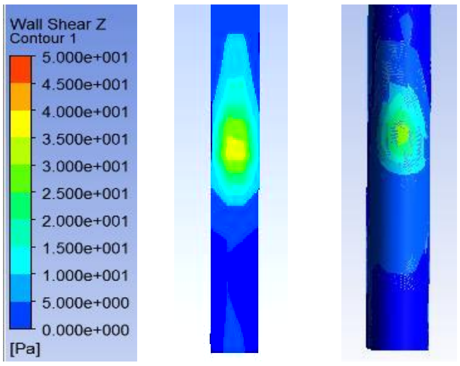
(a) **(b)**

FIG 15: SHEAR CONTOUR IN X DIRECTION ON (a) RECTANGULAR PIER AND (b) CURVED PIER



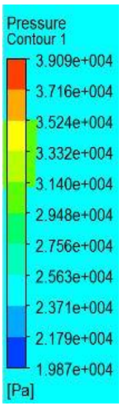
(a) **(b)**

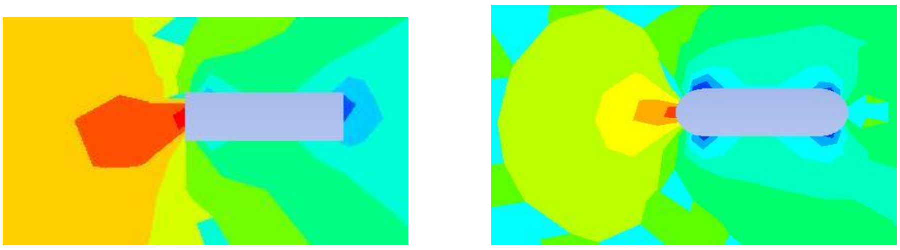
FIG 16: SHEAR CONTOUR IN Y DIRECTION ON (a) RECTANGULAR PIER AND (b) CURVED PIER



(a) **(b)**

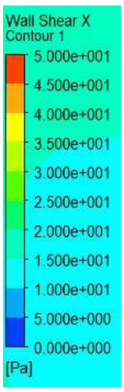
FIG 17: SHEAR CONTOUR IN Z DIRECTION ON (a) RECTANGULAR PIER AND (b) CURVED PIER

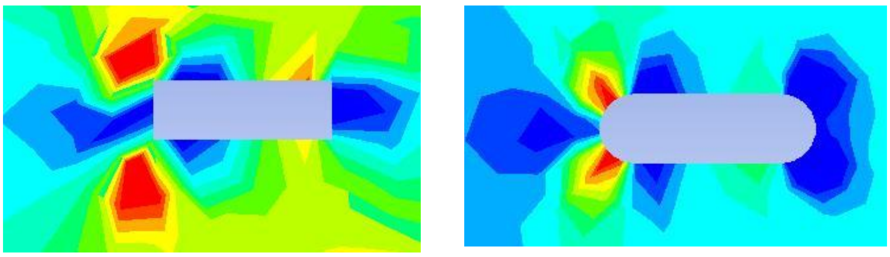




(a) **(b)**

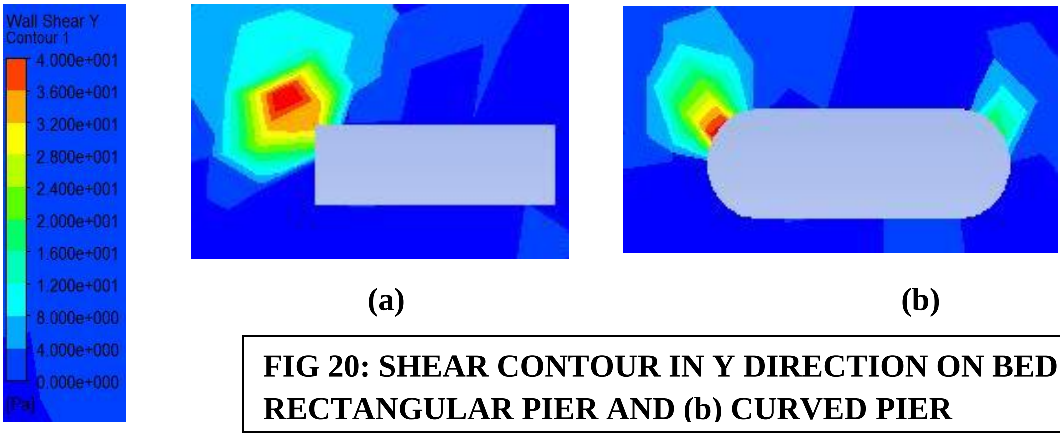
FIG 18: PRESSURE CONTOUR AT BED FOR (a) RECTANGULAR PIER AND (b) CURVED PIER





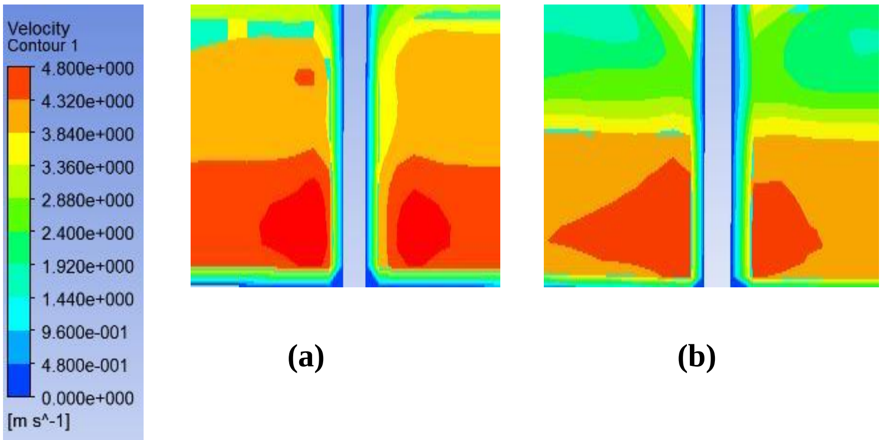
(a) **(b)**

FIG 19: SHEAR CONTOUR IN X DIRECTION ON BED FOR (a) RECTANGULAR PIER AND (b) CURVED PIER



(a) **(b)**

FIG 20: SHEAR CONTOUR IN Y DIRECTION ON BED FOR (a) RECTANGULAR PIER AND (b) CURVED PIER

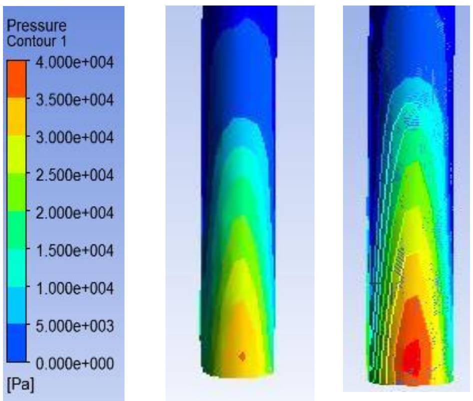


(a) **(b)**

FIG 21: VELOCITY CONTOUR AROUND THE PIER NEAR THE BED FOR (a) RECTANGULAR AND (b) CURVED PIER

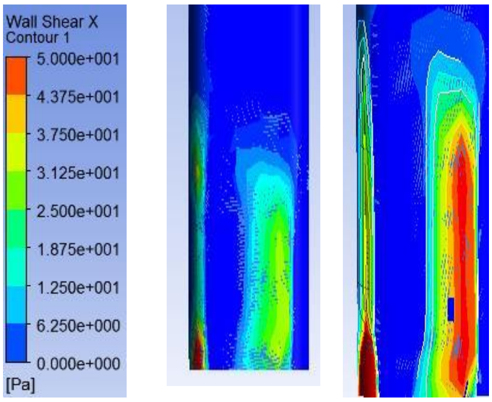
8.2. COMPARISON ON THE BASIS OF DIFFERENT VELOCITIES FOR CURVED PIERS:

Pier width =1m, Depth of flow= 3.5m, Velocity I= 4m/s, Velocity II= 6.5m/s



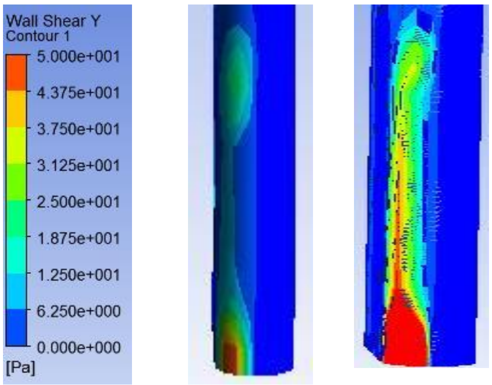
(a) **(b)**

FIG 22: PRESSURE CONTOUR FOR CURVED PIER FOR (a) VELOCITY-I AND (b) VELOCITY-II



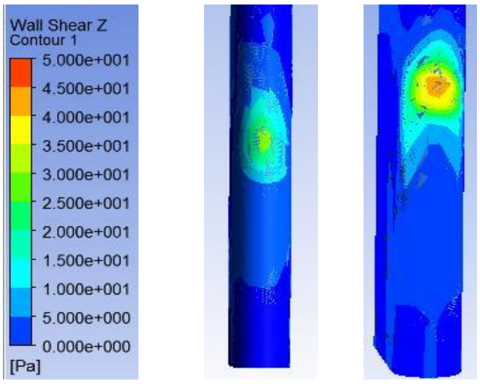
(a) (b)

FIG 23: SHEAR CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II



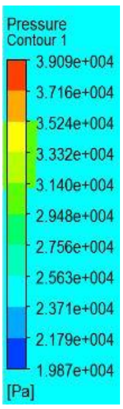
(a) (b)

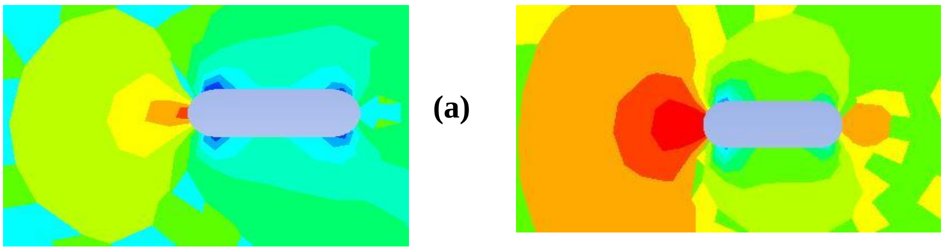
FIG 24: SHEAR CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II



(a) (b)

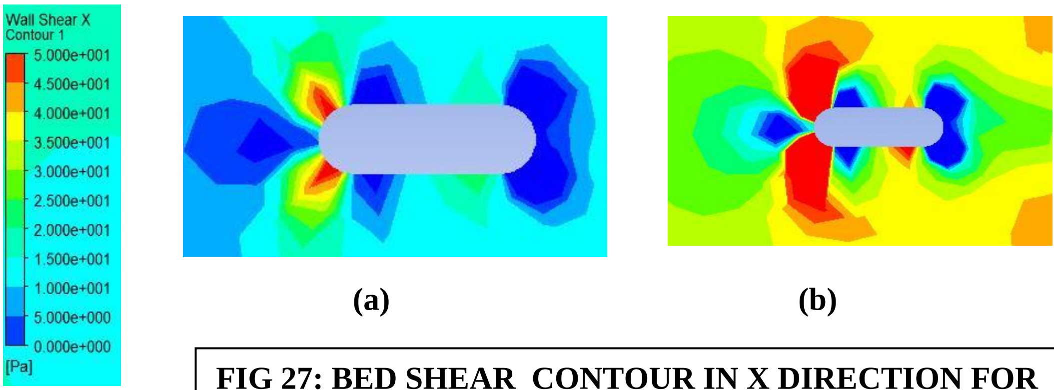
FIG 25: SHEAR CONTOUR IN Z DIRECTION FOR CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II





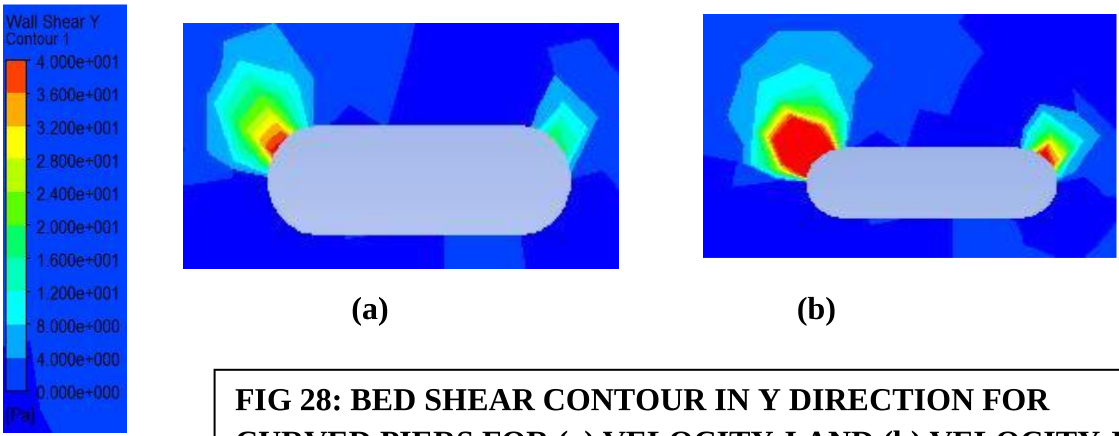
(a)

FIG 26: BED PRESSURE CONTOUR FOR CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II



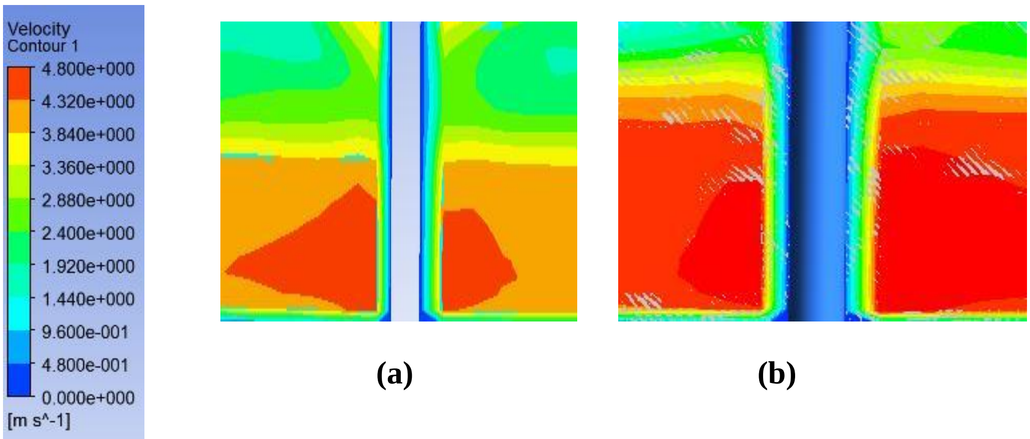
(a) (b)

FIG 27: BED SHEAR CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) VELOCITY-I AND (b)VELOCITY-II



(a) (b)

FIG 28: BED SHEAR CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II

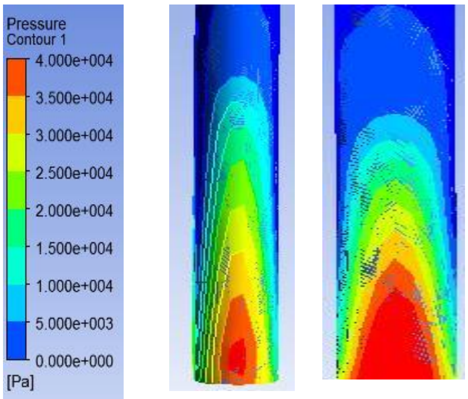


(a) (b)

FIG 29: VELOCITY CONTOUR AROUND THE CURVED PIERS FOR (a) VELOCITY-I AND (b) VELOCITY-II

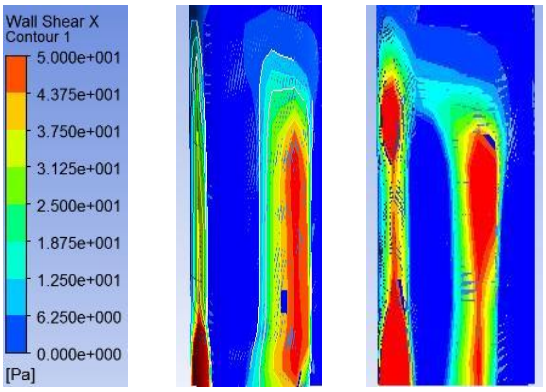
8.3. COMPARISON ON THE BASIS OF WIDTHS FOR CURVED PIERS

Velocity= 6.5m/s, Depth= 3.5m/s, Width-I = 1m and Width-II = 1.5m



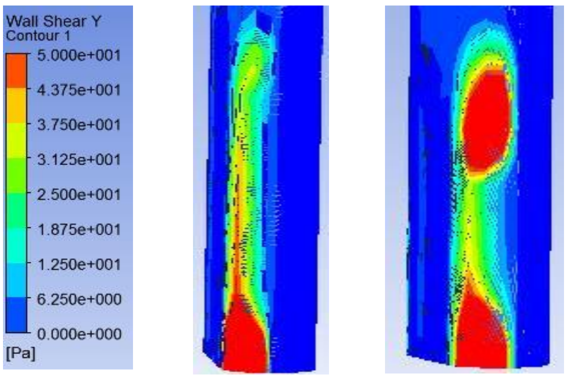
(a) (b)

FIG 30: PRESSURE CONTOUR FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II



(a) (b)

FIG 31: SHEAR CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II



(a) (b)

FIG 32: SHEAR CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II

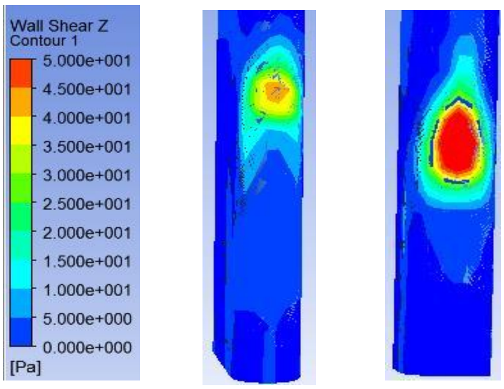
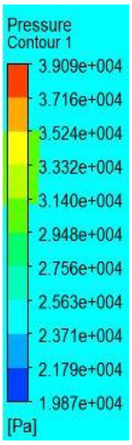
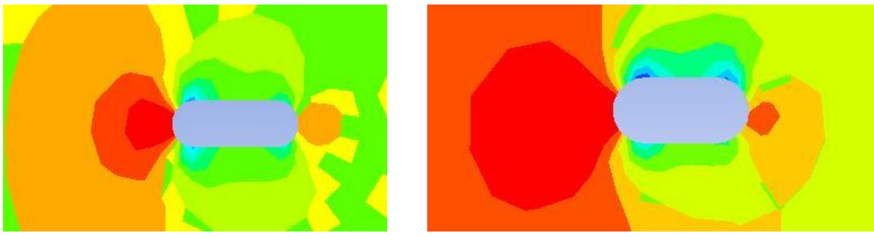


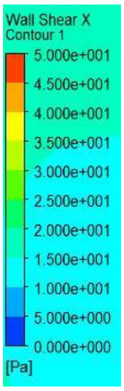
FIG 33: SHEAR CONTOUR IN Z DIRECTION FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II

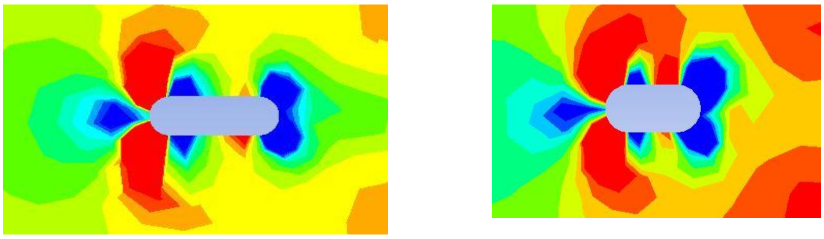




(a) (b)

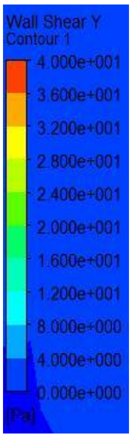
FIG 34: BED PRESSURE CONTOUR FOR CURVED PIERS FOR (a) WIDTH-I AND (b)WIDTH-II

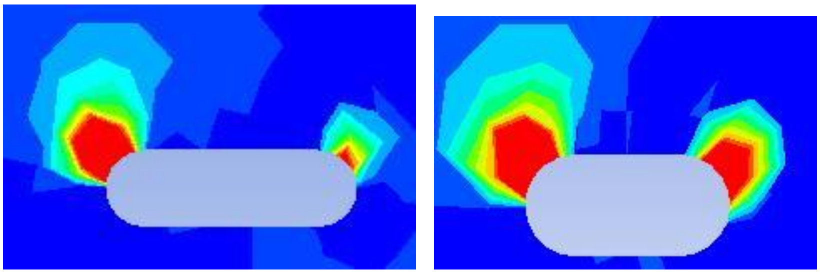




(a) (b)

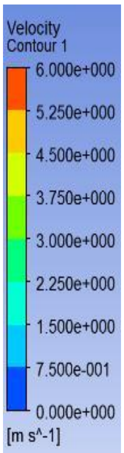
FIG 35: BED SHEAR STRESS CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II





(a) (b)

FIG 36: BED SHEAR STRESS CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II



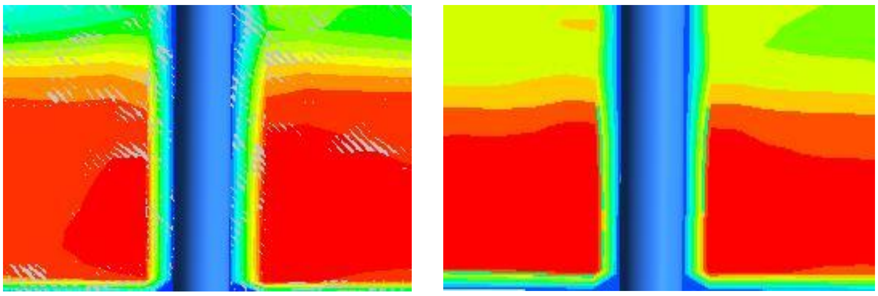
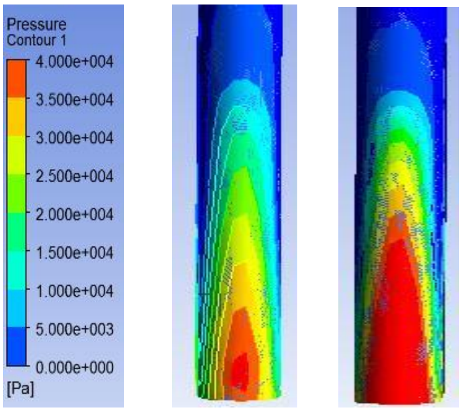


FIG 37: VELOCITY CONTOUR AROUND THE PIER NEAR THE BED FOR CURVED PIERS FOR (a) WIDTH-I AND (b) WIDTH-II

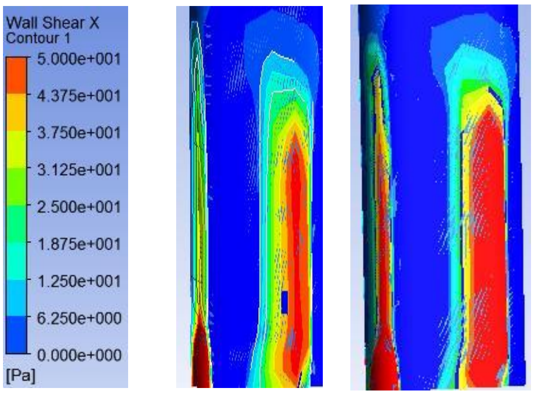
8.4. COMPARISON ON THE BASIS OF DIFFERENT DEPTHS FOR CURVED PIERS

Velocity=6.5m/s, Width of pier = 1m, Depth-I = 3.5m, Depth-II = 4.5m



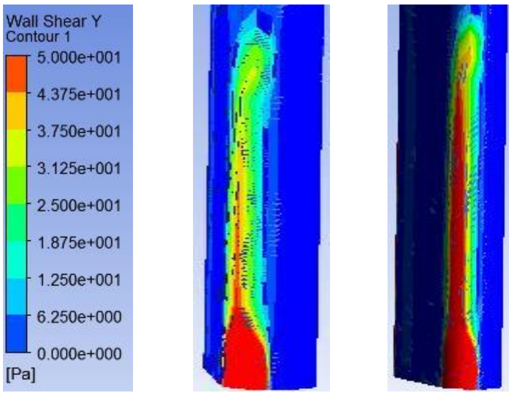
(a) (b)

FIG 38: PRESSURE CONTOUR FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II



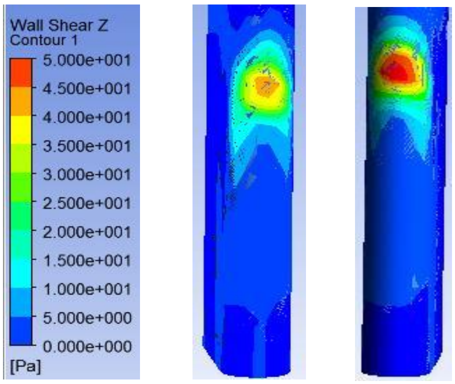
(a) (b)

FIG 39: SHEAR CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II



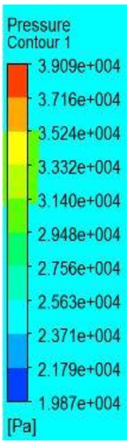
(a) (b)

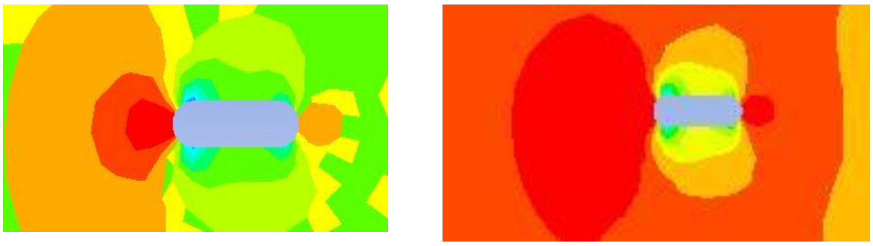
FIG 40: SHEAR CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II



(a) (b)

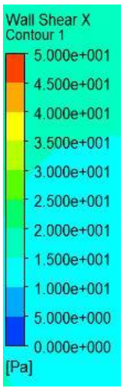
FIG 41: SHEAR CONTOUR IN Z DIRECTION FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II

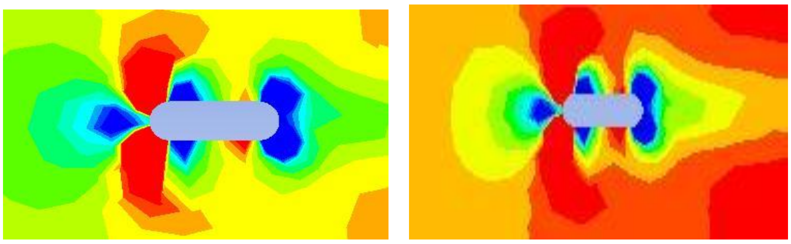




(a) (b)

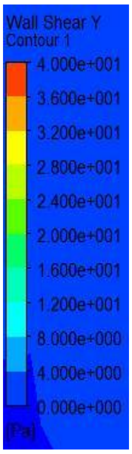
FIG 42: PRESSURE CONTOUR FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II





(a) (b)

FIG 43: BED SHEAR CONTOUR IN X DIRECTION FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II



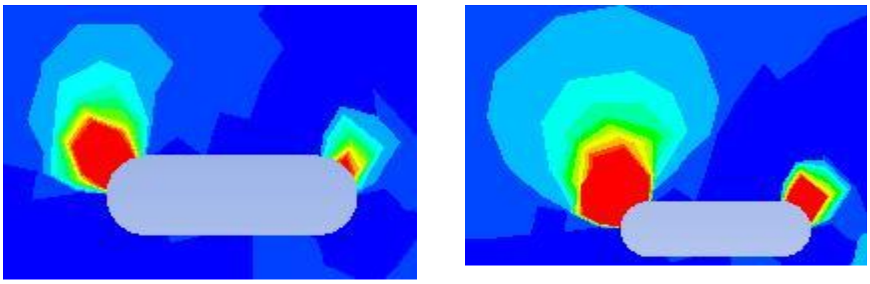
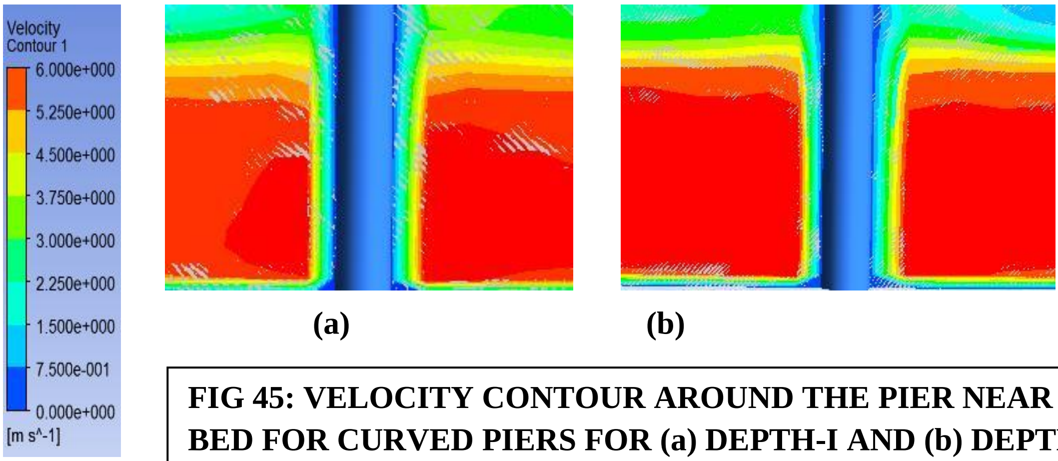


FIG 44: BED SHEAR CONTOUR IN Y DIRECTION FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II



(a) **(b)**

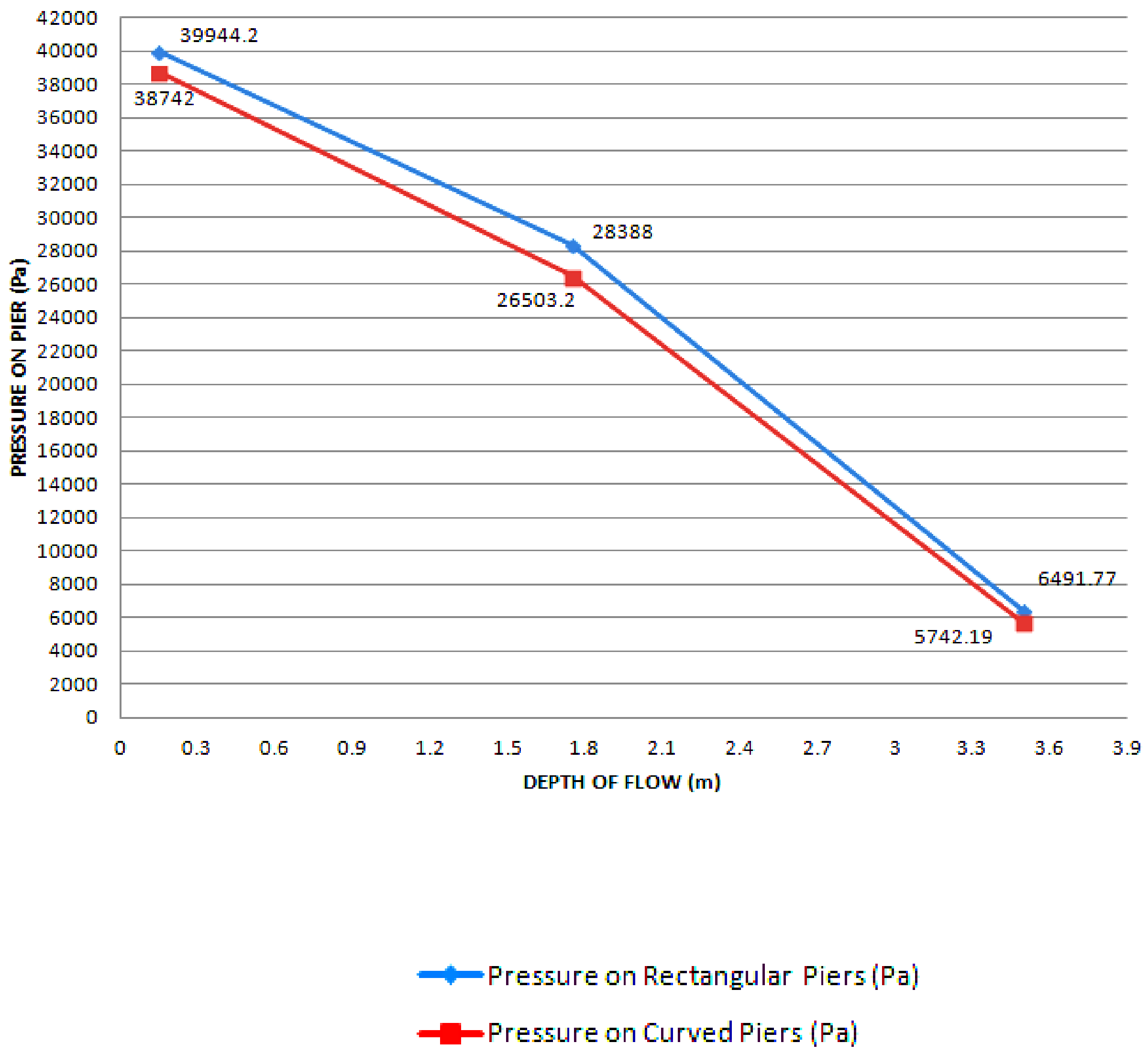
FIG 45: VELOCITY CONTOUR AROUND THE PIER NEAR THE BED FOR CURVED PIERS FOR (a) DEPTH-I AND (b) DEPTH-II

9. RESULTS AND DISCUSSIONS

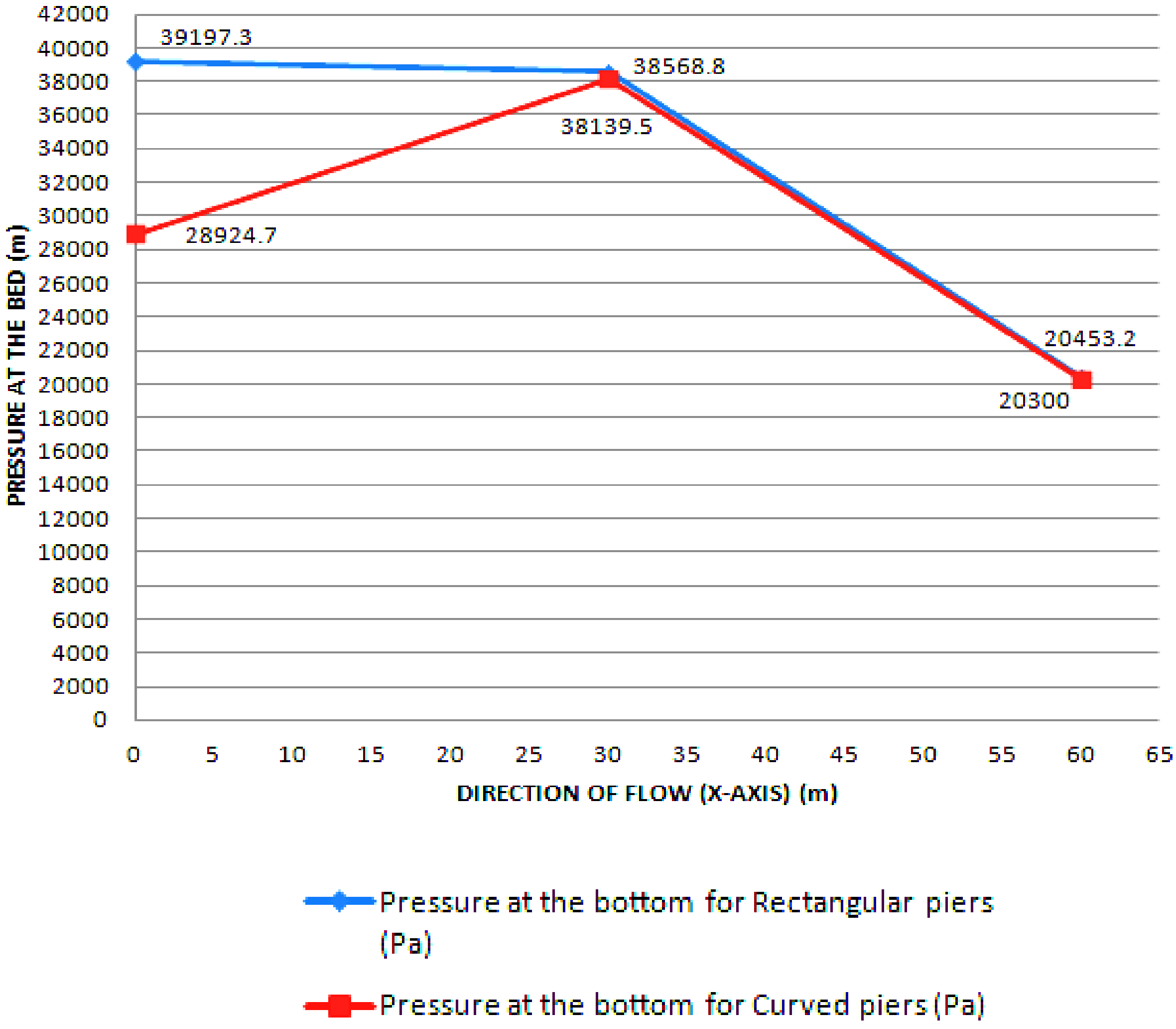
Graphs, comparing the pressure on piers along the depth and pressure at the bed along the flow (x-direction), are plotted for the following cases:

- Comparison between Rectangular and Curved (oval) piers.
- Comparison between two different velocities for Curved piers.
- Comparison between two different widths for Curved piers.
- Comparison between three different depths of flow for Curved piers.

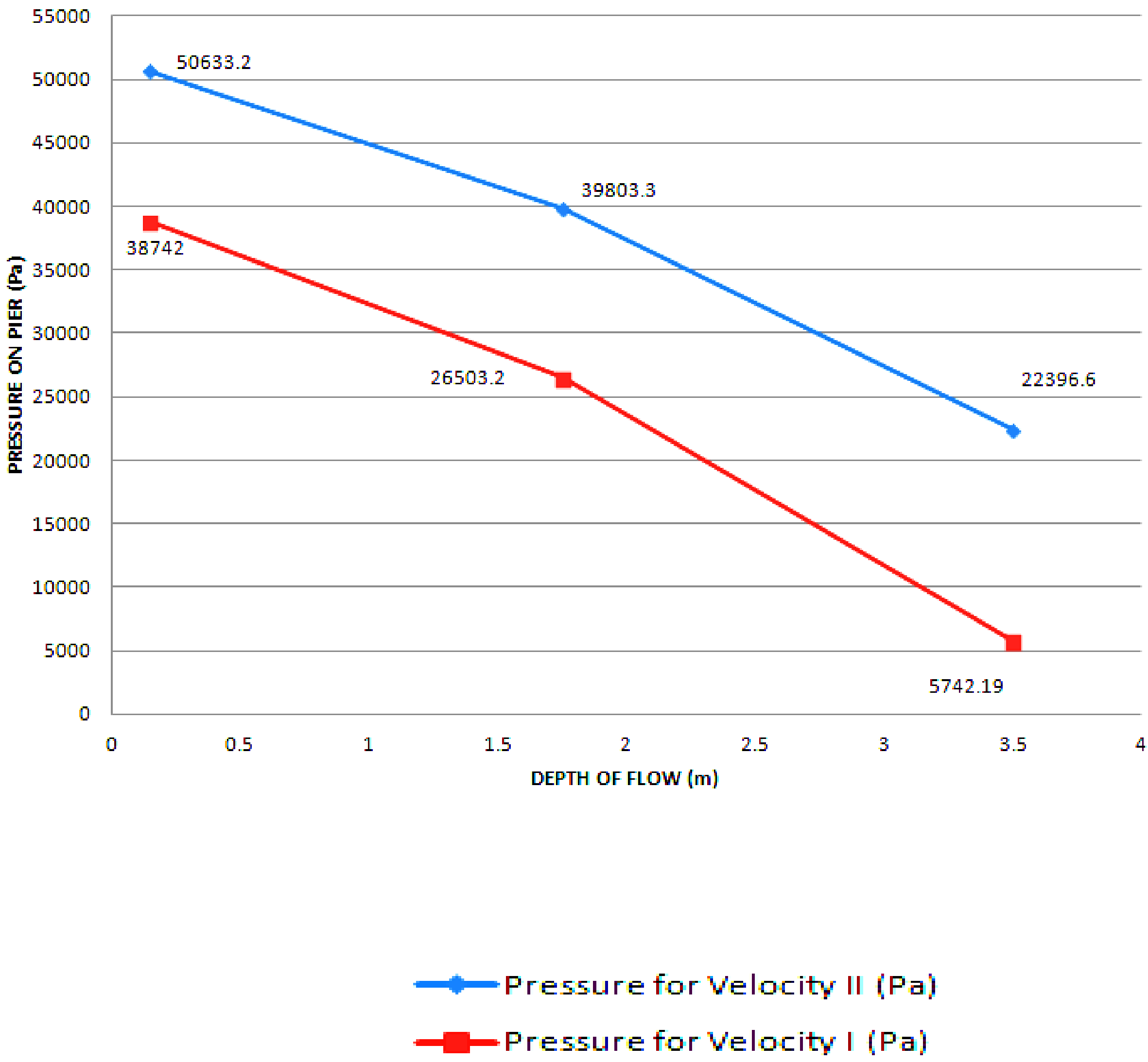
GRAPH 1: PRESSURE ON PIERS ALONG THE DEPTH OF FLOW FOR RECTANGULAR AND CURVED PIERS



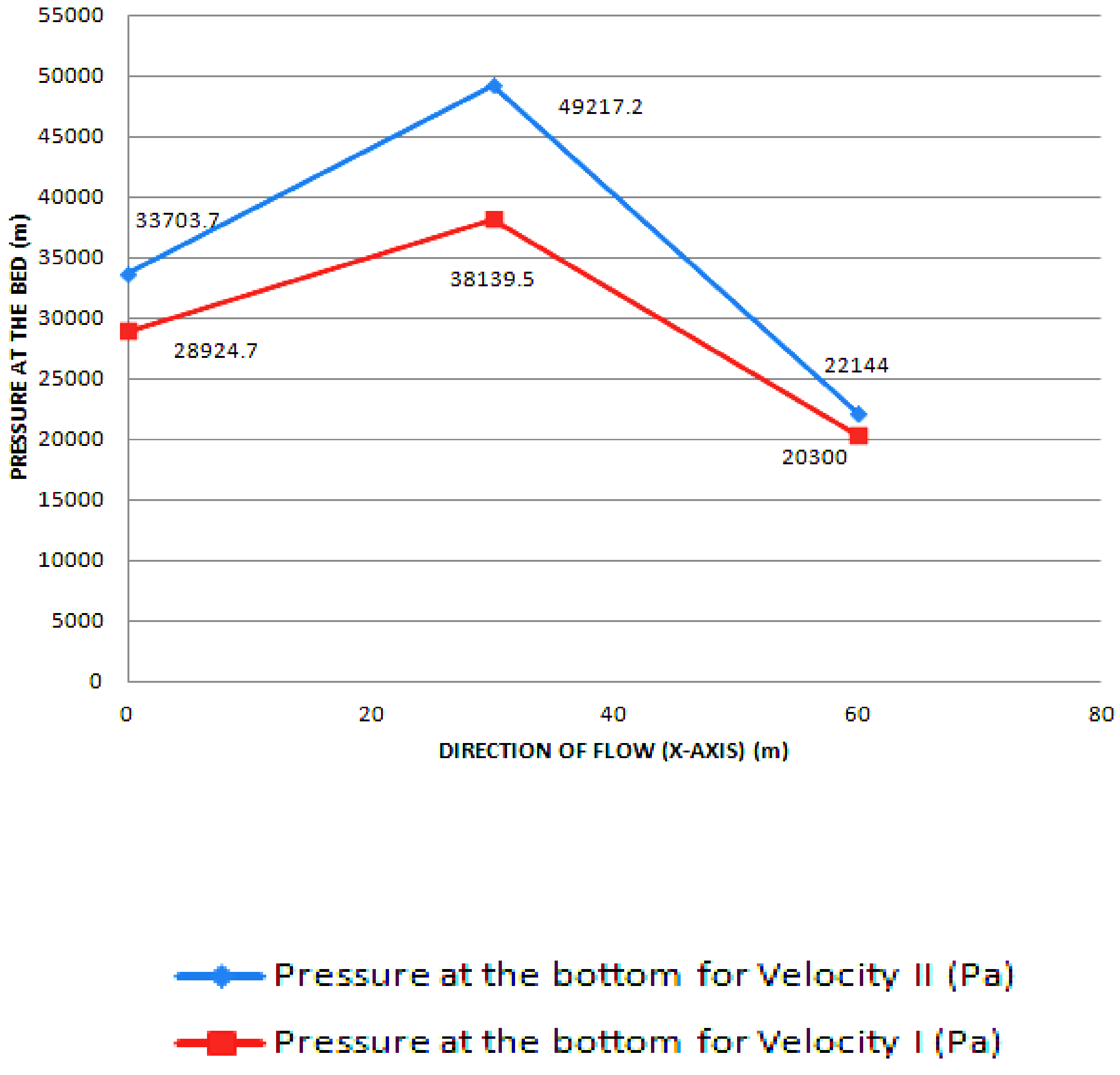
GRAPH 2: PRESSURE AT THE BED SURFACE ALONG THE DIRECTION OF THE FLOW (X-DIRECTION) FOR RECTANGULAR AND CURVED PIERS



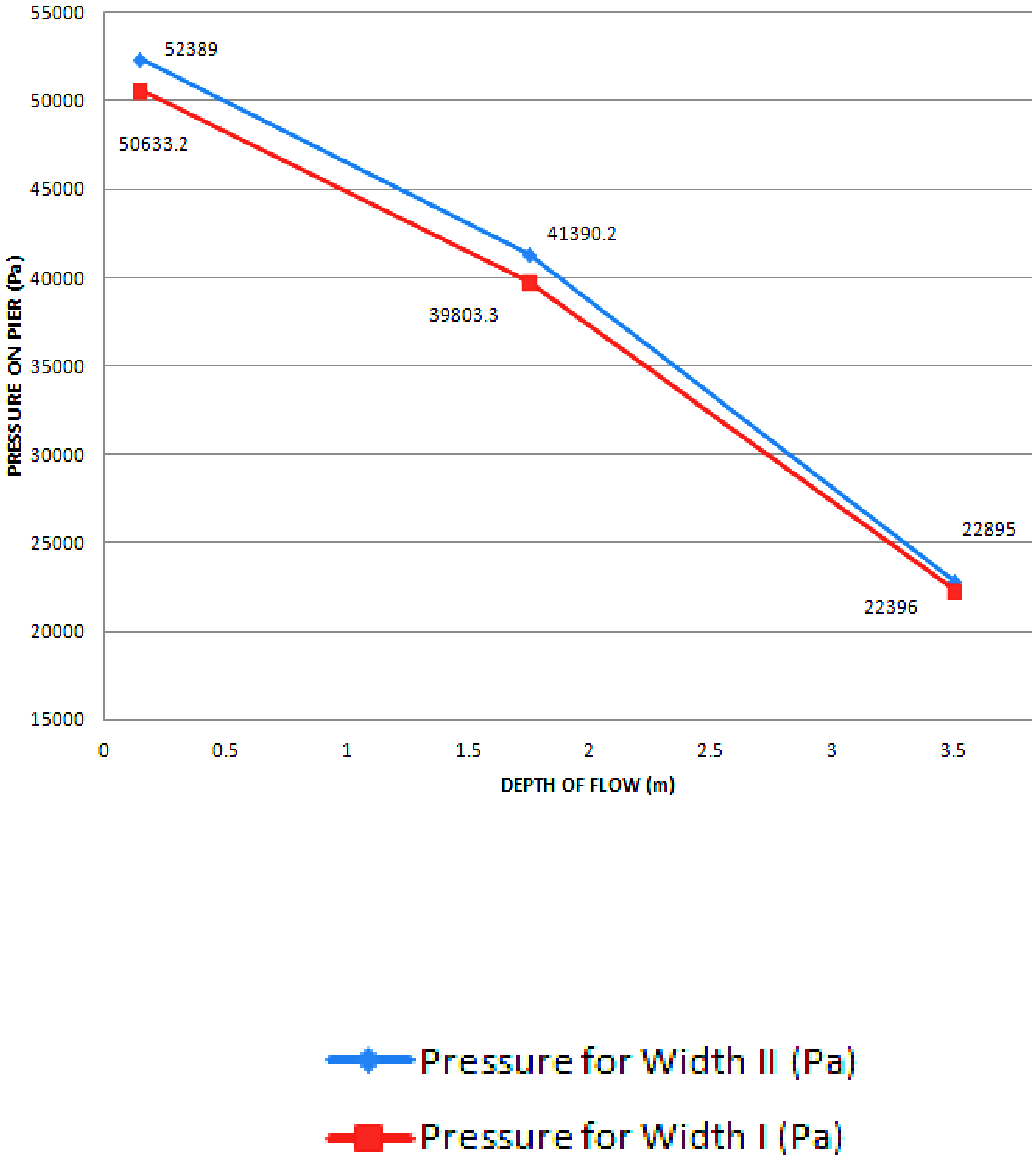
GRAPH 3: PRESSURE ON CURVED PIERS FOR VELOCITY I- 4m/s AND VELOCITY II- 6.5m/s



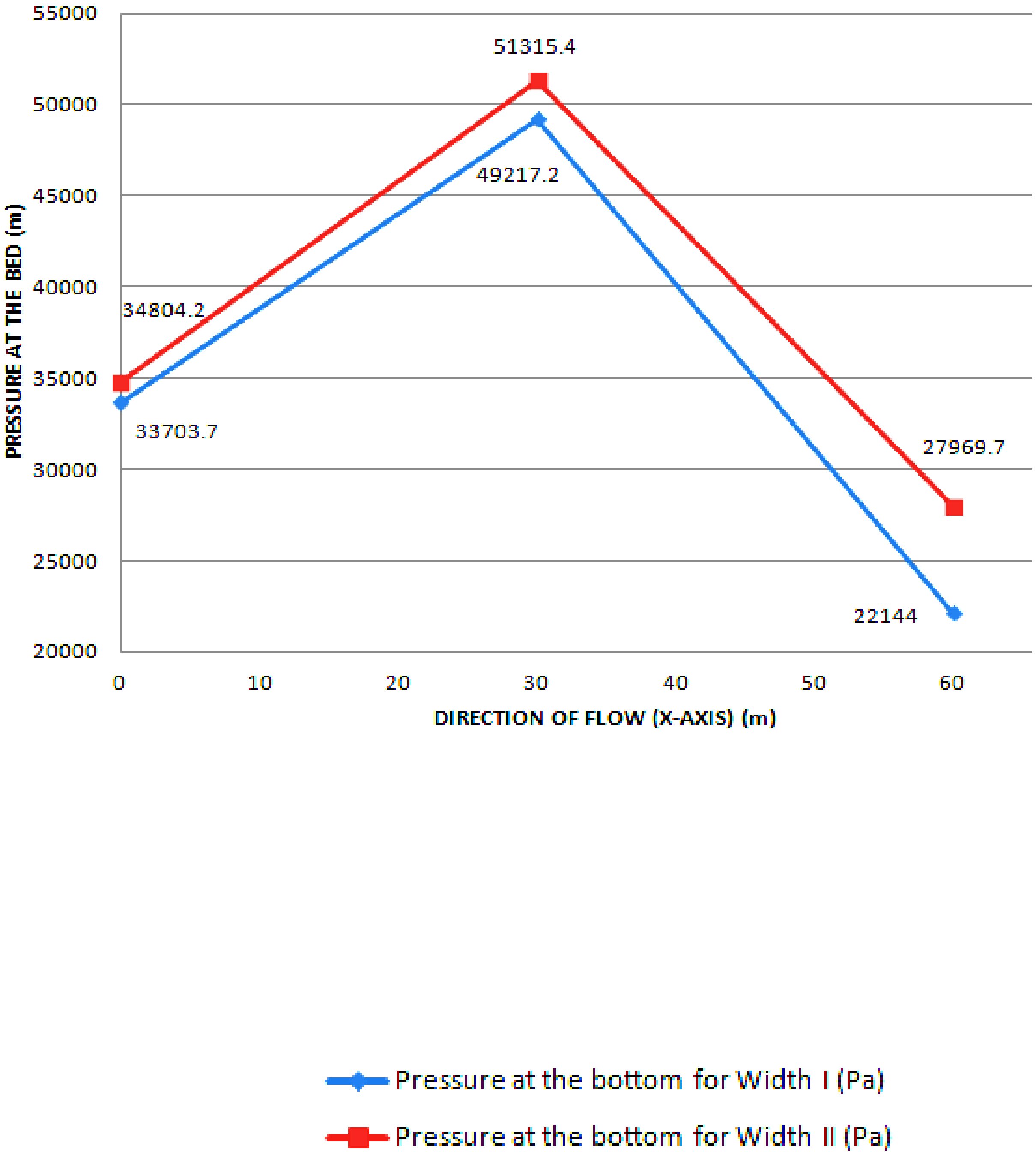
GRAPH 4: PRESSURE AT THE BOTTOM ALONG X-DIRECTION FOR CURVED PIERS FOR VELOCITY I- 4ms AND VELOCITY II- 6.5m/s



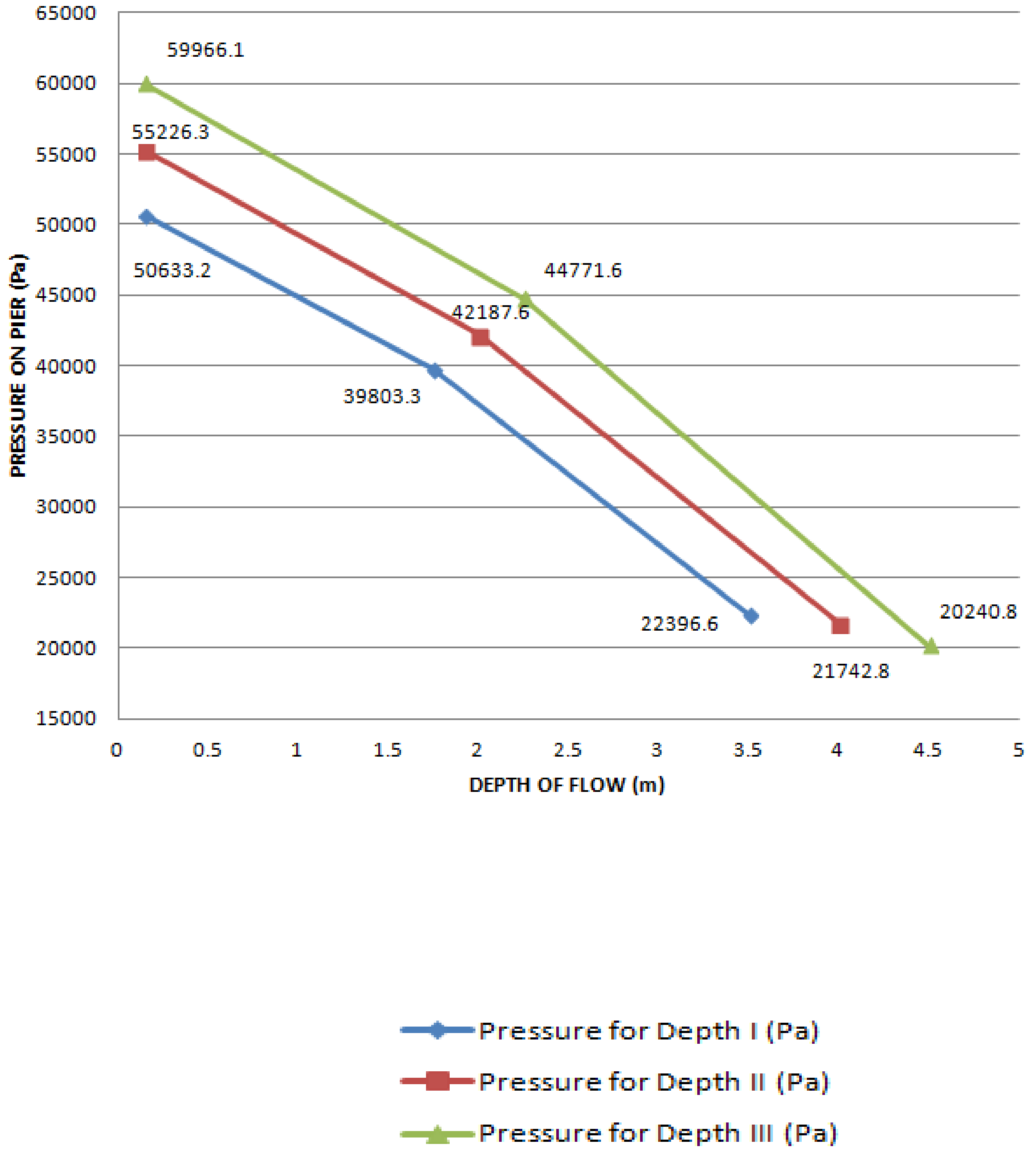
GRAPH 5: PRESSURE ON CURVED PIERS ALONG THE DEPTH WITH WIDTH I- 1m AND WIDTH II- 1.5m



GRAPH 6: PRESSURE AT THE BED ALONG THE X-DIRECTION FOR CURVED PIERS OF WIDTH I- 1m AND WIDTH II- 1.5m



GRAPH 7: PRESSURE ON CURVED PIERS FOR DEPTH I-3.5m, DEPTH II- 4m, DEPTH III- 4.5 m



GRAPH 8: PRESSURE AT THE BED ALONG X-DIRECTION FOR CURVED PIERS FOR DEPTH I-3.5m, DEPTH II- 4m, DEPTH III- 4.5 m

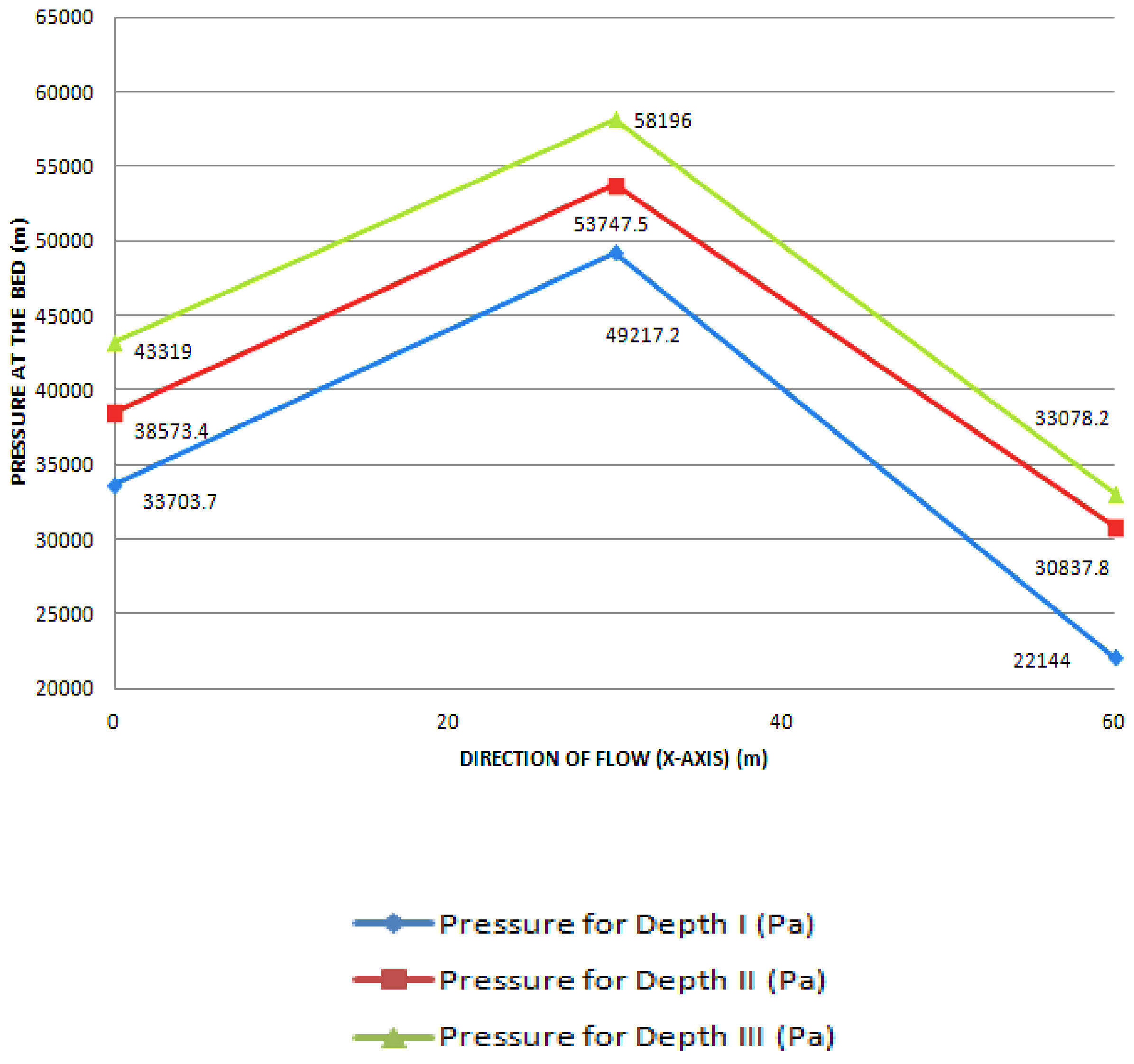


FIGURE 46: VELOCITY VECTORS IN CROSS- SECTION

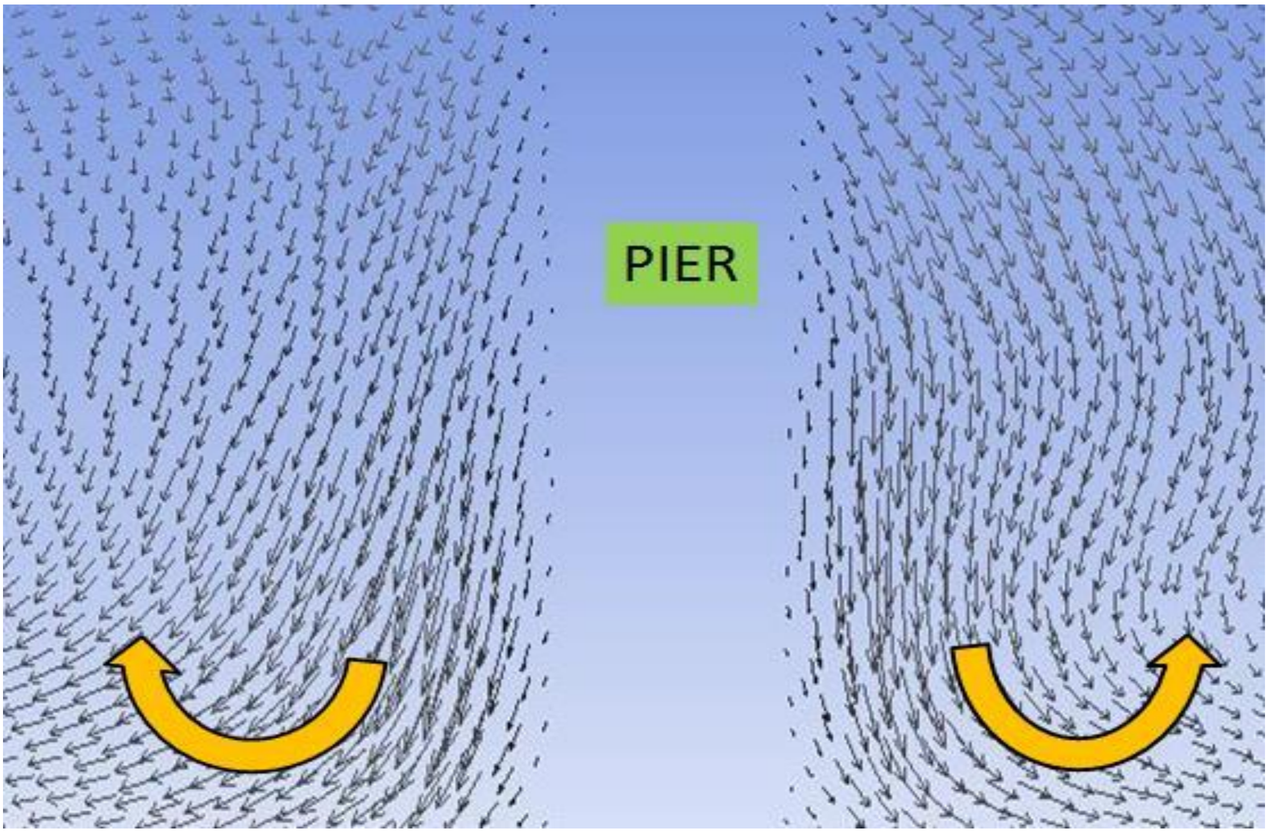
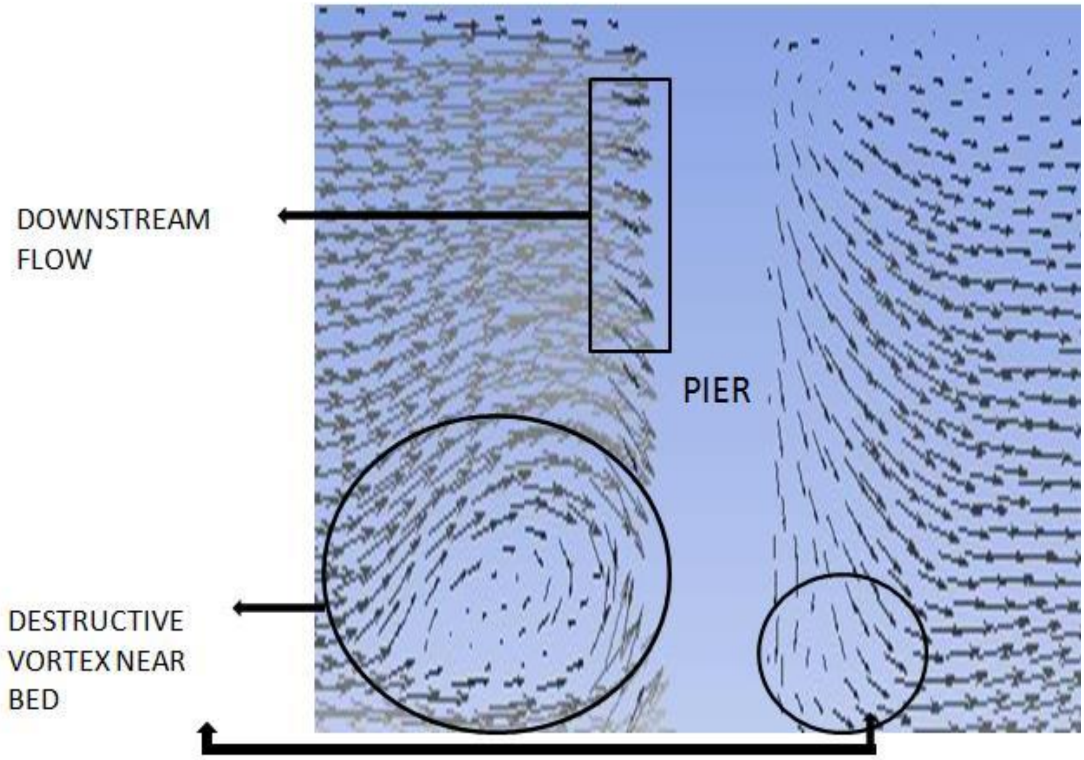


FIGURE 47: VELOCITY VECTORS IN LATERAL- SECTION



- The above graphs show that the pressure on the upstream face of the piers and at the bed surface near the piers increases with the increase in width of the piers; increase in the velocity of the flow or the discharge; and with the increase in the depth of the flow.
- It is also noted that the pressure on the piers and at the bed surface is more in case of a sharp edged pier as compared to the curved nose pier.
- From the graphs, it is seen that the pressure at the piers is maximum near the bed and goes on decreasing as we move bed towards the free surface i.e. along the positive z-direction.
- It is also seen that the pressure at the bed is maximum near the upstream nose tip of the pier. The pressure increases from its value at the inlet to the maximum value, which it attains near the piers; and then its value again decreases.
- From the Velocity vectors, it is noted that due to the presence of a pier in the path of flowing water, the water has to change its path. When the flowing water hits the pier, then because of the effect of gravity, its motion becomes downward. That flow is termed as Downstream flow. The intensity of the downward moving vectors keeps on increasing till they reach the bed surface. This leads to the formation of vortices in front of the piers as well as at the back of them. These vortexes and eddies trigger the shear stress. This is the reason that the shear stress as well as the pressure near the piers is maximum. The change in the direction of the flow increase the velocity magnitude and hence, its ability to scour.
- Greater the stresses at the bed, larger can be the local scouring.

The numerical results for each kind of comparison are as follows:

For the same velocity of 4m/s, same channel dimensions, same pier width of 1m, same flow depth of 3.5m; when the shape is changed from Rectangular to Curved (Oval), the maximum pressure on piers, maximum shear on piers, maximum pressure on the bed, maximum shear on the bed and torque acting on the piers decreases by the following percentage : TABLE: 8

VARIABLES	PERCENTAGE DECREASE
Maximum pressure on piers	5.23
Maximum Shear on piers in x- direction	156
Maximum Shear on piers in y- direction	0.27
Maximum Shear on piers in z- direction	18.15
Maximum pressure at the bed near the pier nose	0.648
Maximum Shear on the bed in x- direction	10.511

Maximum Shear on the bed in y- direction	60.54
Torque acting on the piers	156.06

Now, as it is known that the curved shape is a better shape. The comparisons are done for the curved shape for different widths, velocities and depths.

For same channel dimensions; same depth of flow of 3.5m; same velocity of 6.5 m/s; when the width of the piers is increased by 50% from 1 to 1.5m, all the previously mentioned variables increase by the following percentage : TABLE 9:

VARIABLES	PERCENTAGE INCREASE
Maximum pressure on piers	0.425
Maximum Shear on piers in x- direction	8.80
Maximum Shear on piers in y- direction	6.41
Maximum Shear on piers in z- direction	1.94
Maximum pressure at the bed near the pier nose	6.15
Maximum Shear on the bed in x- direction	14.47
Maximum Shear on the bed in y- direction	15.54
Torque acting on the piers	11.47

For same channel dimensions; same depth of flow of 3.5m; same width of piers of 1m; when the velocity of the flow is increased by 62.5% from 4 to 6.5 m/s , all the previously mentioned variables increase by the following percentage : TABLE 10:

VARIABLES	PERCENTAGE INCREASE
Maximum pressure on piers	32.5
Maximum Shear on piers in x- direction	207.79
Maximum Shear on piers in y- direction	28.54
Maximum Shear on piers in z- direction	35.62

Maximum pressure at the bed near the pier nose	27.9
Maximum Shear on the bed in x- direction	16.48
Maximum Shear on the bed in y- direction	55.76
Torque acting on the piers	612.93

For same channel dimensions; same width of piers of 1m; same velocity of 6.5m/s; when the depth of the flow is increased by 16.67% from 3.5 to 4m , all the previously mentioned variables increase by the following percentage : TABLE 11:

VARIABLES	PERCENTAGE INCREASE
Maximum pressure on piers	10.06
Maximum Shear on piers in x- direction	31.48
Maximum Shear on piers in y- direction	20.68
Maximum Shear on piers in z- direction	1.675
Maximum pressure at the bed near the pier nose	8.907
Maximum Shear on the bed in x- direction	18.75
Maximum Shear on the bed in y- direction	30.550
Torque acting on the piers	33.907

For same channel dimensions; same width of piers of 1m; same velocity of 6.5m/s; when the depth of the flow is increased by 50% from 3.5 to 4.5m , all the previously mentioned variables increase by the following percentage : TABLE 12:

VARIABLES	PERCENTAGE INCREASE
Maximum pressure on piers	20.09
Maximum Shear on piers in x- direction	45.914
Maximum Shear on piers in y- direction	33.83

Maximum Shear on piers in z- direction	4.274
Maximum pressure at the bed near the pier nose	17.61
Maximum Shear on the bed in x- direction	49.93
Maximum Shear on the bed in y- direction	44.51
Torque acting on the piers	71.32

9.1. SCOUR DEPTH FOR ALL THE MODELS

The scour depth for each case has been calculated using MARYLAND SHA BRIDGE SCOUR PROGRAM which is based on the Hydrologic and Hydraulic Design by MDSHA AND HEC-18, FWHA.

INPUT PARAMETERS	
LENGTH OF PIER	3m
FLOW ATTACK ANGLE	0˚
MEDIAN GRAIN SIZE D50 (mm)	0.1
MEDIAN GRAIN SIZE D95 (mm)	0.33
PIER STEM NOSE SHAPE CORRECTION FACTOR (K1)	1.1 (FOR SHARP NOSE PIER) 1 (FOR ROUND NOSE PIER)
OVERRIDE ANGLE OF ATTACK CORRECTION FACTOR (K2)	1
STREAM BED CONDITION CORRECTION FACTOR (K3)	1.1 (CLEAR WATER CONDITION)

OUTPUT PARAMETERS:

LOCAL SCOUR DEPTH	
CASE-I (Comparison of shapes of pier)	
FROUDE NO. = 0.6825	
RECTANGULAR PIER	1.664 m
CURVED PIER	1.413 m

LOCAL SCOUR DEPTH	
CASE-2 (Comparison for curved piers on basis of VELOCITIES)	
VELOCITY-I = 4m/s FROUDE NO. = 0.6825	1.413 m
VELOCITY-II = 6.5m/s FROUDE NO. = 1.109	2.107 m

LOCAL SCOUR DEPTH	
CASE-III (Comparison OF Curved piers on the basis of WIDTHS)	
WIDTH I = 1m	2.107 m
WIDTH II= 1.5m	2.67 m

LOCAL SCOUR DEPTH	
CASE-IV (Comparison OF Curved piers on the basis of DEPTHS)	
DEPTH I= 3.5m	2.107 m
DEPTH II= 4m	2.134 m
DEPTH III= 4.5m	2.159 m

10. CONCLUSIONS

For the same channel section and other flow conditions :

1. The pressure on the piers, pressure at the bed surface of the channel; shear stresses acting on the piers and at the bed surface near the piers; and torque acting on the piers is **more for a sharp nose pier as compared to a curved nose pier**.
2. The pressure on the piers, pressure at the bed surface of the channel; shear stresses acting on the piers and at the bed surface near the piers; and torque acting on the piers increases with the **increases with the increasing Froude No**.
3. The pressure on the piers, pressure at the bed surface of the channel; shear stresses acting on the piers and at the bed surface near the piers; and torque acting on the piers **increases with the increase in the width of the pier**.
4. The pressure on the piers, pressure at the bed surface of the channel; shear stresses acting on the piers and at the bed surface near the piers; and torque acting on the piers **increases with the increase in the depth of the flow** in the channel.
5. It is noted that the shear on the piers increases more in the x-direction and the shear on the bed surface increases more in the y- direction.
6. The reason of this difference in the stresses and pressure for different models is that the flow when obstructed by the piers, moves in the downward direction due to the gravity. For each kind of model, it will be different in terms of magnitude. The flow when hits the pier, changes its path and velocity (both speed and direction). This velocity keeps on increasing until it reaches the bed surface. This downward flow near the upstream side of piers is called Downstream flow, which gives rise to the formation of vortices near the channel bed. The upstream side vortices are horse vortex and downstream side vortices are tail vortex. These **vortices have high velocities and hence they trigger the shear stresses and pressure at the bed. This all leads to the local scouring.**
7. The prediction made that the shear stress at the bed is directly proportional to the amount of local scour i.e. greater the stress, greater is the local scouring through ANSYS FLUENT is verified by the other software MARYLAND SHA BRIDGE SCOUR PROGRAM by giving the exact values of local scour depth for each model.

The above study shows, bridge piers of sharp edges, with higher width, with higher velocity and with higher depth, are more prone to collapsing, twisting and damage due to high pressures, shear stresses and torques. Also, their channel beds are more prone to scouring due to high pressure and shear stresses. This can make the piers unstable as the soil around them gets scooped out making the piers exposed to the harsh effects of flow.

8. The ANSYS FLUENT software is extremely useful in solving the problems that are too difficult and time consuming to solve physically. It just needs accuracy in case of making the geometry, the domain and providing the boundary conditions.

FUTURE SCOPE OF THE WORK

- In future, a three phase flow can be used comprising of all the three states (liquid, gas and solid).

- The channel contractions can be taken into consideration.
- Instead of a straight channel bed, proper slope in the channel bed can be given.
- The addition of debris and ice can be considered in the flow.
- Other softwares like OpenFOAM software combined with HEC-RAS can be used to find the total scour depth around the piers in addition to the local scour.

11. REFERENCES

1. Sabita Madhvi Singh & P. R. Maiti: “Flow Field and Scouring around Cylindrical Structure in Channel Bed”; Second Intl. Conference on Advances In Civil, Structural and Environmental Engineering- ACSEE 2014.

2. Mohammad Vaghef , Hamed Dashtpeyma , Arash Adib , Javad Roohian: “Numerical Analysis of Flow Pattern around square Bridge Piers by New Ansys Software”; 6 th National Congress on Civil Engineering, April 26-27, 2011, Semnan University, Semnan, Iran.

3. Marieh Rajaie , Mohammad Reza Pirestani and Seyed Hossein Ghoreishi Najaf Abadi: “Effects of Oval and Circular Piers on Velocity and Shear Stress Changes by using Open FOAM Software”; Special issue on Current World Environment, Volume 10, April 2015.

4. I. Mistrová, D. Picka [3] : “Determine of velocity field with PIV and CFD during the flow around of bridge piers”; EPJ Web of Conferences- 45, EDP Sciences, 2013.

5. Kamil H. M. Ali, Othman Karim [4] : “Simulation of flow around piers”; Journal of Hydraulic Research 40(2):161-174, March 2002.

6. Kassem Salah El-Alfy [5] : “Backwater Rise Due To Flow Constriction By Bridge Piers”; Thirteenth International Water Technology Conference, IWTC 13,Egypt, 2009.

7. Wenrui Huang, Qiping Yang, Hong Xiao [6] : “CFD modeling of scale effects on turbulence flow and scour around bridge piers”; 18th, January 2008.

8. Sreedhara B M, Sanooj A, Manu , S Mandal [7] : “Simulation of Local Scour around Circular and Round Nosed Bridge Pier using REEF3D”; IJIRSET, volume 5, special issue 9, May 2016.

9. Mete Koken and George Constantinescu [8] : “An investigation of the flow and scour mechanisms around isolated spur dikes in a shallow open channel: 1. Conditions corresponding to the initiation of the erosion and deposition process”; 5th August, 2008.