CFD STUDIES OF LOCAL SCOURING AROUND BRIDGE PIER

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR

THE AWARD OF THE DEGREE

OF

MASTER OF TECHNOLOGY

IN

HYDRAULICS & WATER RESOURCE ENGINEERING

Submitted By:

KAUSTUBH CHAUHAN

2K17/HFE/10

Under the supervision of

Dr. BHARAT JHAMNANI

CIVIL ENGINEERING DEPARTMENT

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

JUNE, 2019

DEPARTMENT OF CIVIL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, Kaustubh Chauhan, 2K17/HFE/10 of M.Tech (Hydraulics & Water Resource Engineering), hereby declare that the project dissertation titled **"CFD STUDIES OF LOCAL SCOURING AROUND BRIDGE PIER"** which is submitted by me to the Department of Civil Engineering, Delhi Technological University, Delhi in fulfillment of the Minor-II Project of Master Of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associate ship, Fellowship or other similar title or recognition.

Place: Delhi **KAUSTUBH CHAUHAN**
Date: (2K17/HFE/10)

DEPARTMENT OF CIVIL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the project dissertation titled **" CFD STUDIES OF LOCAL SCOURING AROUND BRIDGE PIER"** which is submitted by Kaustubh Chauhan, 2K17/HFE/10 of M.Tech (Hydraulics & Water Resource Engineering) Delhi Technological University, Delhi in fulfillment of the Minor-II Of Master Of Technology, is a record of the project work carried out by the him under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi Dr. BHARAT JHAMNANI

Date: **(SUPERVISOR)**

ACKNOWLEDGEMENT

I take this opportunity to express my sincere regards and deep gratitude to Dr. Bharat Jhamnani Asst. Professsor, Department of Civil Engineering, DTU for his consistent guidance, monitoring and constant encouragement throughout the course of this project work. The help, blessing and guidance by him shall carry me a long way in life on which I am going to embark.

Also, I express my gratitude to Dr. Nirender Dev (Head of Department, Department Civil Engineering, DTU) for extending his support and guidance.

Professors and faculties of the Civil Engineering Department, DTU have always been very supportive and cooperative. They have been always present for their kind opinions and suggestions regarding this project work and therefore I am deeply obliged to them.

Last but not the least, I would like to thank my family and friends who encouraged me to bring this work to a successful close.

KAUSTUBH CHAUHAN
Roll No. 2K17/HFE/10
Department of Civil Engineering
Delhi Technological University

ABSTRACT

The threat of local scour around bridge piers has been in research for many years. According to the various studies, local scour around the bridge pier is the prime cause for most of the bridge failures. The main objective of the present study was to investigate the flow behavior and the scour phenomenon around the bridge piers of various shapes namely Circular, Elliptical, Square and Streamlined. Local scouring depends on various factors like depth of flow, upstream flow conditions, pier shape and dimensions. Here, we have taken only pier shape as the primary factor and kept other factors constant. The numerical simulations were even carried out using CFD- Fluent, eulerian, k–epsilon turbulence model, to elaborate the physics behind the scour formation. CFD simulation tool can be used for wide understanding of the flow behavior around the bridge piers even without physical model studies because it saves time and money as compared to experimental studies. Three dimensional simulation of flow behavior around four pier shapes indicates that the streamlined pier is the most efficient pier to use as it allows the flow to pass smoothly around it creating less obstruction to the flow and hence creating less chances of local scouring near the pier toe.

CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 SCOURING AROUND BRIDGE PIERS

Local scouring around bridge foundations can lead to the partial failure or the collapse of bridge piers. In a steady current flow, if a vertical pier is placed on the bed it introduces changes to the flow pattern that interfere with the riverbed. Characteristic flow structures are the horse shoe vortex (formed upstream of the pier) and trailing vortex flow pattern (usually in the form of vortex shedding) that is formed at the downstream side of the pier (Figure 1). A down flow also exists due to the presence of flow deceleration upstream of the pier. These changes in the flow behavior around the piers generally increases the probability of sediment transport, resulting in local scour around the pier.

Scour is the engineering term for the erosion caused by water on the soil nearby any hydraulic structure. Scouring phenomenon is a very complex problem as many parameters such as flow depth, velocity, shape of pier, size of pier, types of bed materials etc. control the scouring. Scouring around piers occurs due to the formation of horse shoe vortex forming in front of the pier. Due to this phenomenon, the bed material dislocates which results in scour. Excessive local scouring can happen due to repeated flood events. The boundary layer present near the pier toe undergoes a 3D separation. The separated shear layer swirls up along the pier wall forming a vortex flow upstream of the bridge pier which is carried far away downstream by the flowing water. The shear stress acting on the river bed is consequently increased and exceeds the threshold value of bed shear stress due to the generation of the horseshoe vortex and the associated down flow around the piers. Therefore, the sediment transport capacity of the flow increases drastically. As a consequence, a deep scour hole is starts forming around the bridge pier. The formation of the scour hole changes the flow pattern causing a reduction in shear stress by the flow and a consequent reduction in the sediment transport capacity. However, the stronger flow increases the scouring rate as compared to weaker flow field. The void or depression formed around the pier, as sediment is carried away from the river bed is called as scour hole.

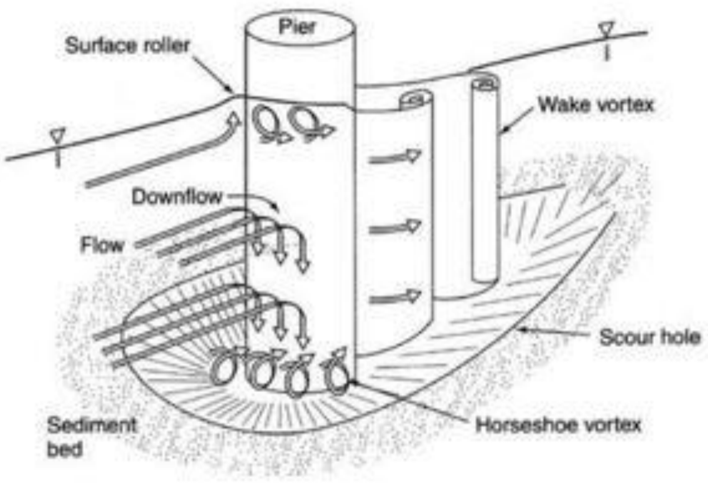

Figure 1.1. Geometry of Circular Pier

The strong vortex motion formed in the presence of the pier, displaces bed sediments near pier toe. The scour hole around the bridge pier gradually deepens as the down flow swirls up and interacts with the upstream flow forming a complex vortex system.

Bridge piers with cylindrical shape (circular cross section) are the most general pier. As the flow goes around a pier, bed sediments from the flat bed geometry are often eroded. These eroded sediments again settle down slowly around the pier. The scouring rate is higher than the sediment settling rate at the beginning and at equilibrium both erosion and settling of sediments reaches a steady rate. The initially higher rate of scouring results in equilibrium fixed scour hole around a pier.

1.2 TYPES OF SCOURING

a) General Scouring- It depend primarily on the sediment transport at the location and may vary considerably from location to location. It occurs irrespective of the existence of the structure.

b) Local Scouring- It is typically due to presence of the structures and especially piles or gravity type foundations. It occurs due to any local obstruction to the flow in ocean.

c) Global or Dishphan Scour- It is basically shallow wide depressions under and around individual hydraulics structures.

d) Overall Sea bed movement- It is generally the soil eroded by the flowing water resulting Erosion, Deposition and Bed form movement fprming ripples on the riverbed.

This numerical modelling research considers the bridge hydraulics problem where two-dimensional (2-D) open-channel flow approaches a fixed scour hole and interacts with a circular and oblong pier. Flow approaching a bridge pier has tendency to move downward towards the channel bed which has implication in removing sediments from the channel bed. Excessive sediment removal or scouring is alarming for the safety of the bridge pier, which can eventually lead to uprooting of pier from the channel bed and yield bridge failure. So accuracy in the ability to predict scouring around a bridge pier brings more confidence in safe pier-foundation design.

1.3 ANSYS FLUENT

ANSYS is an advanced software which enables one to do all the engineering related simulations of problems related to fluid dynamics, chemical engineering, environmental engineering, hydrodynamics, metaphysics, electromagnetic, structural mechanics and so on. In this project we have particularly used fluent because we are concerned with the system's fluid dynamics. Computational fluid dynamics (CFD) is a mathematical tool based on computer programming. The growing interest in the field of CFD based simulations has been widely used by engineers in all those areas where experimental or numerical analysis becomes cumbersome. Determination of movement of fluid in detail by solving a system set of nonlinear governing equations after the use of specified boundary conditions over the ambit of interest is the basic principle which is used in the analysis of CFD problems. The simulations based on CFD are contingent upon combined numerical accuracy, cost of computations and precision of modeling.

Using ANSYS CFD, virtually, the system of fluid flow can be simulated using computer analysis. Analysis can be started by first of all creating a mathematical model of physics problem associated. The CFD method of solving entails 3 approaches:

- Finite Difference Method
- Finite Element Method
- Finite Volume Method

1.3.1 Eulerian Multiphase Model and k- ε Turbulence Model

Eulerian model defined separate volume fraction and velocity field for each flow phase independently. Each phase component's conservation equations can be individually solved so that the coupled equations are being explicit and logical (Subramanian 2013). In the present study, we employed Eulerian multiphase flow model to investigate the interactions between the flowing water and sediments during the entire scouring process. Here, two phases are considered water phase and sediment phase.

The turbulence flow around the pier is one of the main reason accounting for local scour. The most commonly used turbulence model is k-ε model, although its performance is not so good in cases of large adverse pressure gradients. Turbulent properties of the flow are represented by two extra transport equations.

Turbulent kinetic energy, k is the first transported variable that determines turbulence energy whereas the turbulent dissipation, ϵ is the second transported variable representing the turbulence scale.

1.4 OBJECTIVE OF DISSERTATION

The Main Objectives of the Study are:

1. Computational Fluid Dynamics Studies of local scouring around bridge piers of various cross sections namely **Circular, Square, Elliptical** and **Streamlined** shaped piers.
2. To study the variation in pressure, shear stress and turbulent kinetic energy distribution around bridge piers of various shapes.
3. To study the variation in velocity profiles around bridge piers of various shapes.

1.5 ORGANIZATION OF DISSERTATION

The report is fragmented into 5 Chapters:

- Chapter 1 elaborates the objective of the present study.
- Chapter 2 entails the literature review done with the associated topic.
- Chapter 3 presents numerical modelling used in the study.
- Chapter 4 elaborates the results and discussions.
- Chapter 5 includes the conclusion and future scope associated with the work done in the present study.

CHAPTER 2
LITERATURE REVIEW

Prasanna S V S N D L; Suresh Kumar N (2018) investigated the flow behavior and the scour phenomenon around the bridge piers. Moreover, the numerical simulations were even computed using CFD- Fluent, k – epsilon turbulence model, to elaborate the physics behind the scour formation. In the present study, scour depth was estimated making use of well-known empirical formulae. Further, it was also determined experimentally for two plan shapes of bridge piers viz., circular and oblong. The experimental results were in good consonance with the empirical formulae. The simulation results for dynamic and static pressures along with the velocity magnitude profiles, showed good similarities with the experimental results. Hence, CFD simulation tool can be used for wide understanding of the flow behavior around the bridge piers even without physical model studies

Wen Xiong, Pingbo Tang, Bo Kong, C. S. Cai (2016) proposed a three dimensional bridge scour model using eulerian multiphase model. Water is used as the primary phase and sediments are treated as secondary phase. They also investigated the three dimensional scour model for single phase flow using water as the only phase and compare it with multiphase model. During the model development process, they firstly examined the basic conservation equations and four unique simulation issues. The four simulation challenges are numerically solved and examined. Moreover, the single phase and two phase models are solved and analyzed. The two phase flow model for bridge scour is more accurate as compared to single phase model for safe and long term bridge designs in various river environments. They concluded that by optimizing the pier shapes or geometry, local scouring near the pier toe can be reduced.

Mohammad Vaghef, Hamed Dashtpeyma, Arash Adib, Javad Roohian (2011) In this study, they had done the numerical analysis of flow pattern around square bridge piers. Important variation factor they had taken is the Froud number of flow which was taken as 0.14, 0.1, 0,2 and 0.3. The rate of flow is taken as constant as 25 Kg/s. The length, width and height of the flume is taken as 3m, 0.6m and 1m respectively. Conditions that used for modeling is steady state for simulation type, K-epsilon for turbulent model. They showed the variation of velocity in flow domain and shear stress distribution on river bed for each of the four froud number.

From this study, they concluded that as the froud number of the flow increases, the velocity near the bed surface gets increased resulting in the rise of shear stress on lateral sides of the square pier. Unwanted flow behavior around the pier can be reduced by proper designing of bridge pier by considering the ratio of dimensions of width of pier and width of channel.

Zaid Hadi Obeid, Dr. Abdul-Hassan K. Al-Shukur (2016) used the CFD technique to simulate the 3D flow and local scouring around bridge piers. Navier-Stokes equations are solved with finite difference method with RNG k- turbulence model. Volume of fluid (VOF) technique is used to analyze the free water surface. They used six different shapes of bridge piers namely rectangular, oblong, elliptical, sharp nose, hexagonal and streamlined. Scour depth and velocity distribution are measured on bed surface both in numerical and experimental model. The results shown that there is an error ranges from (5.1%-9.8%) and (1.8%-6.1%) for prediction scour depth and maximum velocity respectively. The velocity distribution and scour depth calculations are conducted based on previous laboratory experiments.

J. S. Antunes do Carmo (2005) studied and analyzed the scour process in a fluvial environment in order to prevent a construction from failing.Through their experimental study, they had studied the scour failures. The necessary data common to all empirical formulas were taken as flow depth h and U of design flood, sediment properties expressed as d_{50}, pier size D, pier shape and alignment. A Rectangular cross-section flume of width 0.30 m and length 7.5 m is used for performing the test experiments. Acoustic Doppler Velocimeter(ADV) is used for the measurement of velocity and a Moulinet is used for the measurement of water flow discharge. They had taken four points above, behind, left and right of the pier and their distance from the pier , scour depth at the point and velocity at the point are measured and calculated . They also studied the methods for scour protection such as Riprap in which the surrounding area around the pier is replaced with some granular materials of size larger than the bed sediment size. This results in the considerable decrease in the scour process.

J. A. Vasquez, B. W. Walsh(2018) used a finite volume hydrodynamic model with k-ε turbulence modeling to simulate the 3-D flow around a pile. All the most important attributes of the scour process such as shape of scour hole, horse shoe vortex, sliding of sand on scour hole sides and bed ripples are thoroughly examined and collected by them. The equilibrium scour depth obtained during numerical results agreed fairly well with the experiments. The flow conditions for the simulation were: D = 10 cm, d_{50} = 0.26 mm, depth of flow = 20 cm, V/Vc = 1.6. It takes almost 2.5 hours to reach the equilibrium and 2.5 months for computation time.

Chang et al. (1999) analyzed and solved the to flow equations around a bridge pier with a fixed bed and no scour using a large-eddy simulation (LES) model. *Van Rijn (1984)* bed-load formula is used and the adjusted bed shear stress is induced in it for calculating the sediment transport. The time series data of *Ettema (1980*) was used to test the obtained results . The results are strongly coinciding with the data. Through this study, they made a strong conclusion that the flatbed sediment transport formula with an adjusted shear stress value can be applied to investigate the scour hole process with time.

Wenrui et al, (2009) investigated the scale effects on turbulent flow and scour around bridge piers. The appropriate scour equation was developed for HEC-18 from the laboratory experiments carried out in a relatively small scale. 3D CFD model was used to set up physical scale and boundary velocity, based on the Froude's similarity law to determine the sediment scour. The CFD simulation employed was a 2nd order turbulent model for calculating sediment scour and turbulent velocity.

Aghaee et al, (2010) carried out a 3D numerical simulation to study the turbulent flow around a vertical circular pier. The study adopted fully developed hydrodynamic equations viz., Reynolds Averaged Navier-Stokes equations (RANS) and Space Averaged Navier- Stokes equations. The numerical model results showed that the length and intensity of the wake and the horseshoe vortices were mainly affected with the turbulence models used.

May Than Zaw, Cho Cho Thin Kyi, Win Win Zin (2018) The scour depth around the pier is measured and compared between the numerical simulations and field observation at the Maubin Bridge site which is situated on the Yangnon-Sarmalout-Maubin highway crossing the river Toe (Maubin Township). The Fluent and Gambit software was used for the simulation process to calculate the scour depths around bridge piers and scour patterns are developed with time. Geometry of the bridge site was constructed in Gambit software and the numerical solution is carried out by the Fluent software. The velocity contour, velocity vectors and scour depth are analyzed around the piers. A comparison of scour depth obtained from the numerical simulation and field observation was carried out. From the results, it can be concluded that the mathematical model can simulate the process of local scour around piers and can obtain the equilibrium profiles similar to observation results.

P. X. Ramos, R. Maia, L. Schindfessel, T. De Mulder, J. P. Pêgo (2016) In their study, the three dimensional flow around a cylindrical pier mounted on a flat and fixed bed is examined and numerical results are obtained and analyzed. A three dimensional Navier Stokes model was also set up using Large Eddy Simulation(LES) approach. PISO solver was employed to study the turbulent flows. The experimental and the numerical results of velocity profiles around the pier and on the surface bed showed almost the same trend. Drag force on the pier was also calculated. Based on this study, it can be concluded that the flow behavior around hydraulics structures can be predicted through numerical modeling.

Zhu Zhi-wen, Liu Zhen-qing (2012) studied the local scour hole and its evaluation around a cylindrical bridge pier using numerical methods along with sediment transport theories. The time averaged Reynolds Navier Stokes equations with standard k-epsilon model was used to simulate the three dimensional flow field around the bridge pier. They employed the dynamic mesh approach to simulate the bed surface elevation. The bridge pier wall was taken as rough and the velocity and bed shear stress distribution was calculated around the pier and on the bed surface. The results shown that the location of maximum scour hole depth and the configuration predicted by the numerical model and the experimental study are different. The main reason behind this is due to the ignorance of slope collapse in erosion zone by the numerical model.

Deepika Bhulla, Rajendra Magar (2017) In this study, the scour phenomenon around bridge pier is explained. They also showed to prevent scouring, various factors affecting the scour depth, methods for predicting the scour and the measures that should be considered. According to them, the main factors involves characteristics of stream flow, characteristics of the bed materials, characteristics of pier and the depth of flow. They also studied the recent equations for calculating the scour depth such as *Lacey-Inglish* equation, *Laursen-Toch* equation, *Melville and Sutherland* equations. They suggested the Riprap protection for minimizing the scouring. They also suggested a formula to calculate the thickness of Riprap, twenty bridges were provided Riprap protection in Sweden using this equation and none showed significant scouring.

CHAPTER 3

METHODOLOGY

3.1 Procedure of FLUENT Simulation

The basic procedure steps of FLUENT 18.1 are shown below:

1) Generate the Model Geometry.
2) Select 2D or 3D Model.
3) Create the Mesh.
4) Select the solver formulation.
5) Material properties are defined.
6) Boundary Conditions are specified.
7) Adjust the solution control parameters.
8) Flow field is initialized.
9) Calculate the solution.
10) Examine the result.

3.2 CFD Model

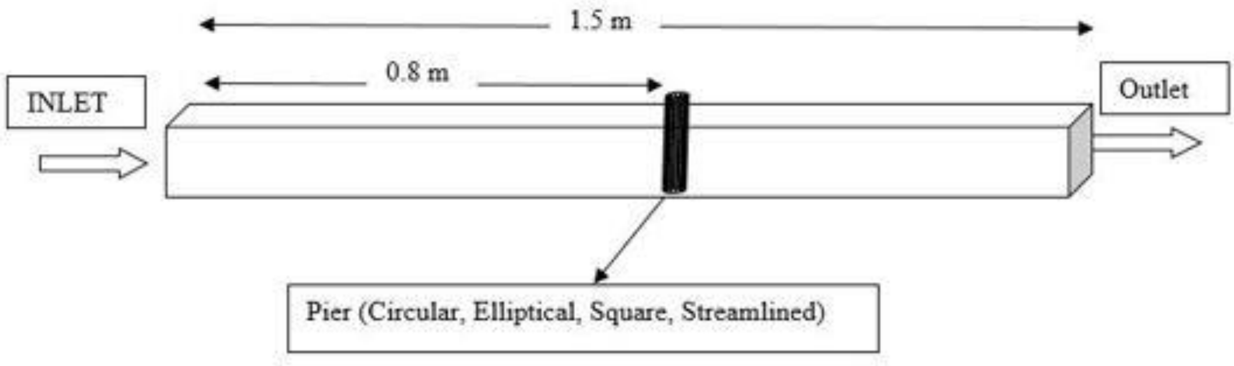

Figure 3.1. Specifications of the Study

- Inlet Velocity = 0.5 m/s
- Dimensions of the Rectangular Flume, Length = 1500 mm

  Width = 400 mm

  Height = 300 mm
- Dimensions of the Pier
  - Circular Pier, Diameter = 80 mm
  - Elliptical Pier, Major length = 80 mm

    Minor length = 40 mm
  - Square Pier, Length = 80 mm
  - Streamlined Pier, Major length = 160 mm

    Minor length = 80 mm.

3.3 Pre-processing

3.3.1 Geometry

The Geometry of the three dimensional model is constructed for all the four Pier shapes.The length of the rectangular flume is taken as 1.5 meters, width of the flume is taken as 0.4 meters and height of the flume is taken as 0.3 meters. Separate zones are assigned to sand bed and flow domain each of height 0.3 meters. There are four different cases of pier shapes are taken namely, Circular, Elliptical, Square and Streamlined shaped. The Dimensions of the circular, elliptical, square and streamlined pier are taken as diameter 80 mm, major length 80 mm, minor length 40 mm(elliptical), square side 80 mm and major length 160 mm, minor length 80 mm(streamlined). The fixed scour hole is considered for each pier shape and the depth of scouring for circular, elliptical, square and streamlined shape is taken as 3 cm, 2 cm, 3.5 cm and 1.5 cm respectively. The scour hole geometry is constructed using the Loft operation.

3.3.1.1 Circular Pier

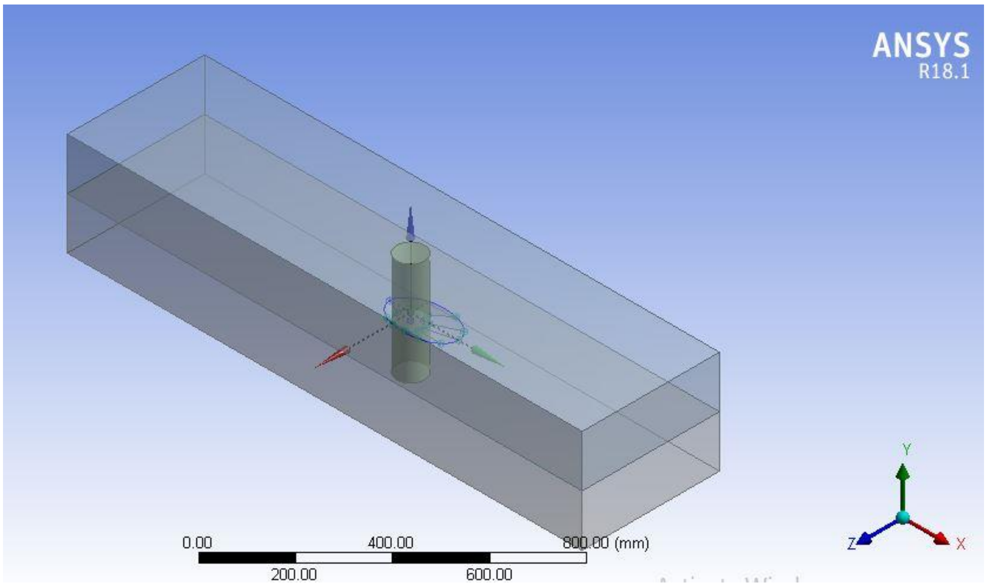

Figure 3.2 Geometry of Circular Pier

3.3.1.2 Elliptical Pier

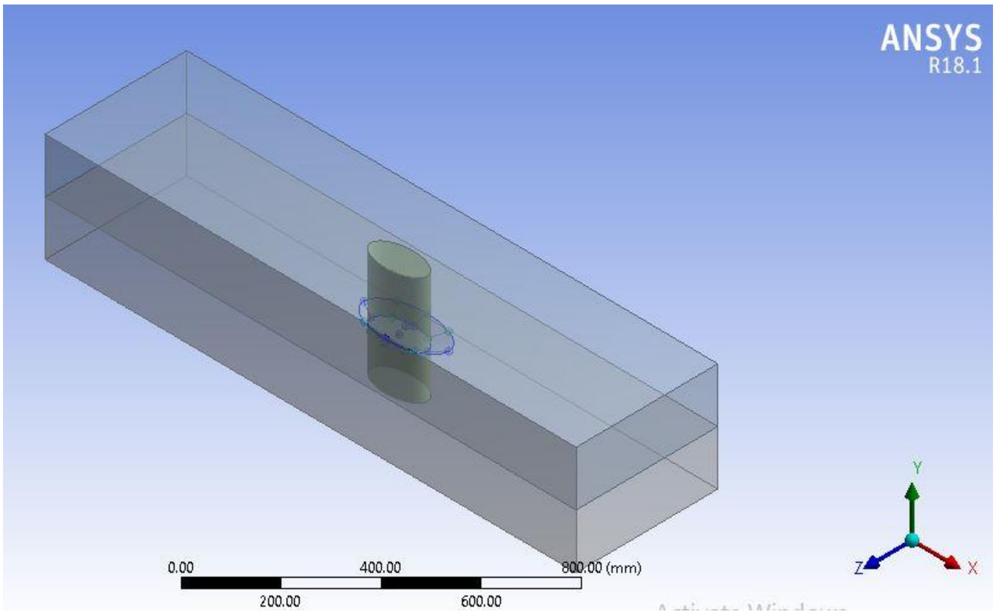

Figure 3.3 Geometry of Elliptical Pier

3.3.1.3 Square Pier

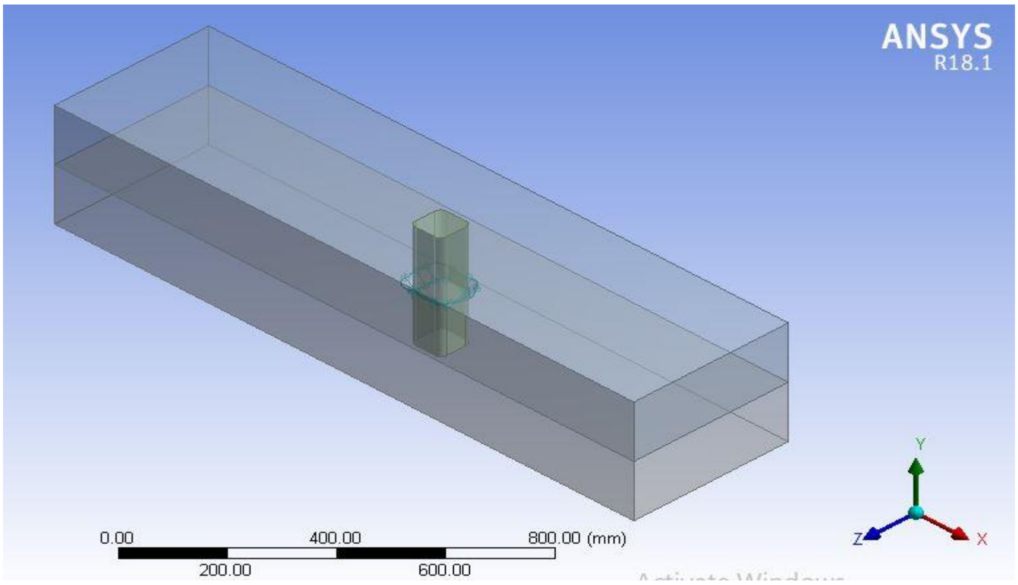

Figure 3.4 Geometry of Square Pier

3.3.1.4 Streamlined Pier

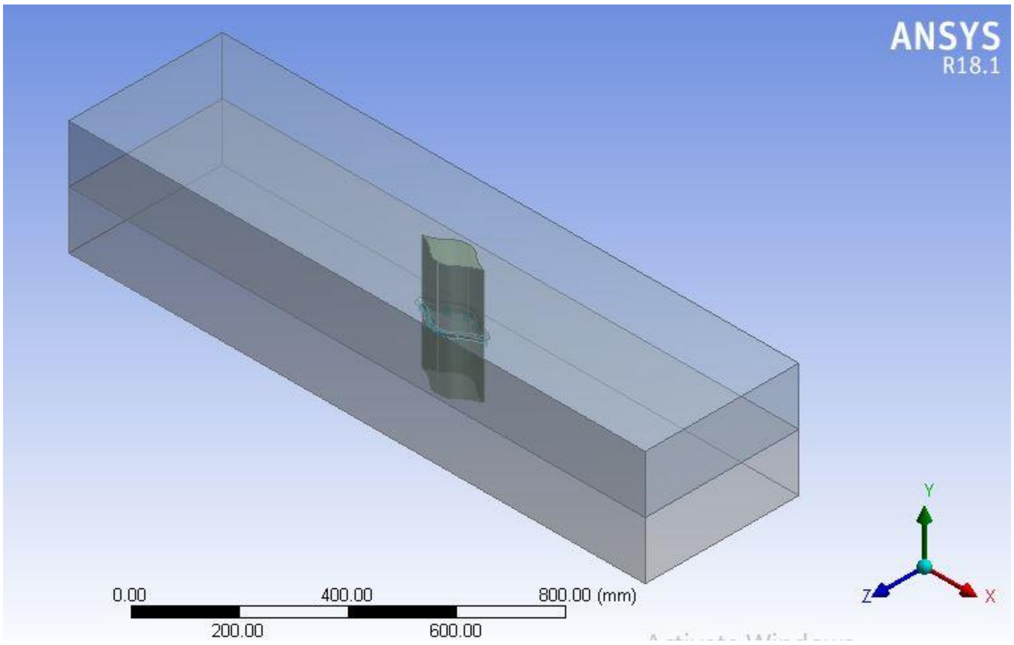

Figure 3.5 Geometry of Streamlined Pier

3.3.2 Mesh Generation

The accurateness of the numerical results increases with the increase in the fineness of the meshing. But it also leads to greater time requirement to solve the model. Mesh generation is an important pre-processing step in CFD.

For two dimensional flow situations, simple meshing such as rectangular grid meshing can be used and for three dimensional flow or complex flow situation, tetrahedral meshing is used.

In the present study tetrahedral meshing is used in all the four cases. Size function is taken as uniform; relevance center is taken as fine. Speaking about the smoothening, it is taken as high. Patch conforming method is used for flow domain and sand bed using tetrahedron meshing. Edge sizing is carried out for the pier and scour hole geometry using number of divisions method. The minimum size of the cells is taken as 2.3082e-004 m.

The meshing in all the four cases of pier is constructed with the same method. The meshing for square pier is given below along with the zoom view.

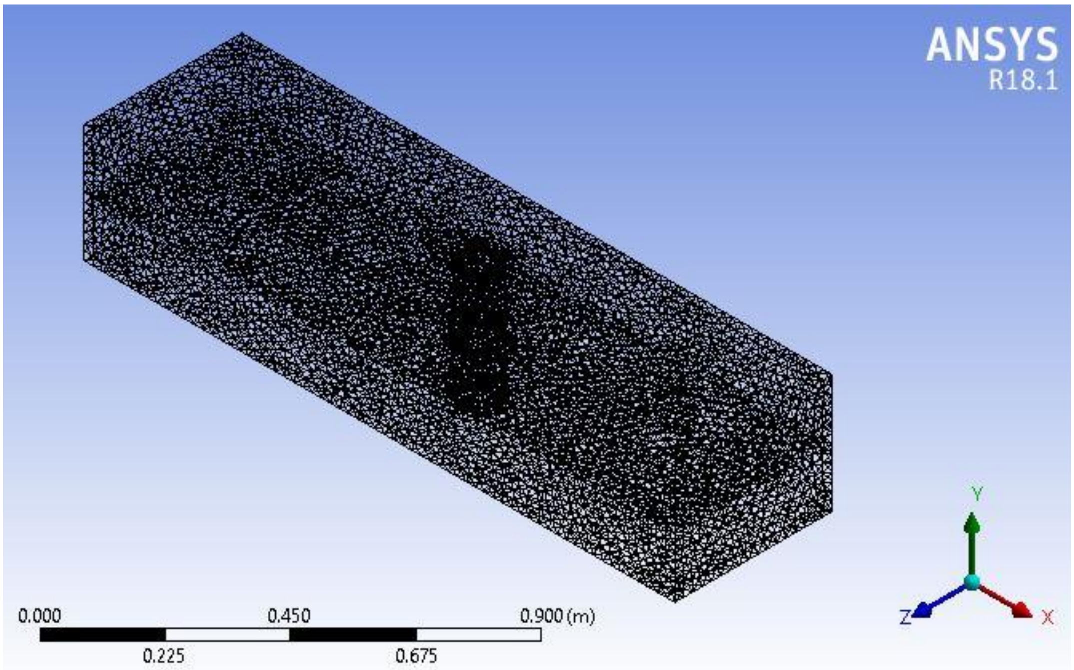

Figure 3.6 Meshing of Square Pier

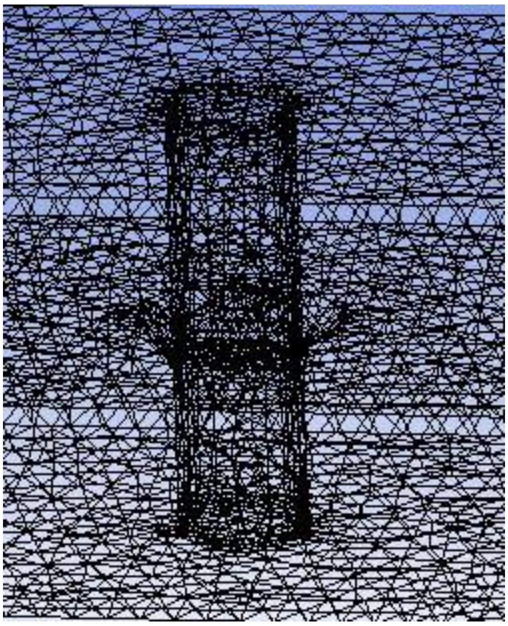

Figure 3.7 Zoom View of Mesh (Square Pier)

The detailed description of Meshing for Square Pier is tabulated below,

Table 3.1 Details of Mesh (Square Pier)

Sizing	
Size Function	Uniform
Relevance Center	Fine
Transition	Fast
Min Size	Default (2.3082e-004 m)
Max Face Size	Default (2.3082e-002 m)
Max Tet Size	Default (4.6165e-002 m)
Growth Rate	Default (1.850)
Automatic Mesh Based Defeaturing	On
Defeature Size	Default (1.1541e-004 m)
Minimum Edge Length	2.3516e-002 m
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)
Smoothing	High
Mesh Metric	None
Statistics	
Nodes	14986
Elements	74401

3.3.3 Named Selections

Named Selections is being carried out for all the four 3-Dimensional model. The inlet is taken at the left end of the flume and outlet is taken at the right end. The flume height is divided into two equal parts, the upper half is named as flow domain and the below half is taken as sand bed. Named selection is also done for the pier wall, flow domain walls and the sand bed wall. The top surface is considered as free surface with zero shear stress condition.

The Named selections for all the four cases are shown as follows,

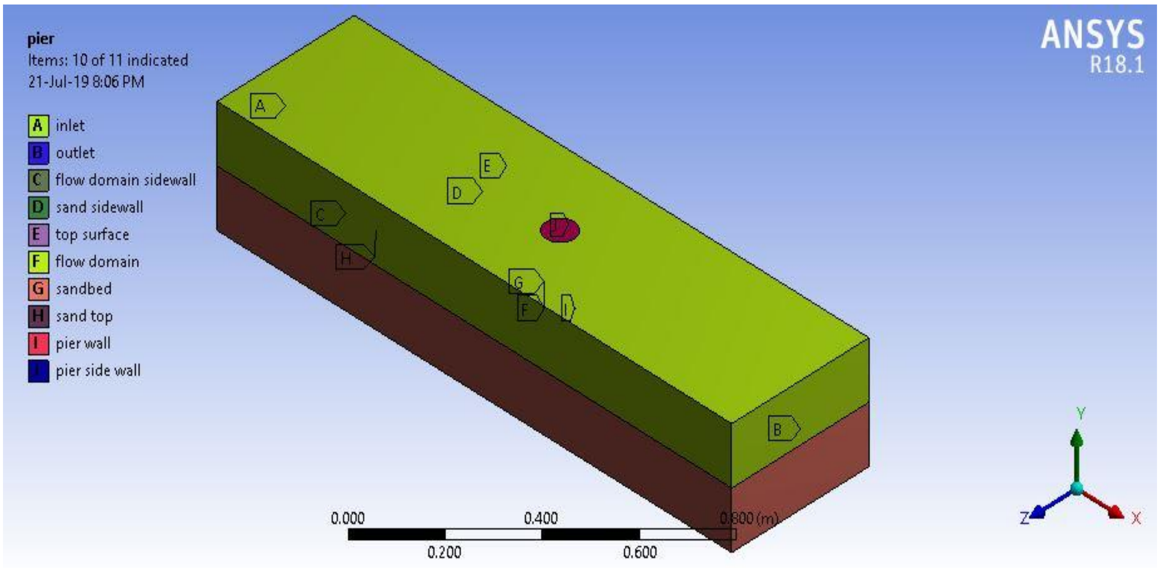

Figure 3.8 Named Selections of Circular Pier

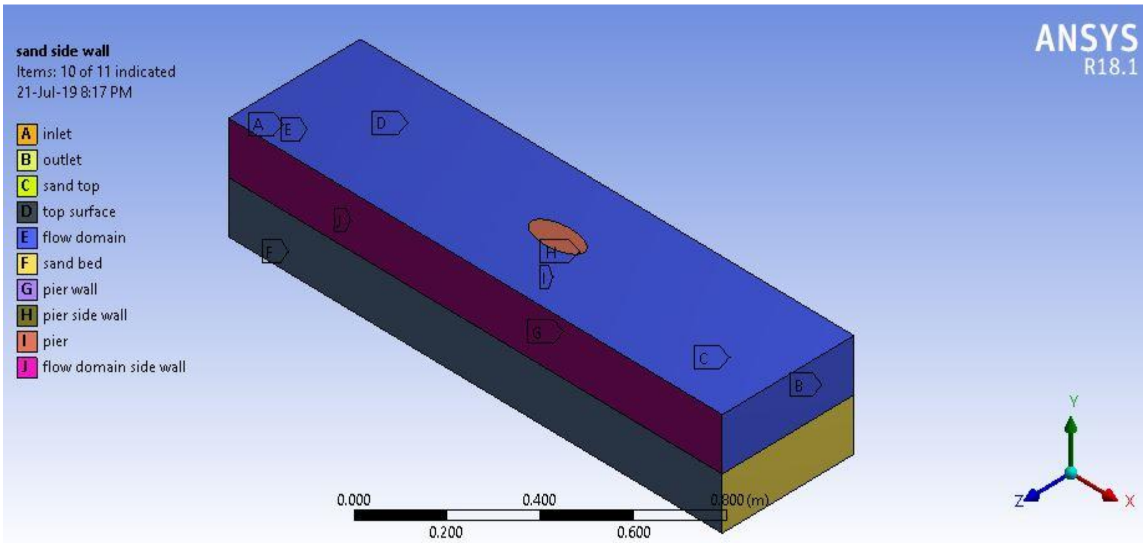

Figure 3.9 Named Selections of Elliptical Pier

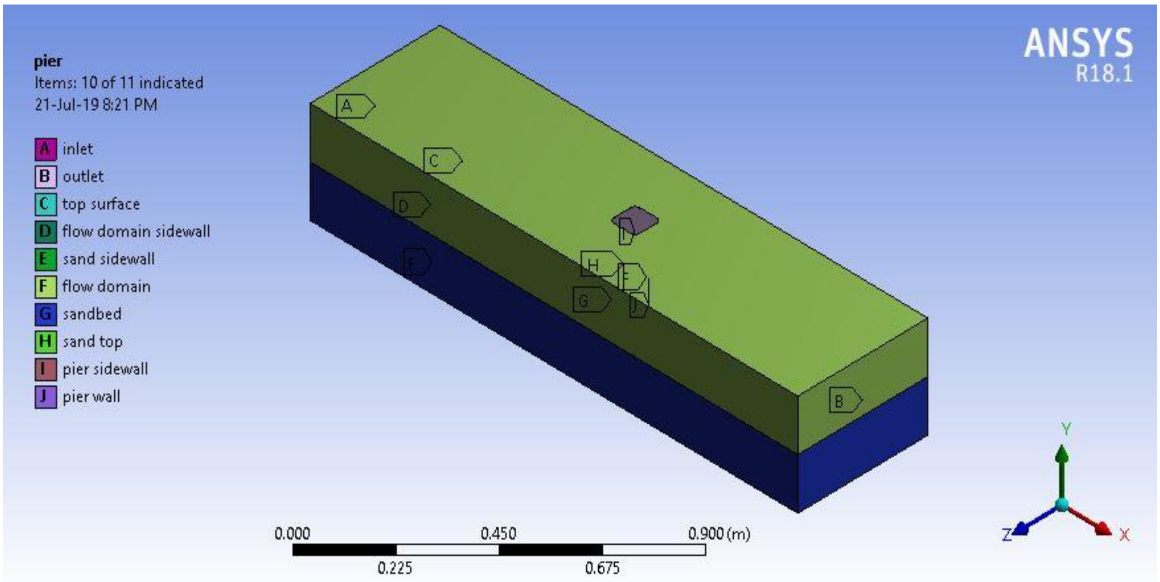

Figure 3.10 Named Selections of Square Pier

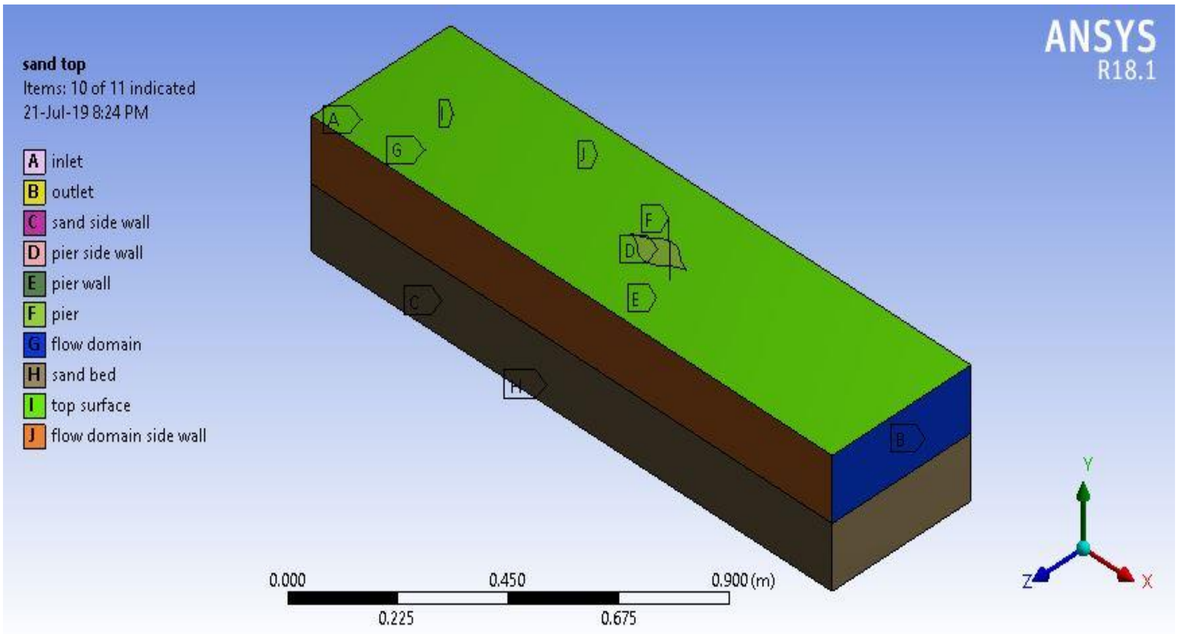

Figure 3.11 Named Selections of Streamlined Pier

3.3.4 Setup

Setup is the most important pre-processing step in Ansys Fluent. In setup, we have to decide which model should be employed to solve the continuity equation, momentum equation and energy equation. For single phase flow multiphase model is switched off and for two phase or three phase flow, multiphase model is switched on. Multiphase modeling can be done by Eulerian multiphase model, Volume of fluid(VOF) multiphase model and Mixture model. If the flow is laminar, Laminar viscous model is used and if the flow is turbulent, we can use either of k-ε turbulence model or k-ω turbulence model or Reynolds average navier stokes equations(RANS) model.

The Material properties can be altered as per the flow situations. In case of multiphase flow like scouring on river bed, water liquid is represented as the primary phase and sediment load is defined as the secondary phase. The region in the vicinity of the pier is patched with sand with volume fraction as 1 and diameter 0.0005 m. The packing limit of the sand is taken as 0.63 and granular viscosity 1e-05 kg/m-s.

In the present study, for all the four cases circular, elliptical, square and streamlined piers, the model description is as follows,

Table 3.2 Details of Numerical Setup

Model Phase	Multiphase Eulerian Model
Turbulence Model	Realizable k- ε Turbulence Model with Standard Wall Function
Material Properties	Water liquid - Primary Phase
	Sediment(Sand)- Secondary Phase
Acceleration due to Gravity	Included, g = 9.81m/s2 in negative y-direction
Cell Zone Conditions	Flow Domain - Fluid
	Pier - Solid
	Sand bed - Solid

Table 3.3 Boundary Conditions

Section	Type	Initial Value
Inlet	Velocity Inlet	0.5 m/s^2
Outlet	Pressure Outlet	Gauge Pressure = 0
Flow Domain Sidewall	Wall	Stationary Wall with No slip condition
Pier Wall	Wall	Stationary Wall with No slip condition
Top Surface	Wall	Stationary Wall with Zero Shear Stress (Free Surface)
Sand Top	Wall	Stationary Wall with No slip condition

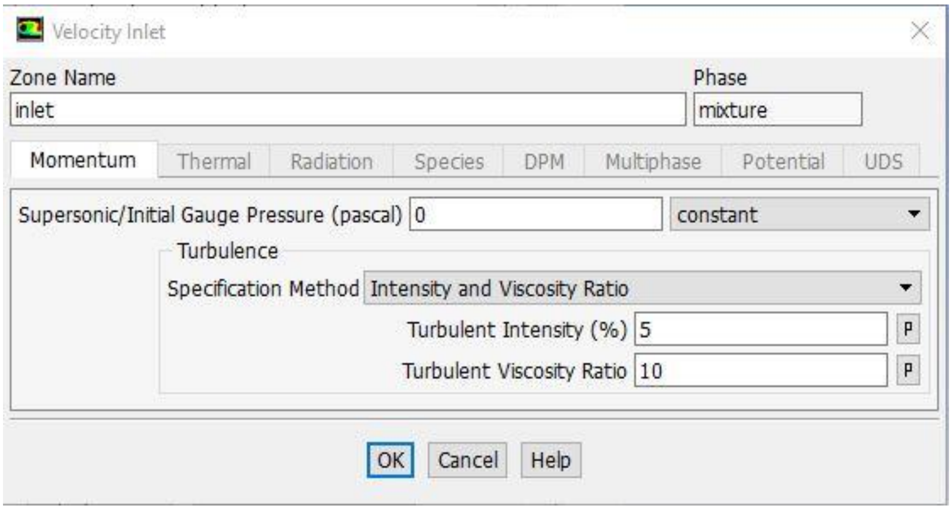

Figure 3.12 Inlet Boundary Condition(Mixture)

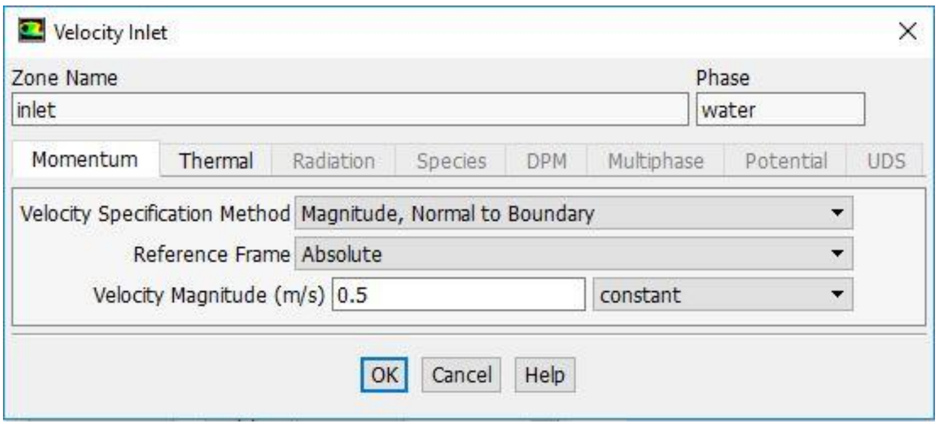

Figure 3.13 Inlet Boundary Condition(Water)

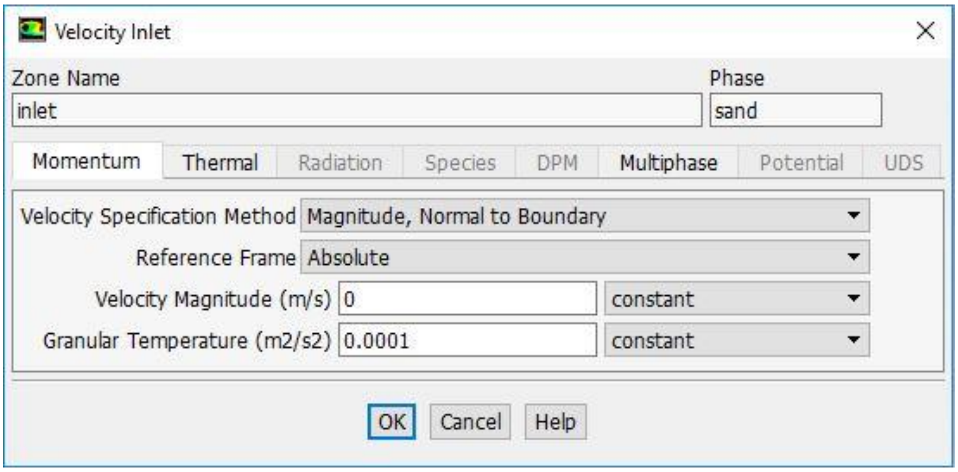

Figure 3.14 Inlet Boundary Condition(Sand)

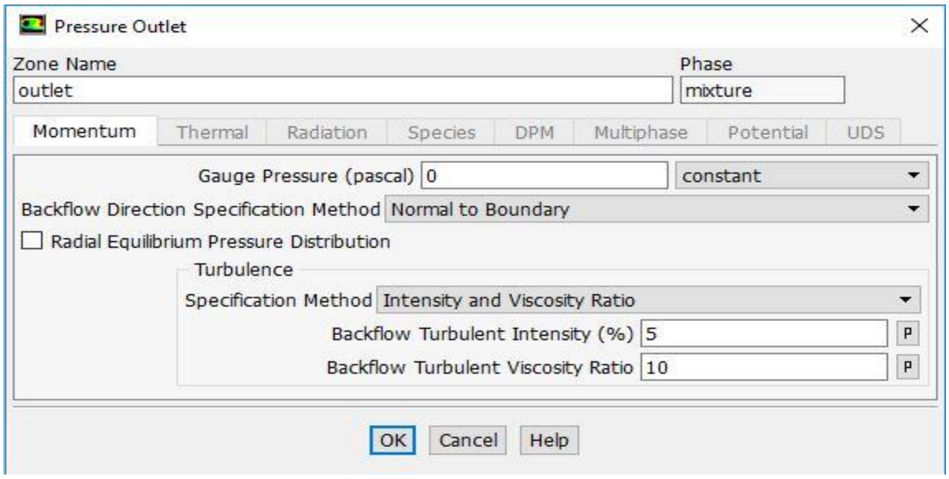

Figure 3.15 Outlet Boundary Condition(Mixture)

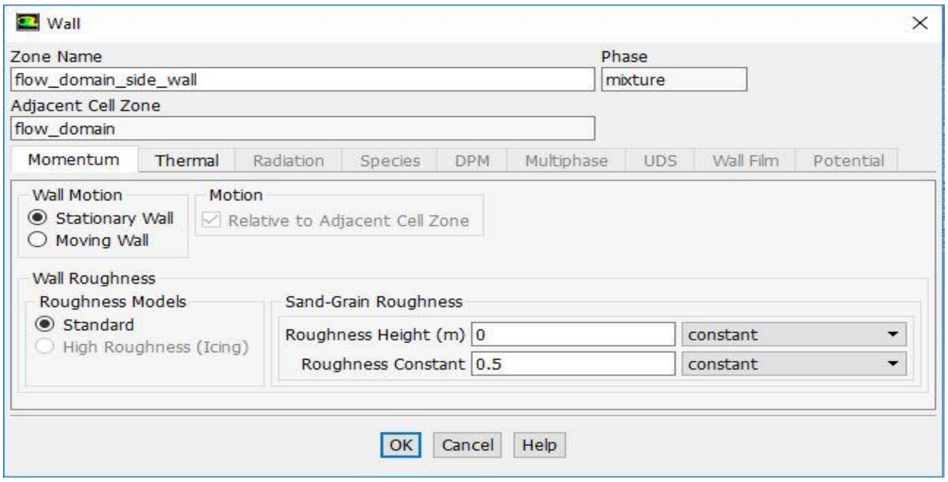

Figure 3.16 Flow Domain Sidewall (Mixture)

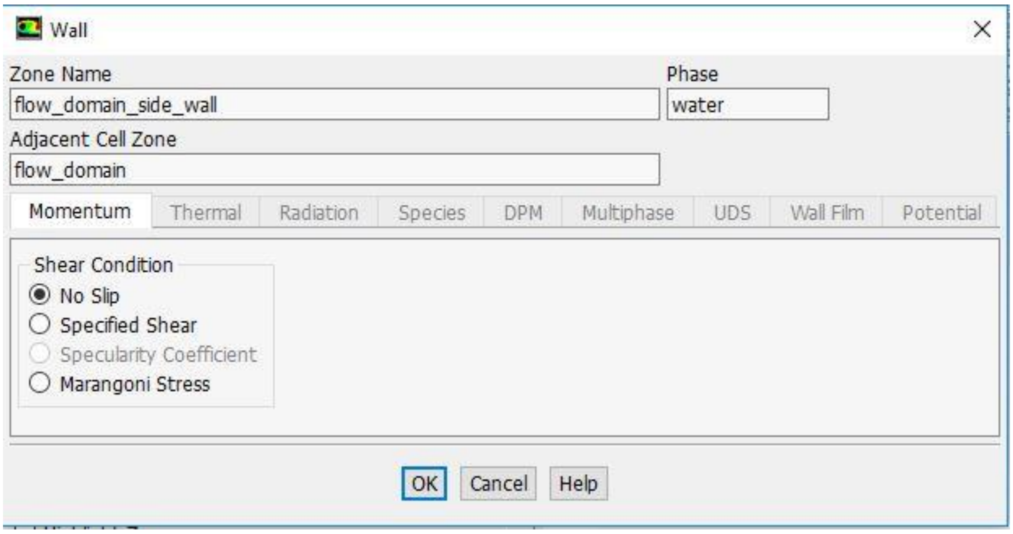

Figure 3.17 Flow Domain Sidewall (Water)

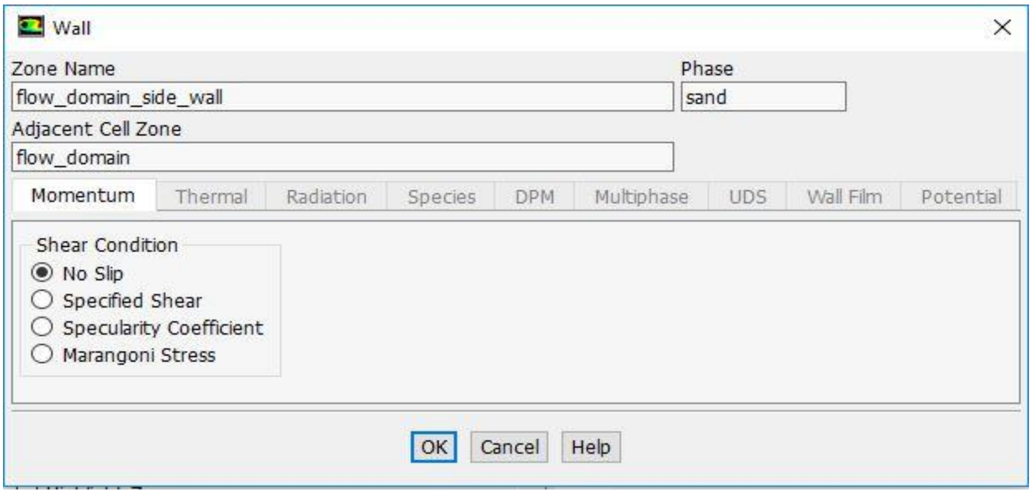

Figure 3.18 Flow Domain Sidewall (Sand)

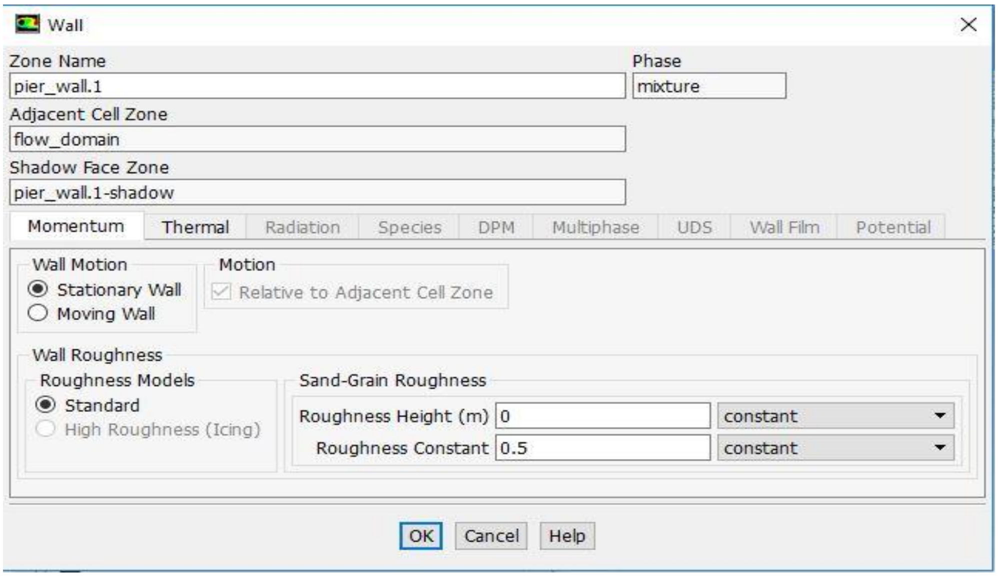

Figure 3.19 Pier Wall (Mixture)

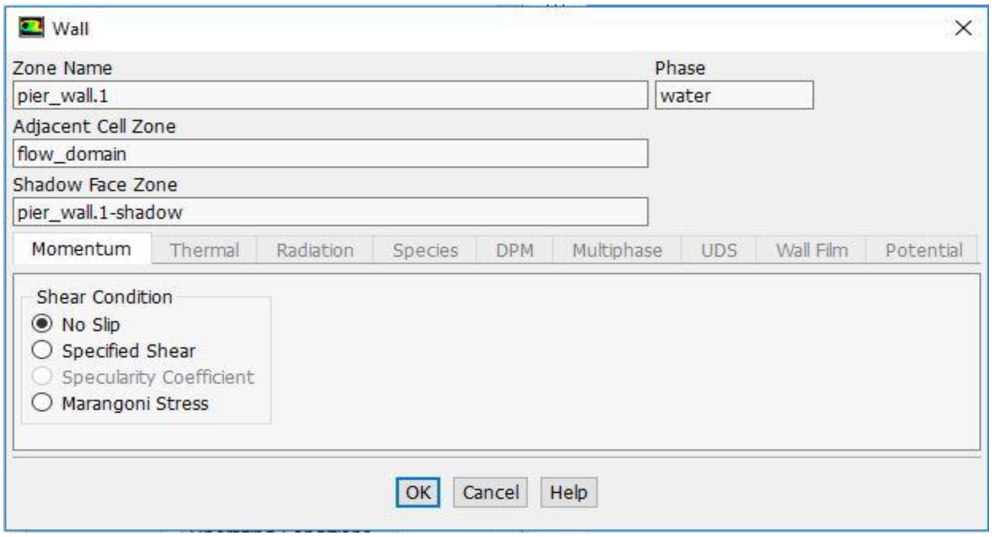

Figure 3.20 Pier Wall (Water)

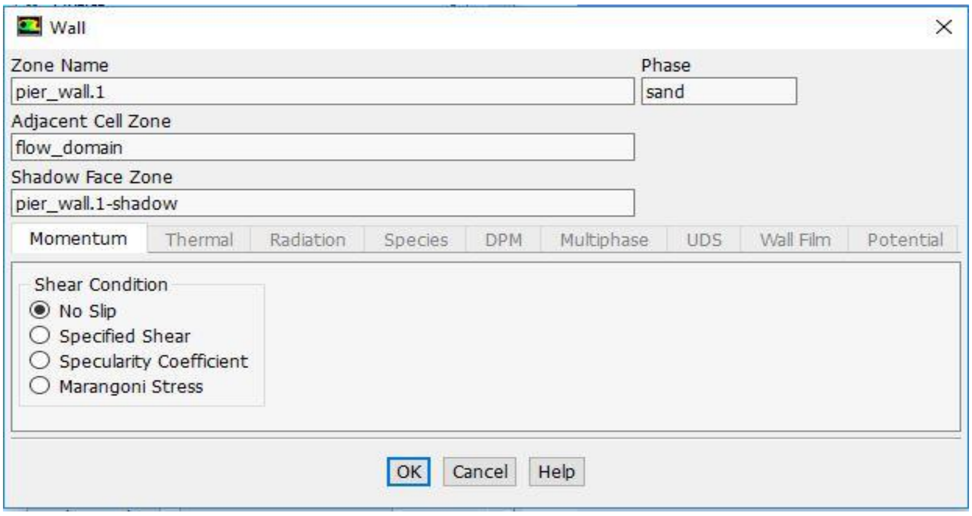

Figure 3.21 Pier Wall (Sand)

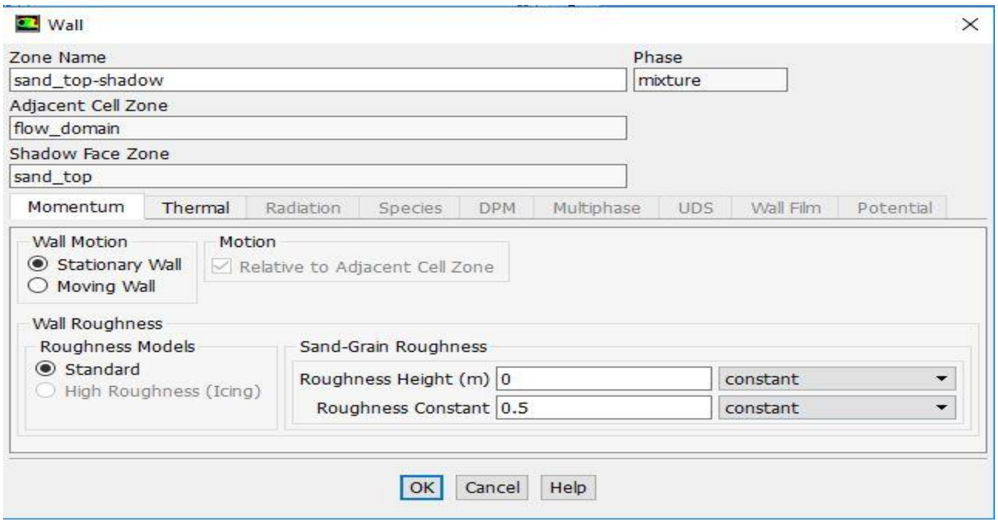

Figure 3.22 Sand Top (Mixture)

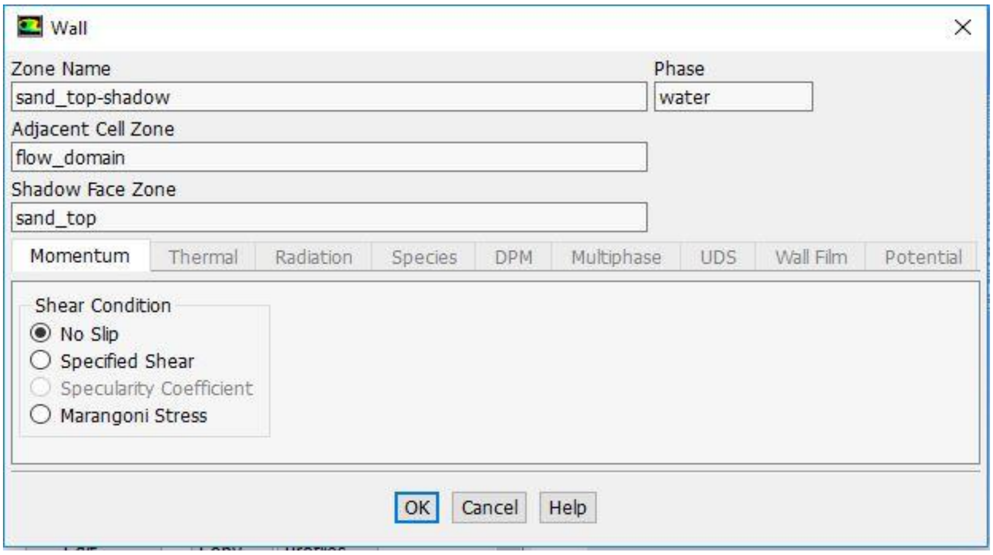

Figure 3.23 Sand Top (Water)

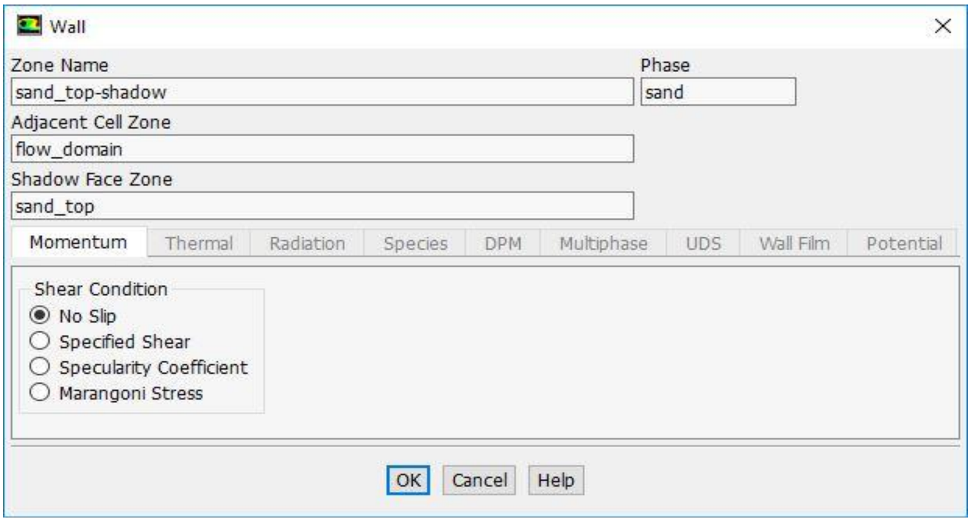

Figure 3.24 Sand Top (Sand)

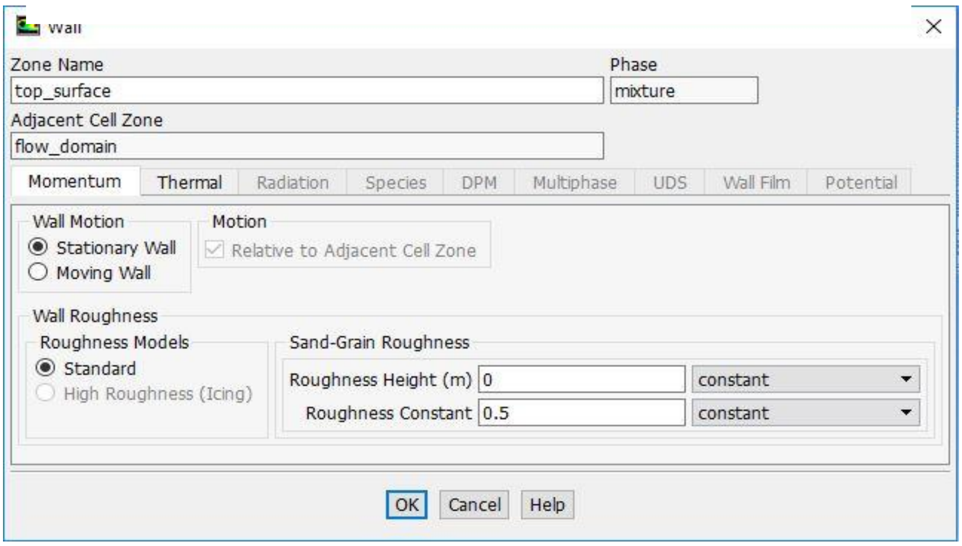

Figure 3.25 Top Surface (Mixture)

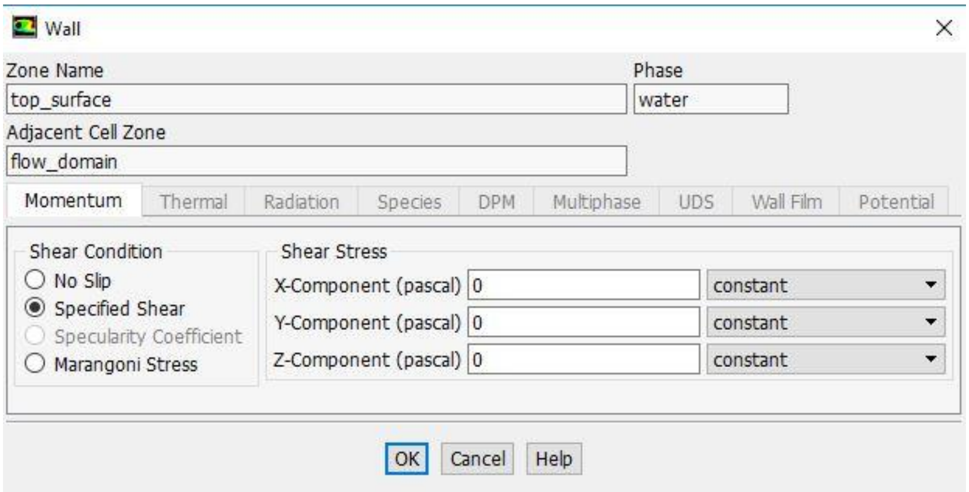

Figure 3.26 Top Surface (Water)

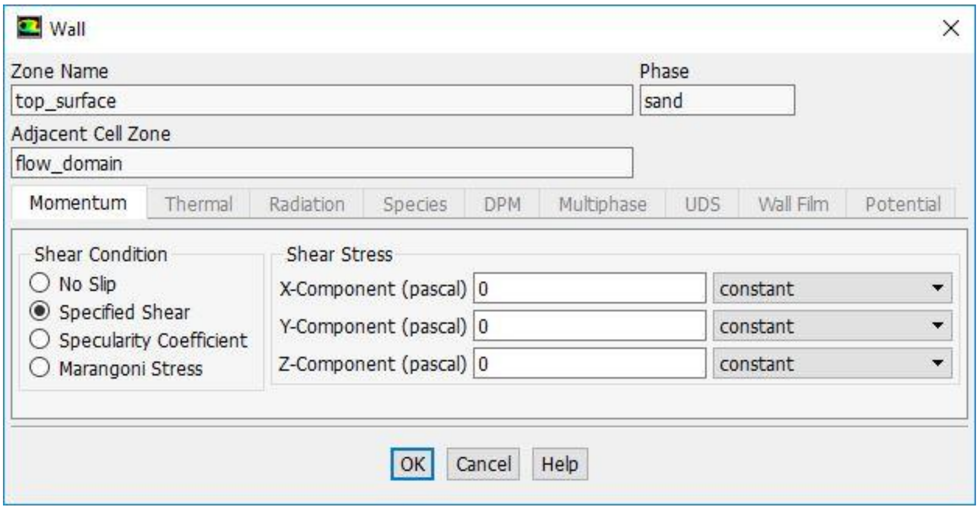

Figure 3.27 Top Surface (Sand)

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Results of the Flow Models

The numerical modelling procedure of the multiphase flow behavior is consisting of two main steps. The first step is the creation of geometry and mesh generation. The second consists of calculation of flow field in the flow domain. After generating the geometry and mesh, the solution setup includes defining the general and multiphase model, primary and secondary phases and their interaction. Turbulence and viscous model, boundary conditions, initialization of the flow field. After the convergence of the solution is obtained, post-processing and analysis of results are made.

After getting the numerical solution, the values of the flow parameters like pressure, velocity, shear stress etc. are predicted in the x, y and z direction.

4.1.1 Residual Error Iteration Plot

As shown below in the residual error iteration plot, the errors are gradually decreases with increase in number of iterations. The residual errors are also below the zero line; this shows that the solution is going in the right direction. Two main equations are continuity equation and momentum equation whose residual error graphs are going down smoothly. Number of iterations are 280 with time step size as 0.05 sec and number of time steps as 70. Number of iterations per time step is four.

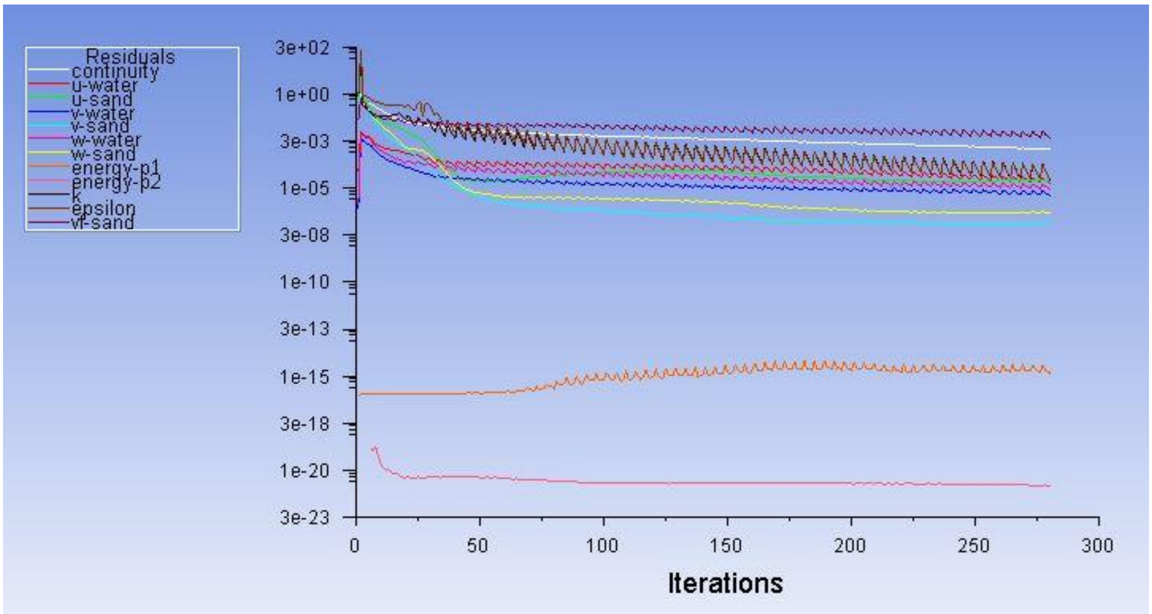

Figure 4.1 Residual Error Plot of Circular Pier

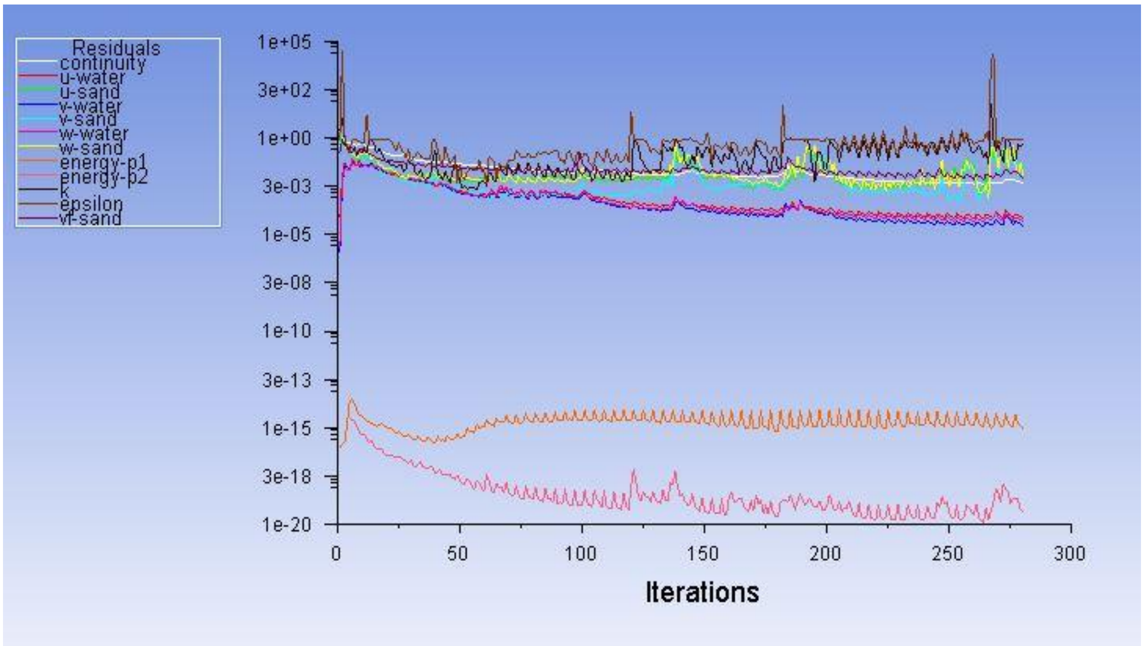

Figure 4.2 Residual Error Plot of Elliptical Pier

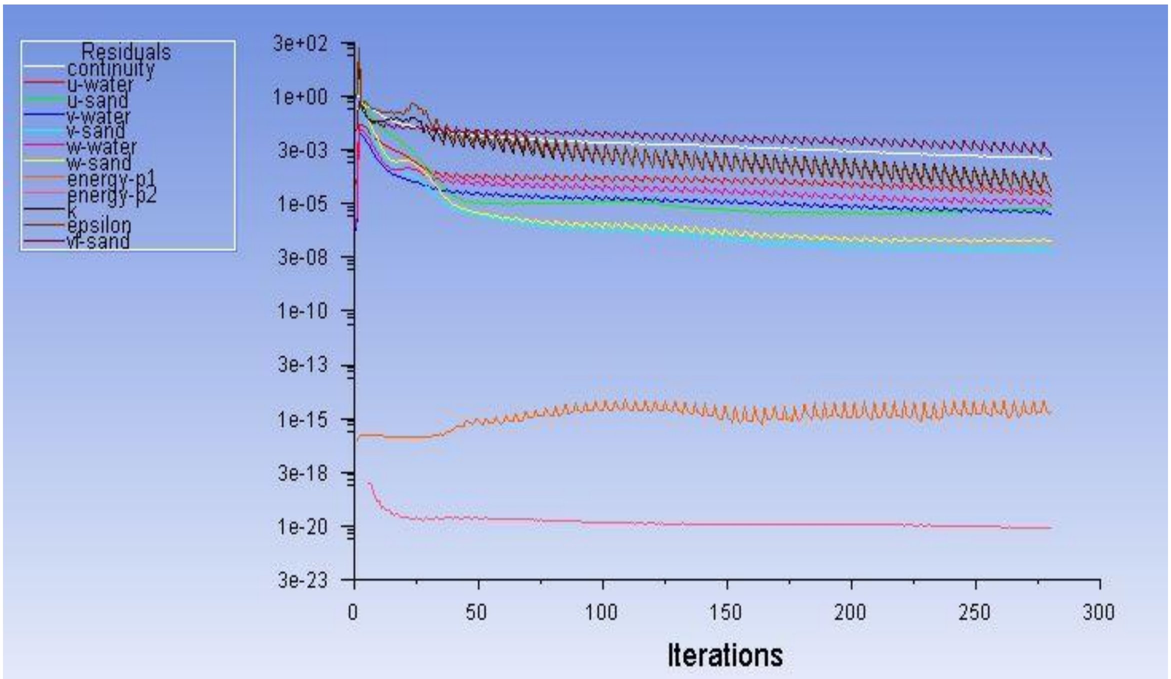

Figure 4.3 Residual Error Plot of Square Pier

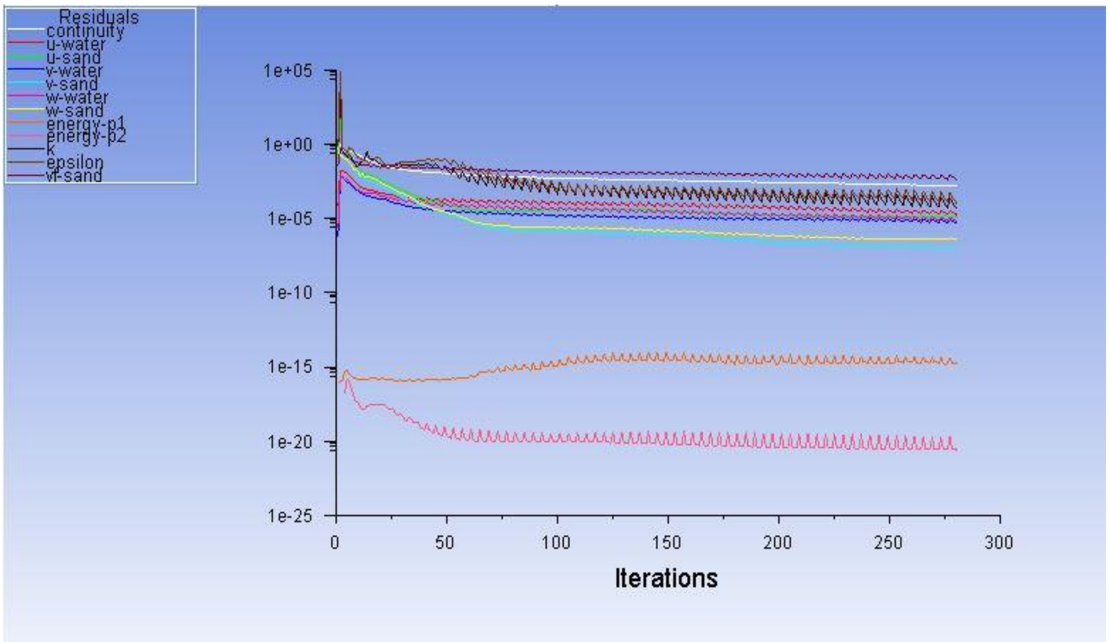

Figure 4.4 Residual Error Plot of Streamlined Pier

4.1.2 Pressure Contour variations in flow domain

The pressure distribution variation is simulated in the flow domain through numerical modeling. The pressure contours are shown on the pier wall, sand bed and on 2 planes one longitudinal and other is lateral to the flume passing through the pier. The region where the pressure values are greater indicates the presence of local scouring. The pressure values are also collected and compared on the 3 lines Line1, Line2 and Line3 on the pier wall for all the four pier shapes.

4.1.2.1 Pressure Contour on Pier Wall

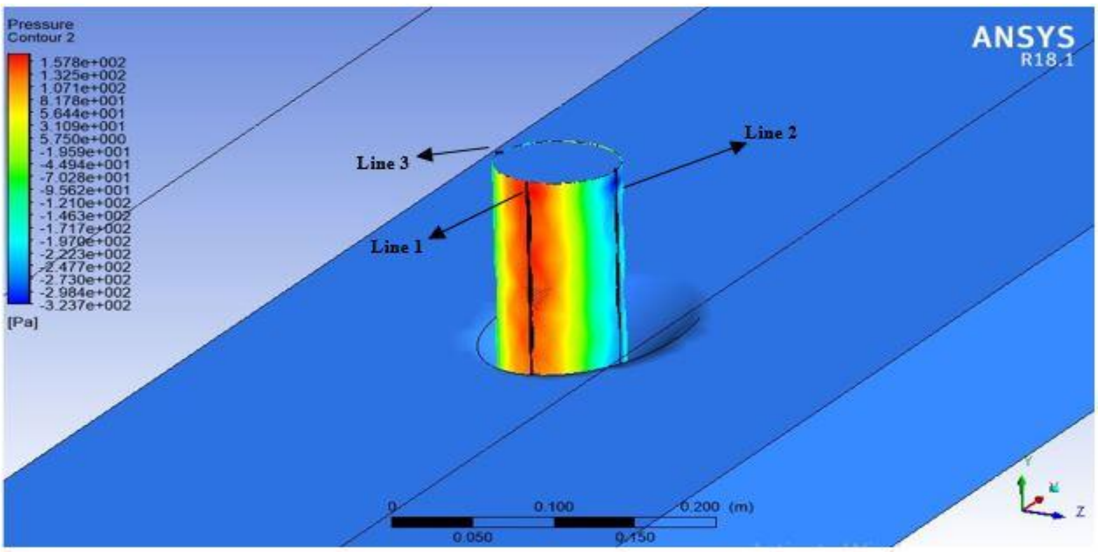

Figure 4.5 Pressure Contour on Circular Pier

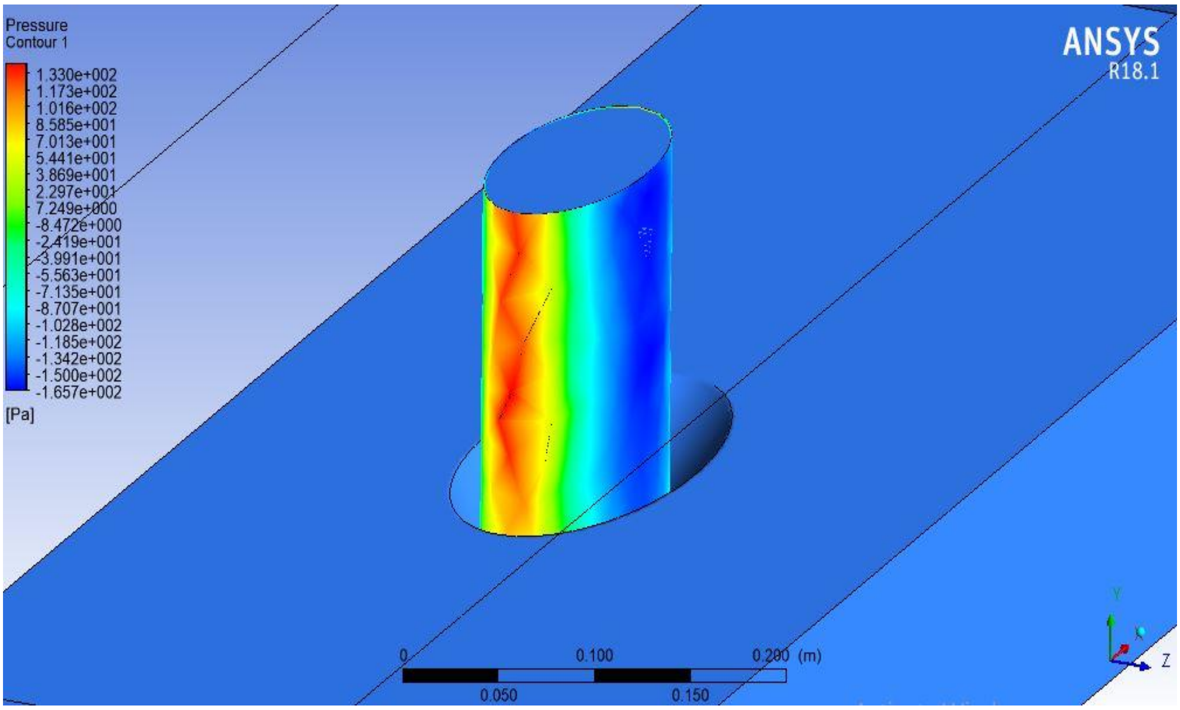

Figure 4.6 Pressure Contour on Elliptical Pier

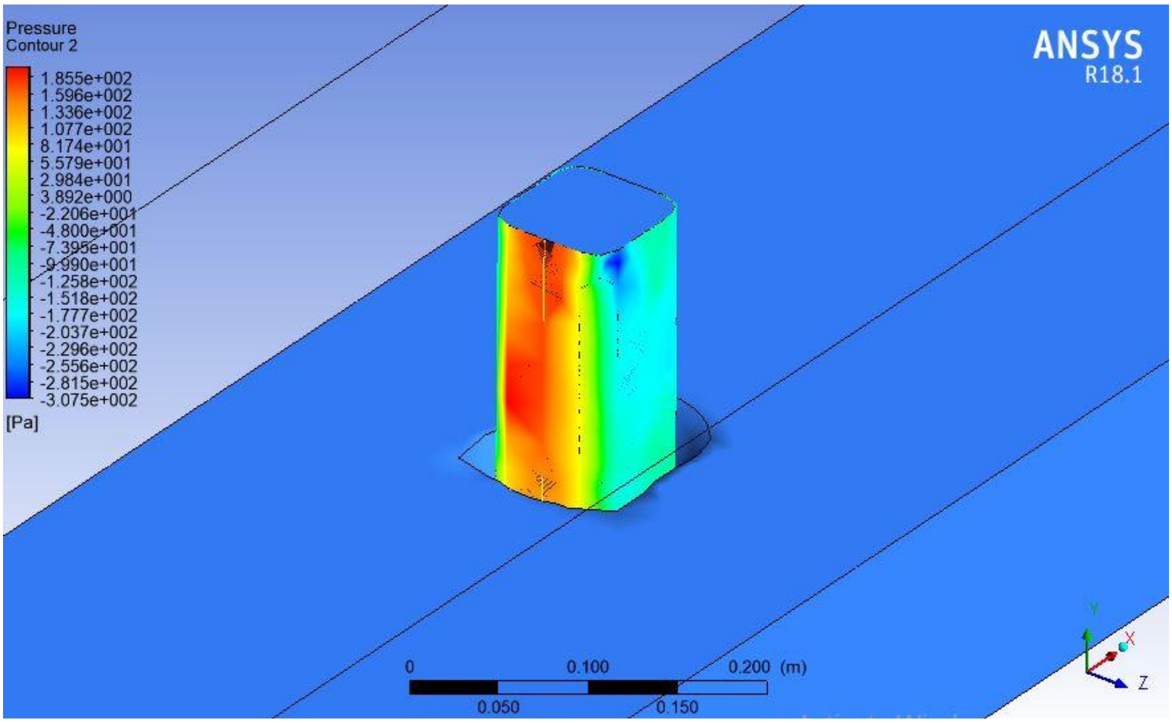

Figure 4.7 Pressure Contour on Square Pier

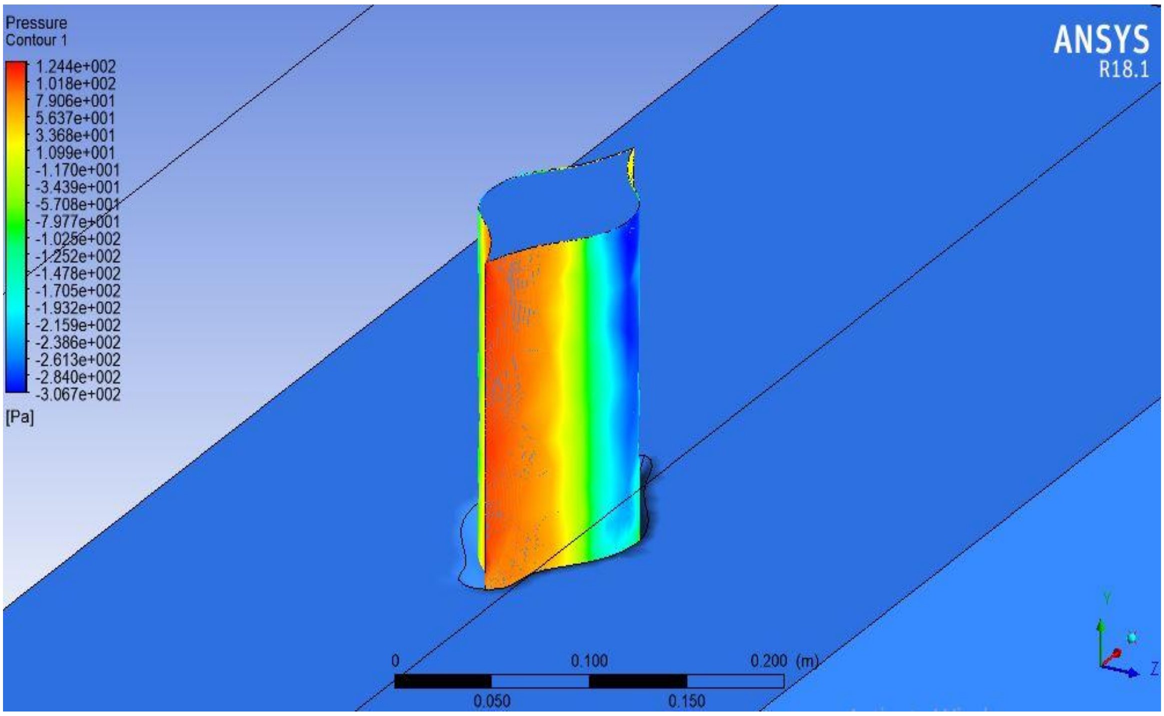

Figure 4.8 Pressure Contour on Streamlined Pier

The Graphs showing the pressure comparisons for 4 pier shapes on line 1, line 2 and line 3 are shown below.

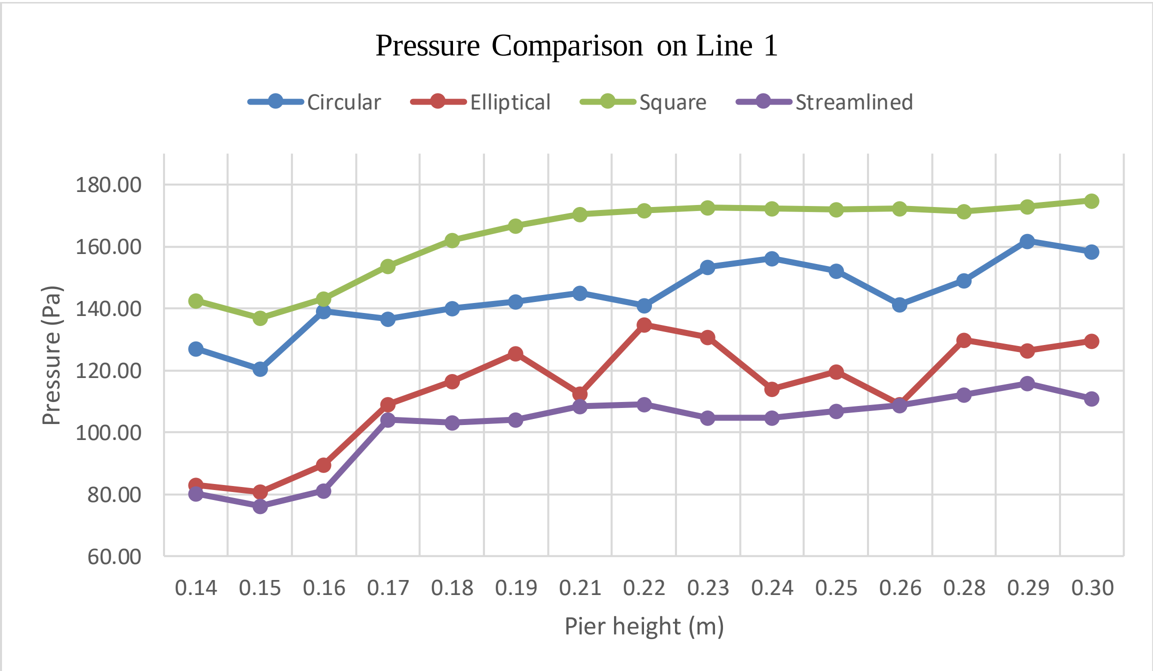

Figure 4.9 Pressure Comparison on Line 1

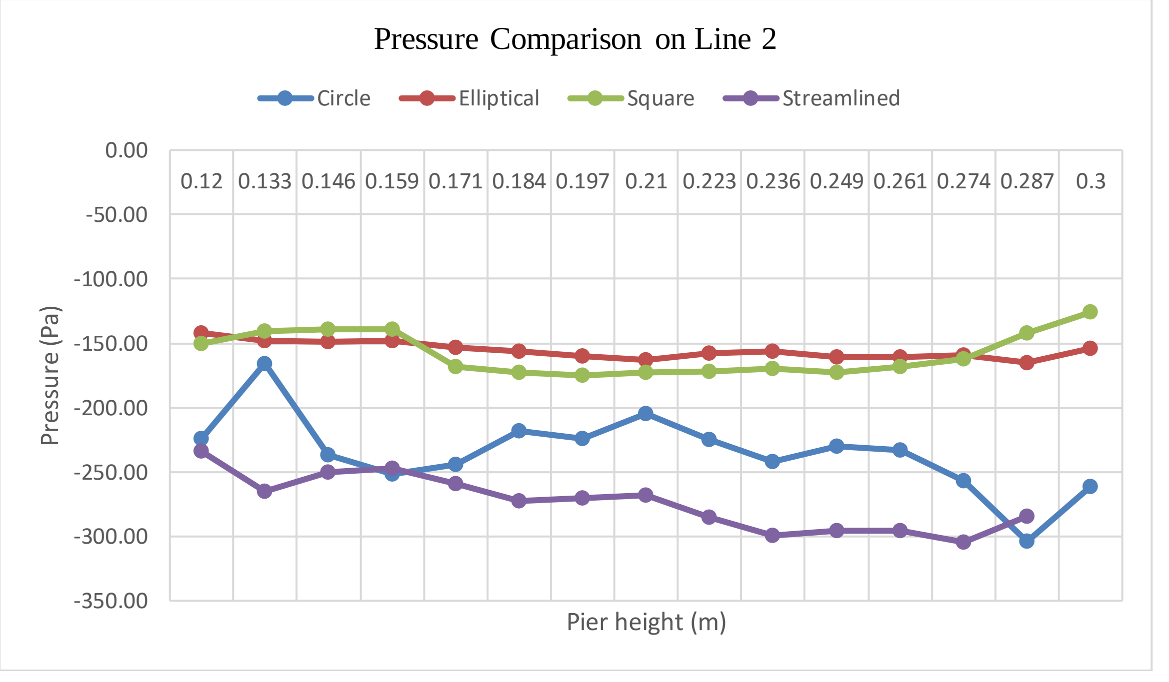

Figure 4.10 Pressure Comparison on Line 2

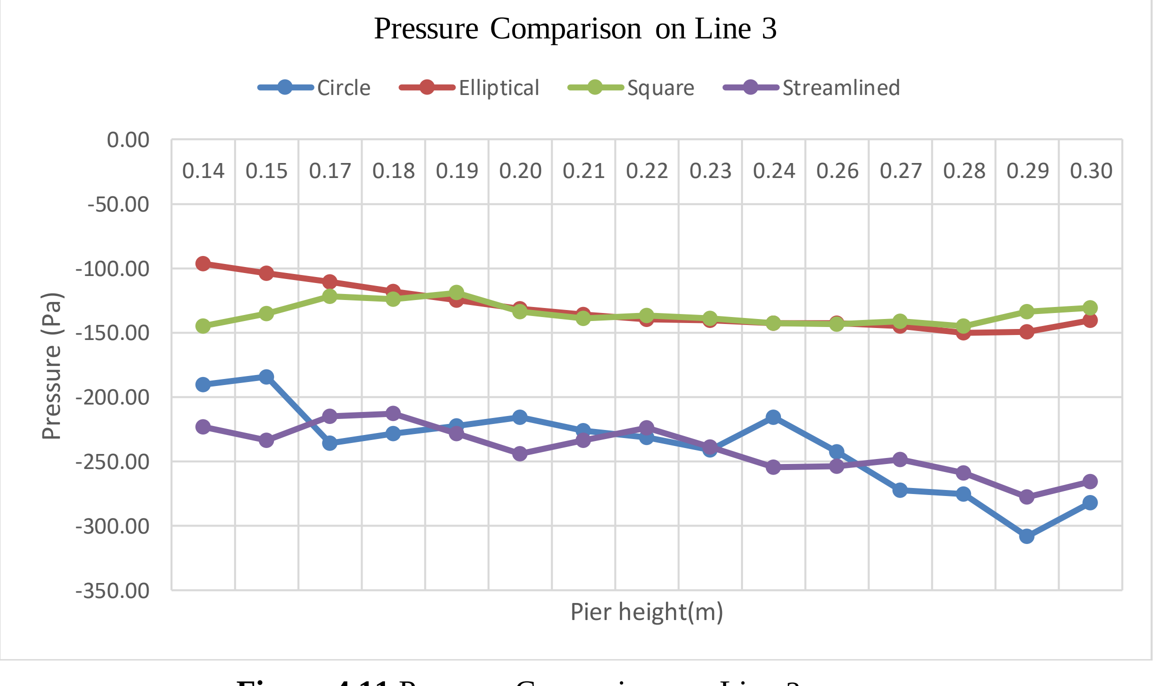

Figure 4.11 Pressure Comparison on Line 3

The table showing the pressure values on pier wall at Line 1, Line 2 and Line 3 is given below,

Table 4.1 Pressure values on Pier Wall on Line 1

Pier Height(m)	Pressure(Pa) on Line 1			
	Circular	Elliptical	Square	Streamlined
0.14	127.00	83.03	142.37	80.09
0.15	120.39	80.69	136.90	76.19
0.16	139.26	89.56	143.21	81.14
0.17	136.68	109.08	153.77	104.07
0.18	140.04	116.55	161.92	103.24
0.19	142.36	125.31	166.62	104.12
0.21	144.90	112.57	170.53	108.34
0.22	141.10	134.72	171.70	108.92
0.23	153.52	130.80	172.48	104.82
0.24	156.02	114.06	172.25	104.64
0.25	152.20	119.61	172.06	106.76
0.26	141.31	108.90	172.16	108.59
0.28	148.99	129.82	171.42	112.16
0.29	161.83	126.45	173.04	115.75
0.30	158.31	129.61	174.89	110.84

Table 4.2 Pressure values on Pier Wall on Line 2

Pier Height(m)	Pressure on Line 2			
	Circular	Elliptical	Square	Streamlined
0.14	-224.19	-142.46	-150.25	-260.57
0.15	-217.71	-148.01	-140.78	-265.32
0.16	-250.00	-148.88	-138.60	-242.82
0.17	-251.55	-148.21	-139.48	-251.36
0.18	-243.95	-152.56	-167.96	-262.65
0.19	-218.16	-156.03	-172.92	-273.05
0.21	-223.92	-160.20	-175.06	-270.31
0.22	-204.94	-162.87	-173.45	-267.57
0.23	-224.64	-158.10	-171.94	-281.19
0.24	-242.20	-155.97	-170.28	-296.30
0.25	-229.68	-160.55	-173.14	-297.32
0.26	-232.60	-160.98	-168.30	-294.23
0.28	-257.44	-159.05	-162.14	-297.79
0.29	-303.98	-164.85	-142.12	-305.55
0.30	-260.77	-153.83	-125.78	-284.23

Table 4.3 Pressure values on Pier Wall on Line 3

Pier Height(m)	Pressure on Line 3			
	Circular	**Elliptical**	**Square**	**Streamlined**
0.14	-190.64	-96.61	-144.89	-223.11
0.15	-184.24	-104.32	-135.39	-233.86
0.17	-235.54	-110.57	-121.73	-215.06
0.18	-228.30	-118.18	-123.96	-213.00
0.19	-222.66	-125.00	-119.26	-228.55
0.20	-215.70	-131.26	-133.81	-244.34
0.21	-226.29	-135.98	-139.36	-233.39
0.22	-231.76	-139.49	-136.55	-223.85
0.23	-240.76	-140.43	-138.87	-238.94
0.24	-215.86	-142.92	-142.79	-254.85
0.26	-242.62	-142.63	-143.59	-253.45
0.27	-272.70	-145.30	-141.19	-248.27
0.28	-275.52	-150.26	-145.13	-259.14
0.29	-308.25	-149.67	-133.82	-277.70
0.30	-282.35	-140.22	-131.02	-265.98

4.1.2.2 Pressure Contour on Sand Bed

The pressure in the region near to the pier is comparatively more as compared to other regions far away from the pier. The results shown that the square pier offers maximum pressure while on the other hand streamlined pier applies minimum. This indicates that streamlined pier is more efficient as compared to other pier shapes. The pressure values are also noted on two lines one longitudinal and other lateral to flume passing through the pier.

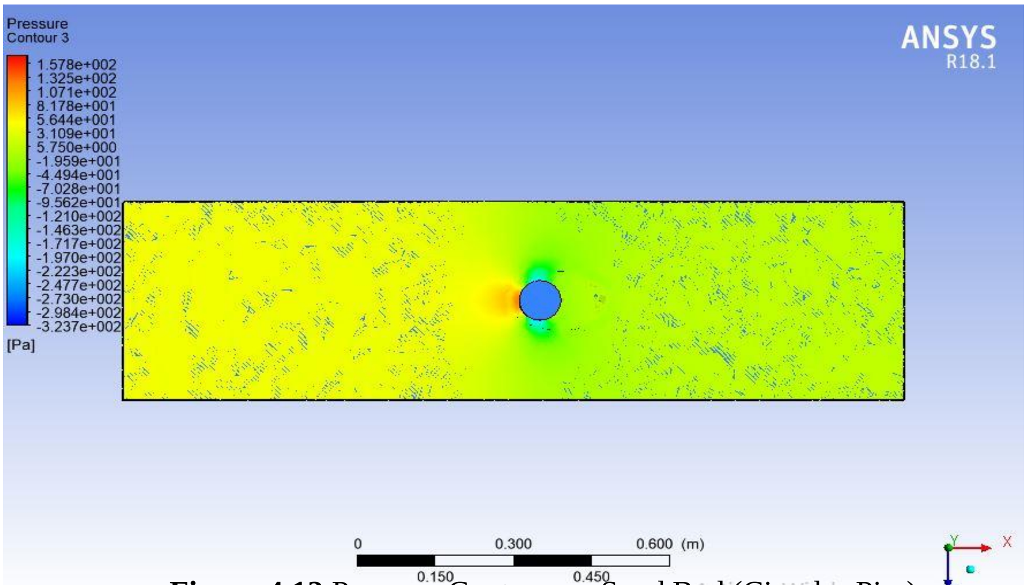

Figure 4.12 Pressure Contour on Sand Bed (Circular Pier)

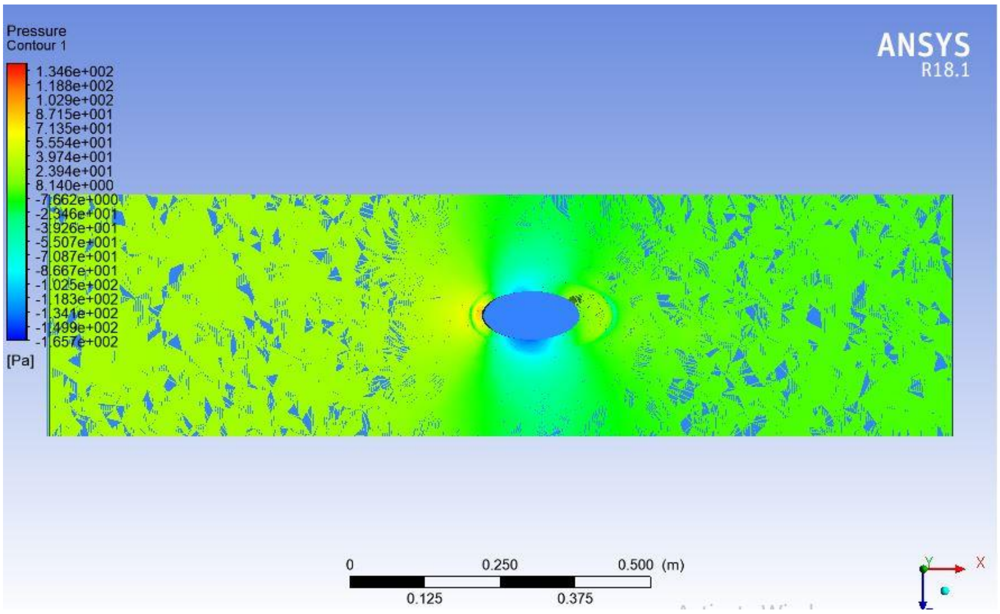

Figure 4.13 Pressure Contour on Sand Bed (Elliptical Pier)

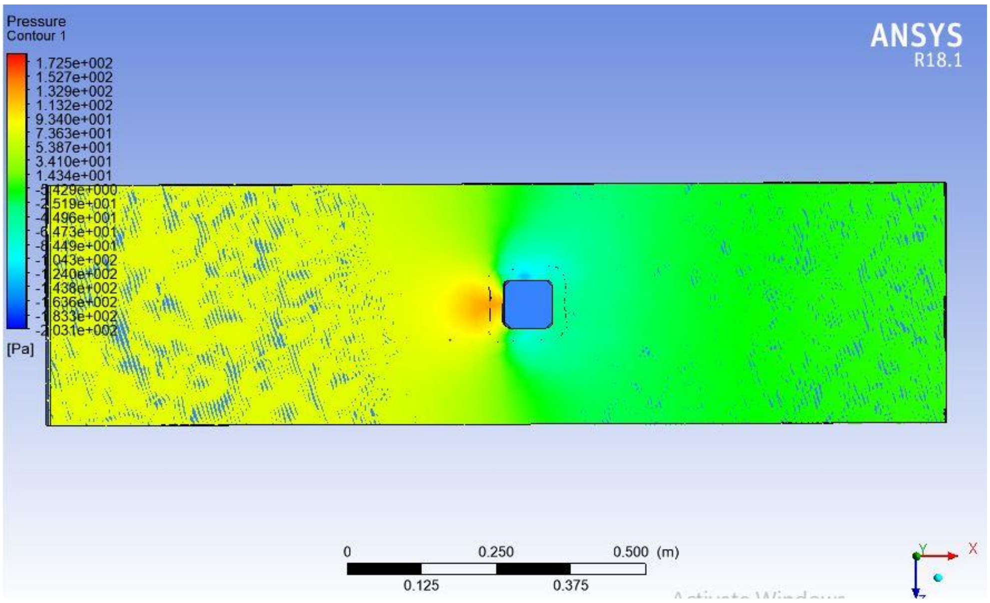

Figure 4.14 Pressure Contour on Sand Bed (Square Pier)

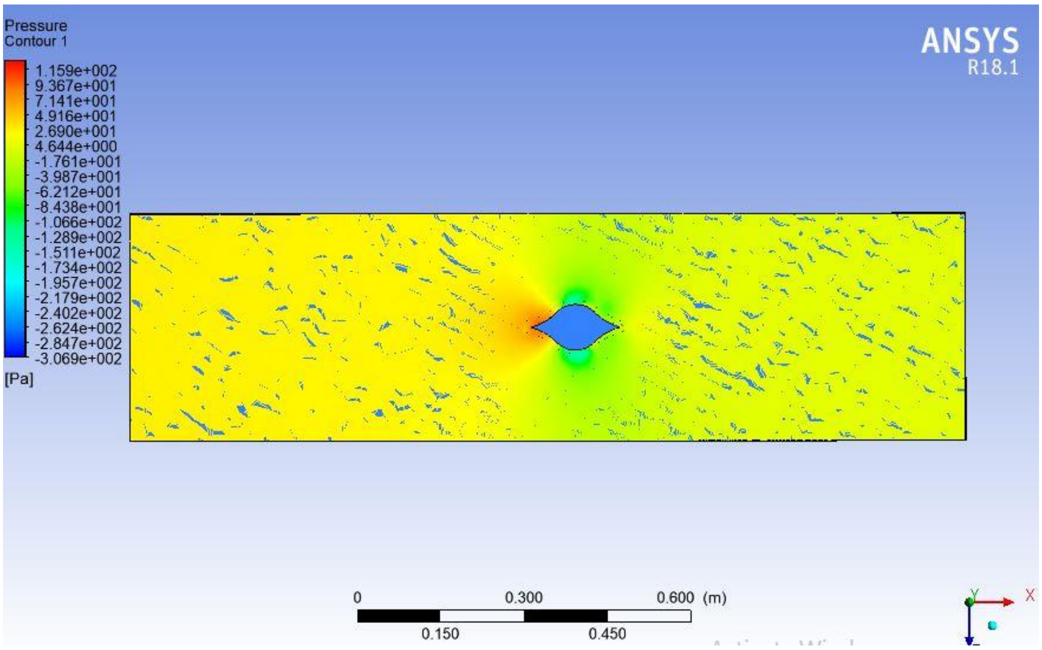

Figure 4.15 Pressure Contour on Sand Bed (Streamlined Pier)

The Graphs showing the pressure comparisons for 4 pier shapes on longitudinal and lateral lines on the sand bed are shown below.

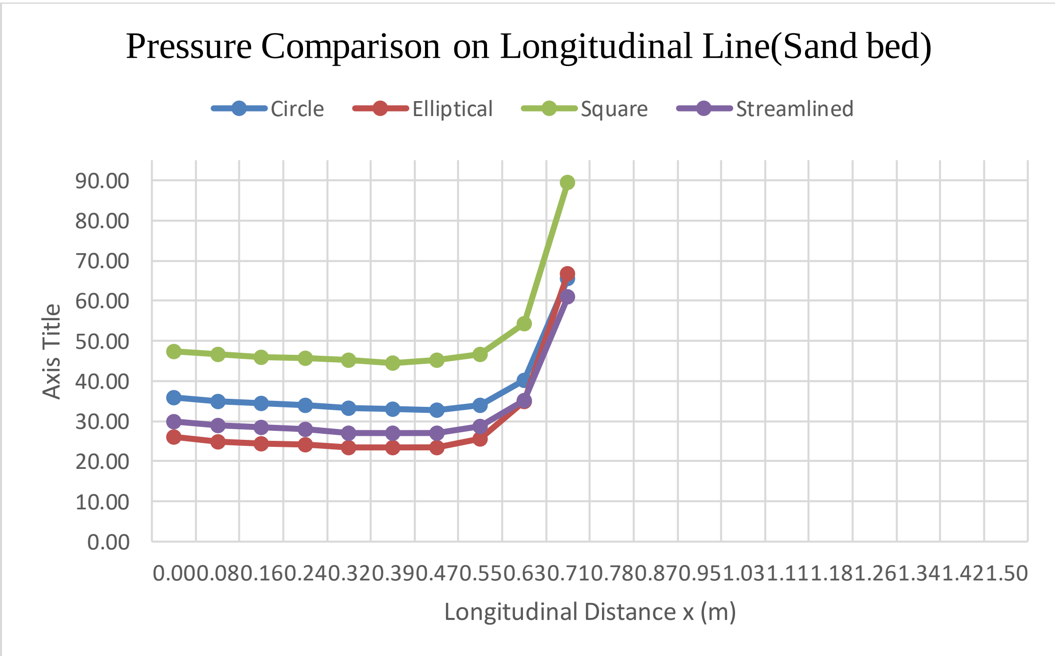

Figure 4.16 Pressure Comparison on Longitudinal line (Sand bed)

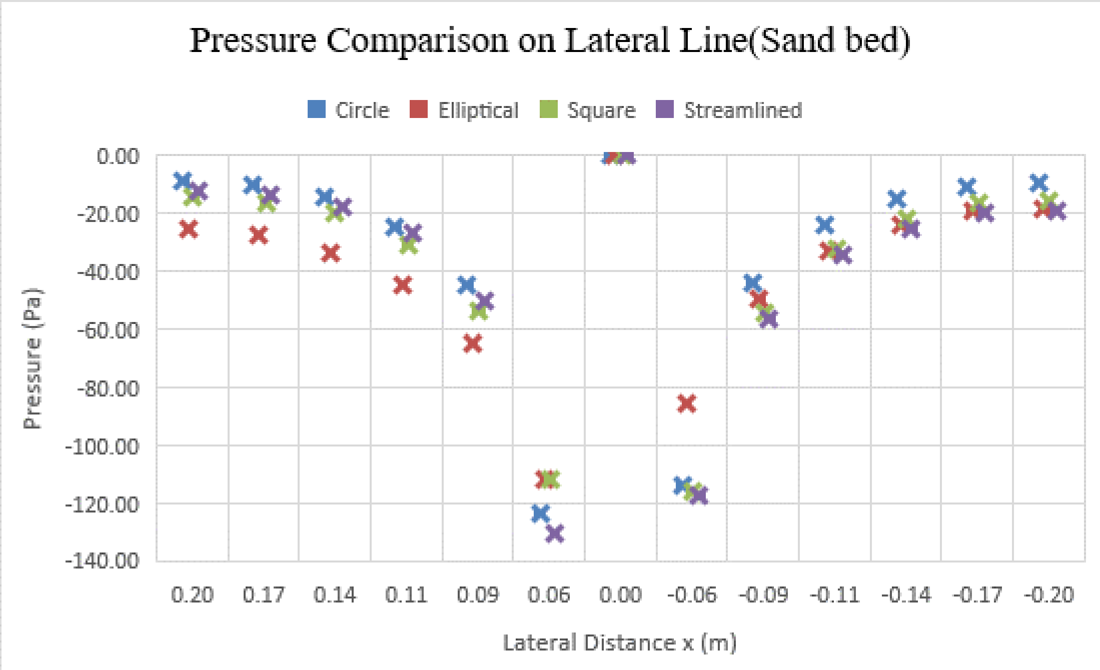

Figure 4.17 Pressure Comparison on Lateral line (Sand bed)

The table showing the pressure values on sand bed on longitudinal and lateral line is given below,

Table 4.4 Pressure Values on Longitudinal line (Sand bed)

Longitudinal Distance x (m)	Pressure (Pa)			
	Circular	Elliptical	Square	Streamlined
0.00	35.96	26.06	47.51	29.99
0.08	34.88	24.86	46.60	28.99
0.16	34.50	24.39	46.07	28.46
0.24	33.94	24.14	45.67	27.91
0.32	33.38	23.55	45.12	27.09
0.39	33.14	23.54	44.46	27.03
0.47	32.74	23.48	45.34	27.14
0.55	34.06	25.53	46.56	28.65
0.63	40.18	34.95	54.34	35.15
0.71	65.56	66.87	89.59	60.97

Table 4.5 Pressure Values on Lateral line (Sand bed)

Lateral Distance z (m)	Pressure (Pa)			
	Circular	Elliptical	Square	Streamlined
0.20	-8.64	-25.70	-14.37	-12.34
0.17	-10.09	-27.65	-16.50	-13.50
0.14	-14.13	-33.82	-20.17	-18.15
0.11	-24.44	-44.82	-31.06	-26.51
0.09	-44.72	-65.15	-53.89	-50.66
0.06	-123.47	-112.17	-111.82	-130.49
0.00	0.00	0.00	0.00	0.00
-0.06	-113.77	-85.27	-115.76	-117.07
-0.09	-44.36	-49.76	-54.55	-56.29
-0.11	-24.38	-33.00	-32.57	-34.53
-0.14	-15.07	-24.27	-22.04	-25.21
-0.17	-10.99	-19.40	-16.37	-19.63
-0.20	-9.67	-18.43	-15.76	-19.30

4.1.2.3 Pressure Contour on Longitudinal Plane

The Pressure contours on the longitudinal plane for the four piers are shown below. The contour results indicate that the pressure near the square pier in maximum and minimum in case of streamlined pier. Larger values near the pier indicates more obstruction to the flow velocities resulting in more local scouring near the pier toe.

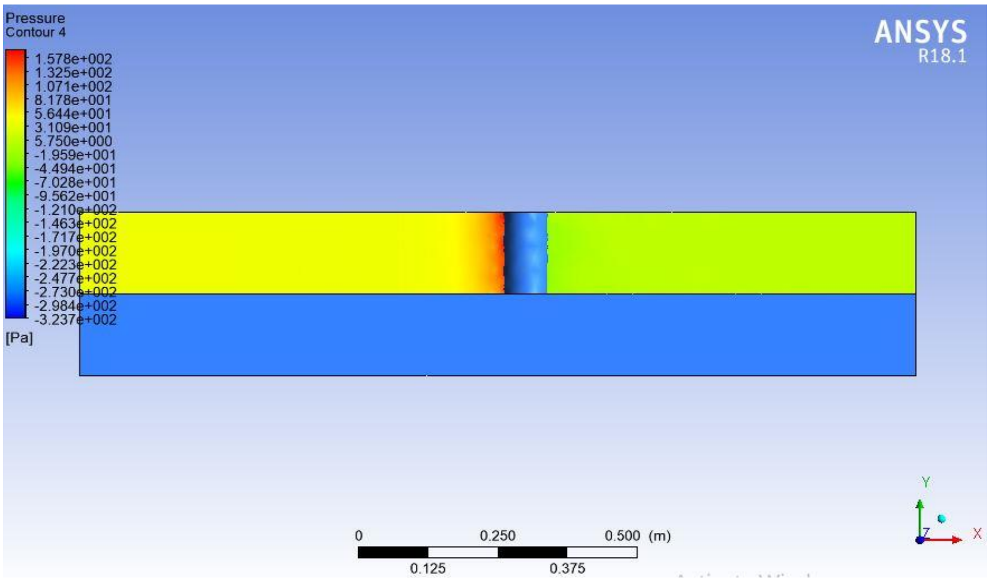

Figure 4.18 Pressure Contour on Longitudinal Plane (Circular Pier)

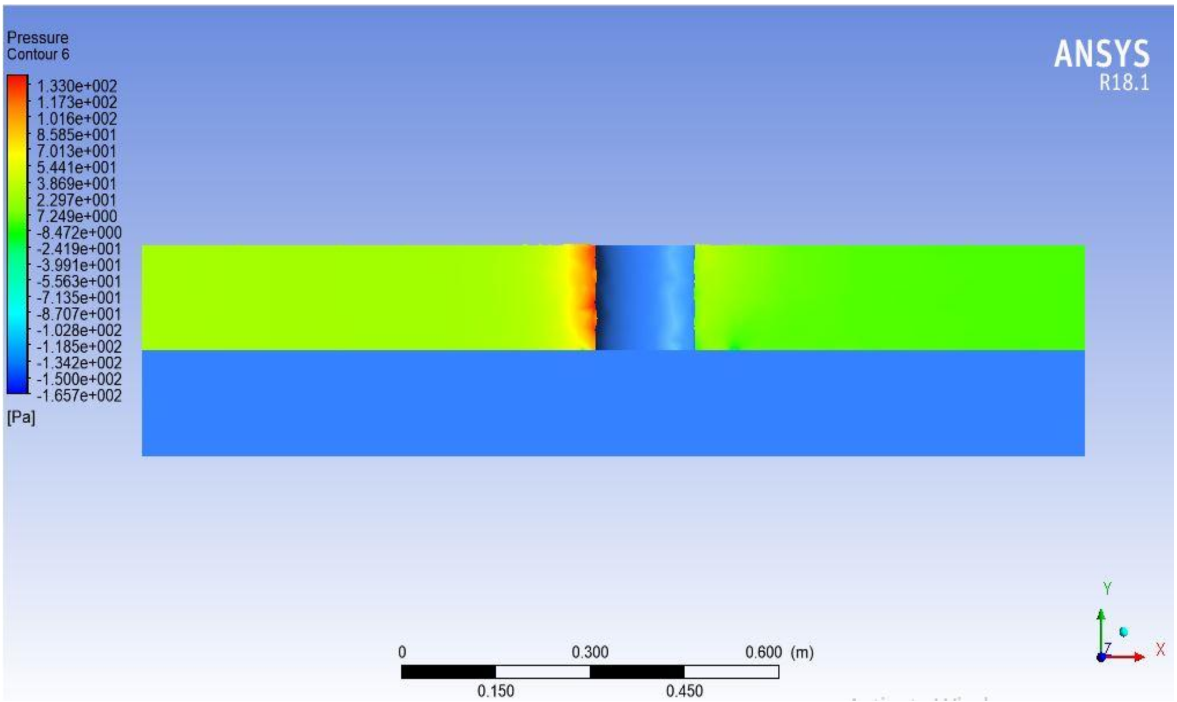

Figure 4.19 Pressure Contour on Longitudinal Plane (Elliptical Pier)

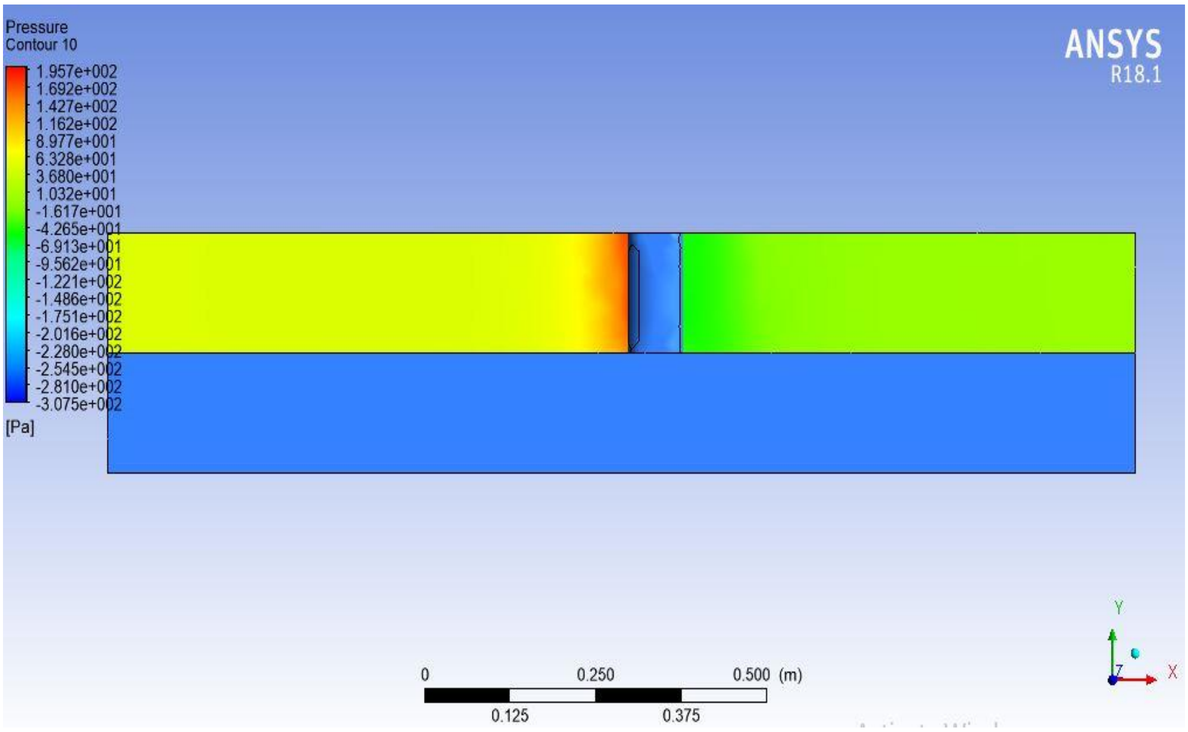

Figure 4.20 Pressure Contour on Longitudinal Plane (Square Pier)

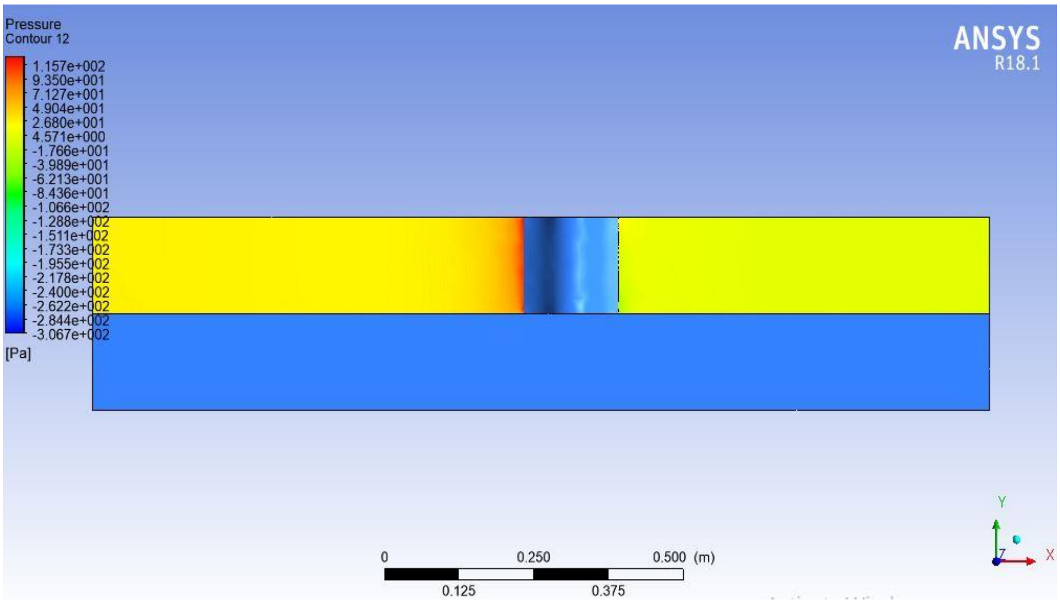

Figure 4.21 Pressure Contour on Longitudinal Plane (Streamlined Pier)

4.1.3 Shear Stress Distribution

When the flow field gets passed around the pier, it gets obstructed by the pier wall resulting in the increase in shear stress near the toe of pier . The shear stress when exceeds the threshold value, local scouring gets induced around the pier. Shear stress distribution is shown on the pier wall and sand bed. The region where shear stress value is more are prone to local scouring. The shear stress values are also collected and compared on 3 lines Line 1, Line 2 and Line 3 on the pier wall.

4.1.3.1 Shear Stress on the Pier wall

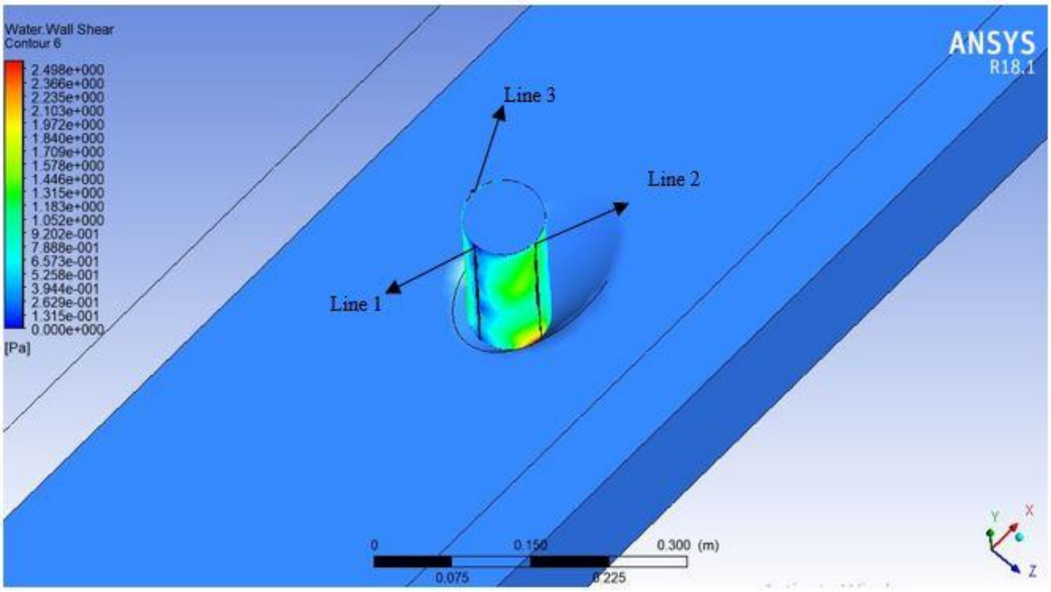

Figure 4.22 Shear Stress Contour on Circular Pier

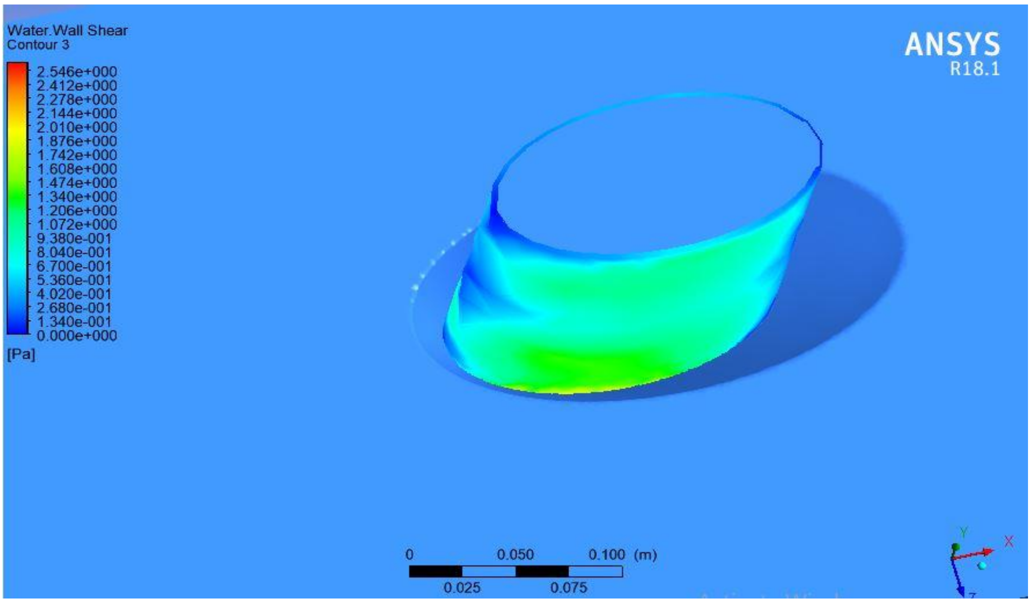

Figure 4.23 Shear Stress Contour on Elliptical Pier

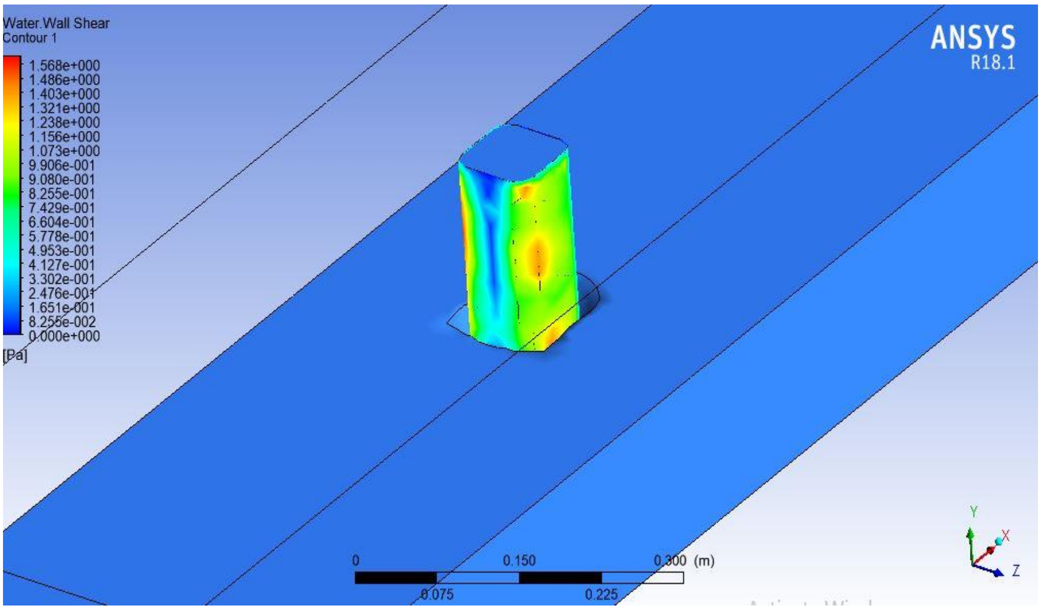

Figure 4.24 Shear Stress Contour on Square Pier

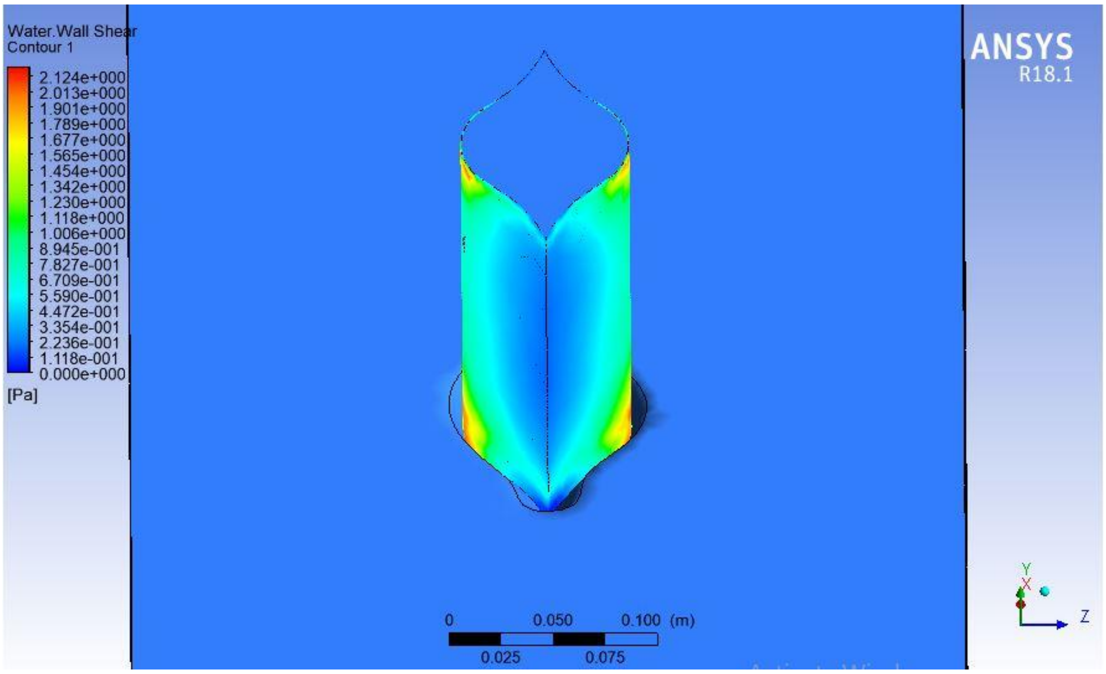

Figure 4.25 Shear Stress Contour on Streamlined Pier

The Graphs showing the shear stress comparisons for 4 pier shapes on line 1, line 2 and line 3 on the pier wall are shown below,

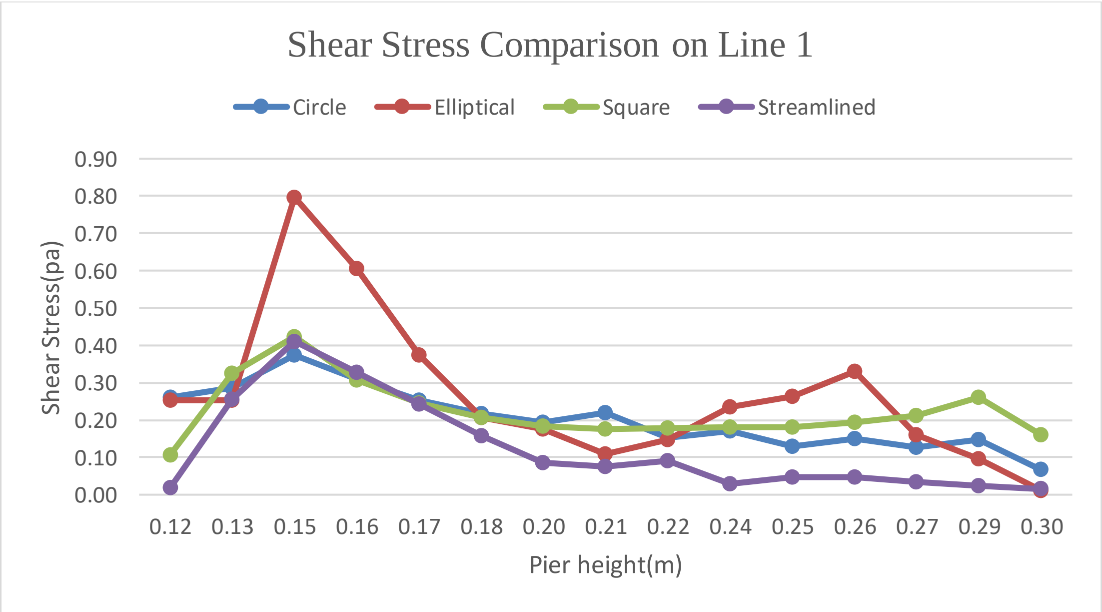

Figure 4.26 Shear Stress Comparison on Line 1

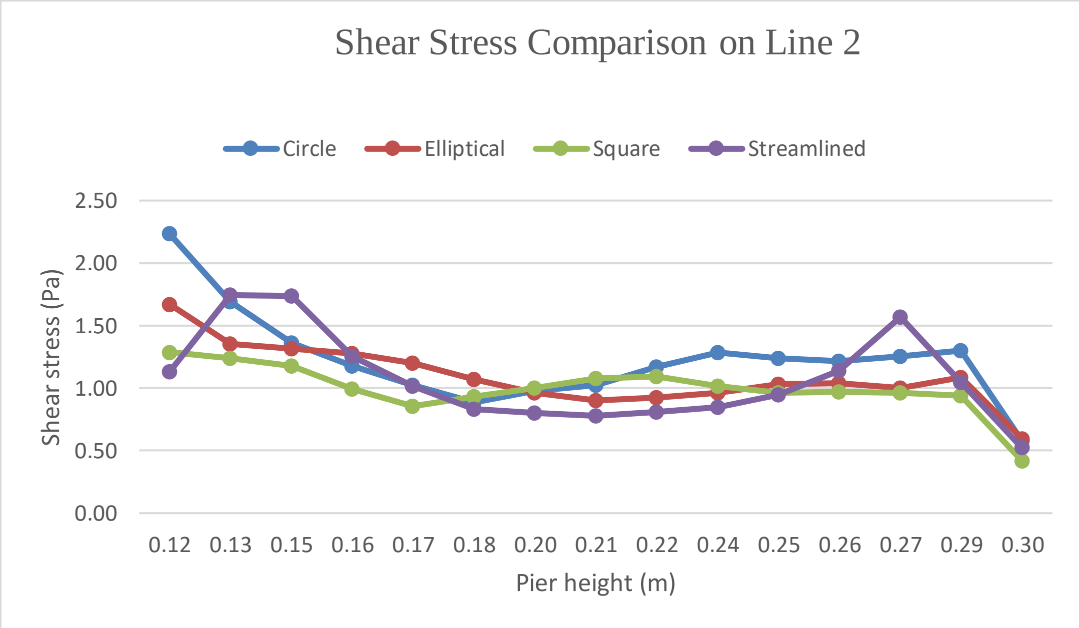

Figure 4.27 Shear Stress Comparison on Line 2

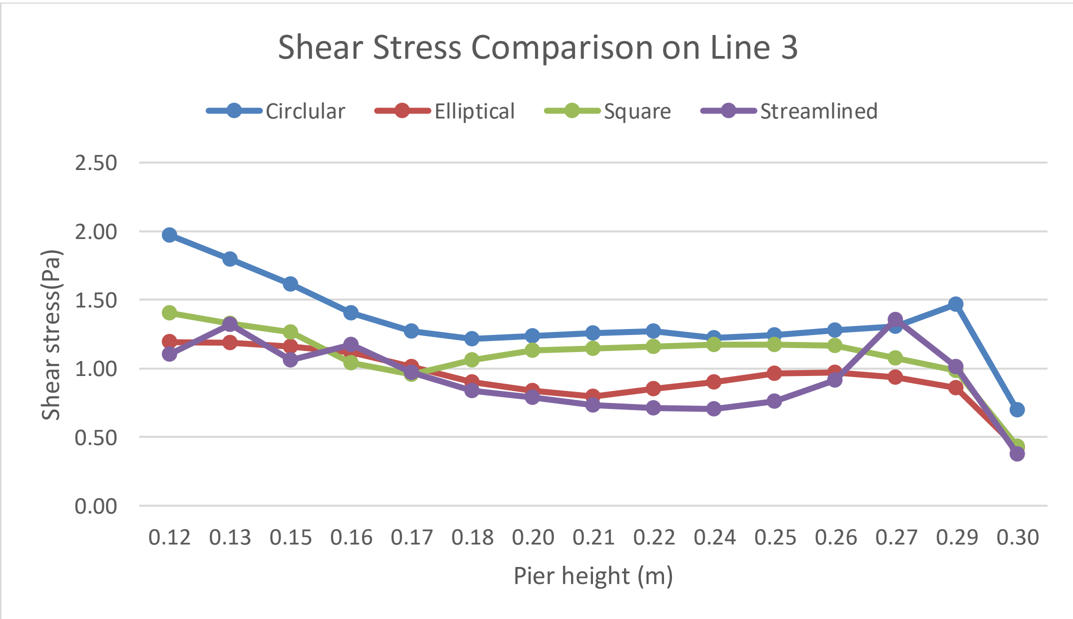

Figure 4.28 Shear Stress Comparison on Line 3

Table 4.6 Shear Stress values on Pier Wall on Line 1

Pier height (m)	Shear stress on Line 1			
	Circular	Elliptical	Square	Streamlined
0.12	1.97	0.25	0.11	0.02
0.13	1.80	0.25	0.33	0.26
0.15	1.62	0.80	0.42	0.41
0.16	1.40	0.60	0.31	0.33
0.17	1.27	0.37	0.24	0.24
0.18	1.22	0.21	0.21	0.16
0.20	1.24	0.17	0.18	0.09
0.21	1.26	0.11	0.17	0.08
0.22	1.27	0.15	0.18	0.09
0.24	1.22	0.24	0.18	0.03
0.25	1.24	0.26	0.18	0.05
0.26	1.28	0.33	0.19	0.05
0.27	1.31	0.16	0.21	0.04
0.29	1.47	0.09	0.26	0.03
0.30	0.70	0.01	0.16	0.02

Table 4.7 Shear Stress values on Pier Wall on Line 2

Pier height (m)	Shear stress on Line 2			
	Circular	Elliptical	Square	Streamlined
0.12	2.23	1.67	1.29	1.13
0.13	1.69	1.35	1.24	1.74
0.15	1.36	1.32	1.18	1.74
0.16	1.18	1.28	0.99	1.26
0.17	1.02	1.20	0.86	1.01
0.18	0.88	1.07	0.93	0.83
0.20	0.98	0.97	1.00	0.80
0.21	1.03	0.90	1.08	0.78
0.22	1.17	0.92	1.09	0.81
0.24	1.29	0.96	1.02	0.85
0.25	1.24	1.03	0.96	0.95

0.26	1.21	1.04	0.97	1.14
0.27	1.25	1.00	0.97	1.57
0.29	1.30	1.09	0.94	1.05
0.30	0.58	0.60	0.42	0.52

Table 4.8 Shear Stress values on Pier Wall on Line 3

Pier height (m)	Shear stress on Line 3			
	Circular	Elliptical	Square	Streamlined
0.12	1.97	1.19	1.40	1.10
0.13	1.80	1.19	1.33	1.32
0.15	1.62	1.16	1.27	1.06
0.16	1.40	1.12	1.04	1.17
0.17	1.27	1.02	0.96	0.97
0.18	1.22	0.90	1.06	0.84
0.20	1.24	0.83	1.13	0.79
0.21	1.26	0.80	1.15	0.73
0.22	1.27	0.85	1.16	0.71
0.24	1.22	0.90	1.17	0.70
0.25	1.24	0.96	1.17	0.76
0.26	1.28	0.97	1.17	0.91
0.27	1.31	0.94	1.08	1.36
0.29	1.47	0.86	0.98	1.01
0.30	0.70	0.42	0.43	0.38

4.1.3.2 Shear Stress on the Sand Bed

The velocity field when passing around the pier gets obstructed by the pier resulting in the rise of shear stress in the region near to the pier toe. When the value of shear stress exceeds threshold value , local scouring begins. The results shown that the shear stress value is minimum in case of streamlined pear as compared to other piers while on the other hand , it is maximum in case of square pier and circular pier. This shows that streamlined is the most efficient and less obstructed shape that can be used for pier geometry.

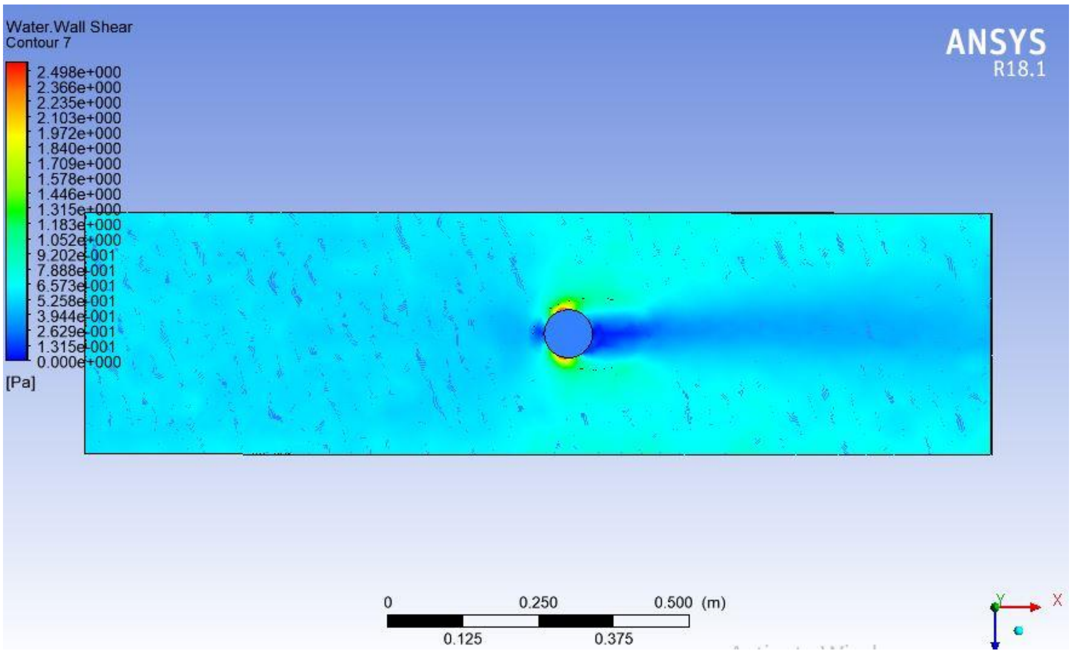

Figure 4.29 Shear stress on sand bed (Circular Pier)

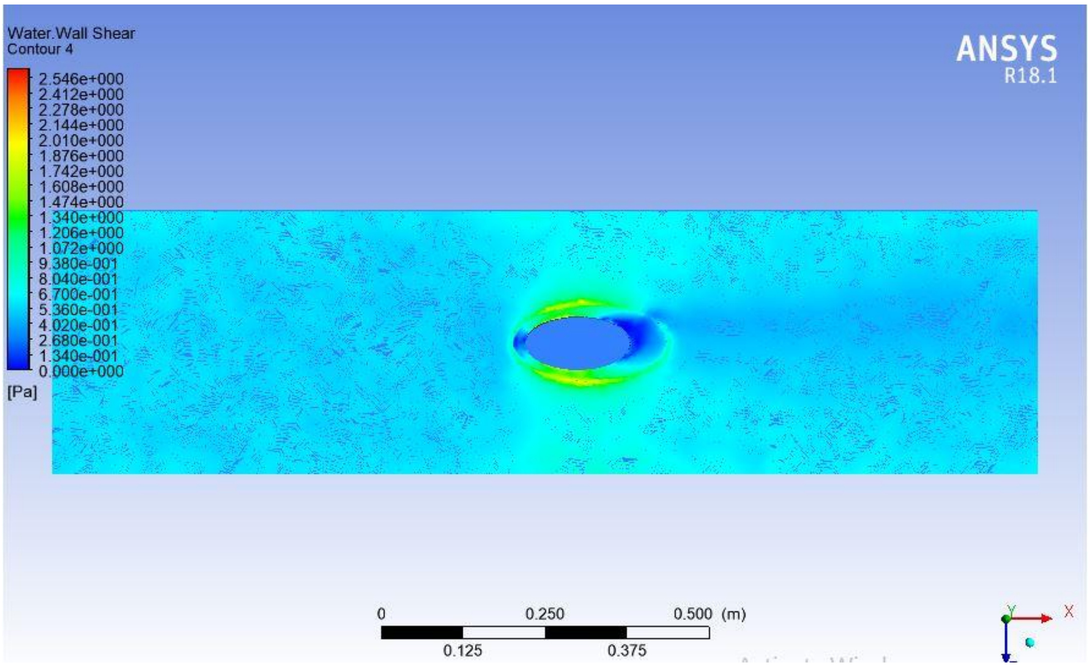

Figure 4.30 Shear stress on sand bed (Elliptical Pier)

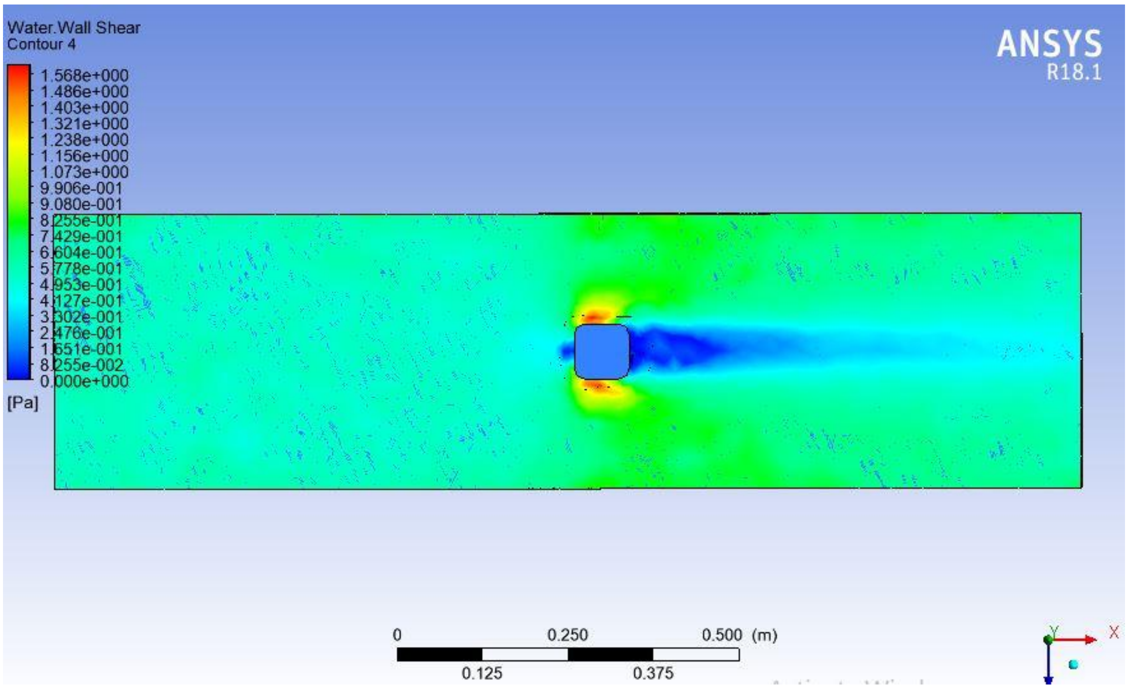

Figure 4.31 Shear stress on sand bed (Square Pier)

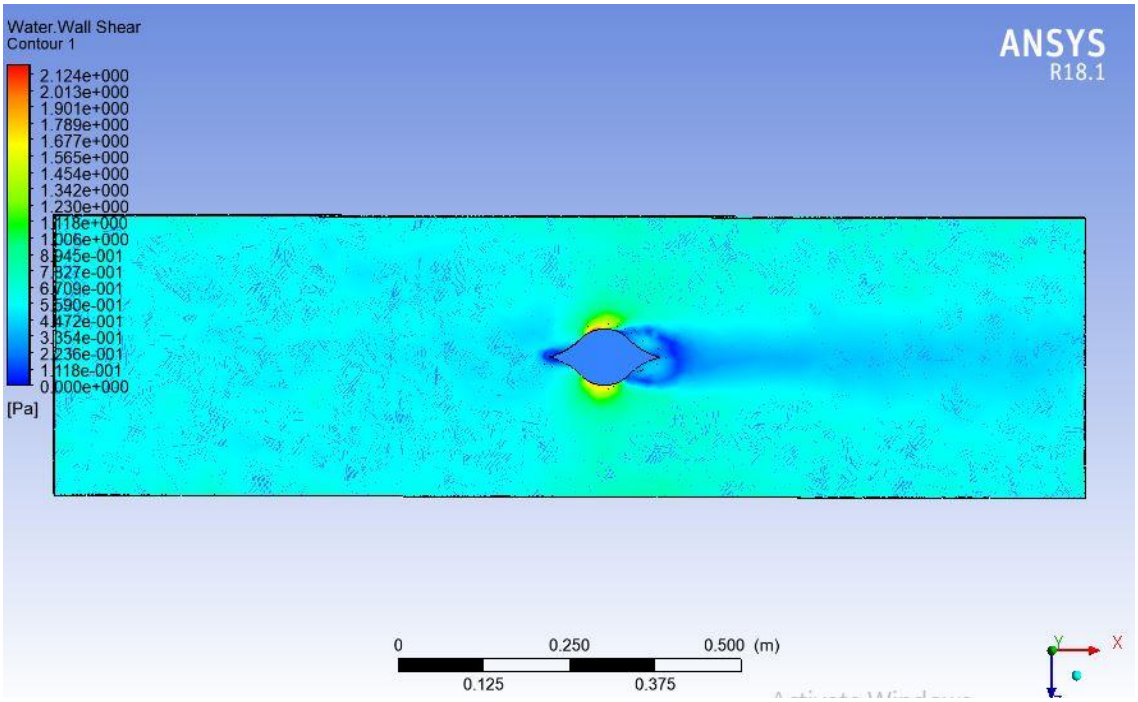

Figure 4.32 Shear stress on sand bed (Streamlined Pier)

The graphs showing the comparison of shear stress on longitudinal line on the sand bed is shown below,

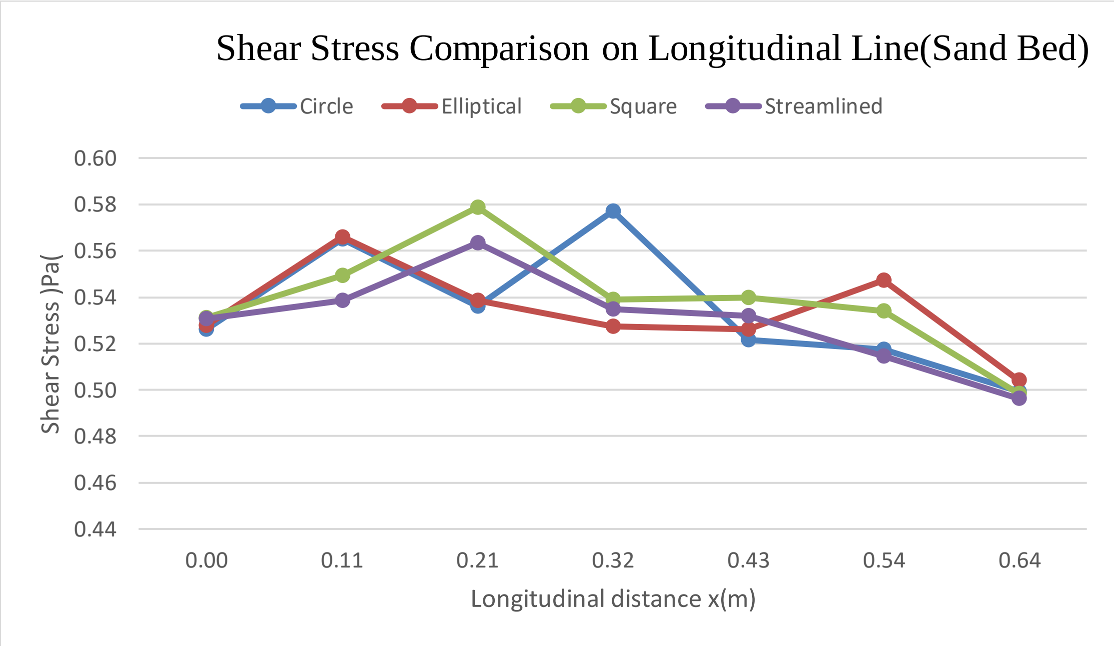

Figure 4.33 Shear Stress Comparison on Longitudinal Line (Sand Bed)

Table 4.9 Shear Stress Values on Longitudinal line(Sand bed)

Longitudinal Distance x (m)	**Shear Stress Value(Pa)**			
	Circular	**Elliptical**	**Square**	**Streamlined**
0.00	0.53	0.53	0.53	0.53
0.11	0.57	0.57	0.55	0.54
0.21	0.54	0.54	0.58	0.56
0.32	0.58	0.53	0.54	0.53
0.43	0.52	0.53	0.54	0.53
0.54	0.52	0.55	0.53	0.51
0.64	0.50	0.51	0.54	0.49

4.1.4 Velocity Distribution in Flow Domain

The turbulent flow around the pier gets deflected by the the pier wall which leads to the formation of horse shoe vortex on the upstream side of the pier . Horse shoe vortex is the first stage of beginning of scouring process. Streamlined pier shows negligible or very less chances of formation of horse shoe vortex where as piers like circular pier shows horse shoe vortex in front of pier wall.

4.1.4.1 Velocity Vectors

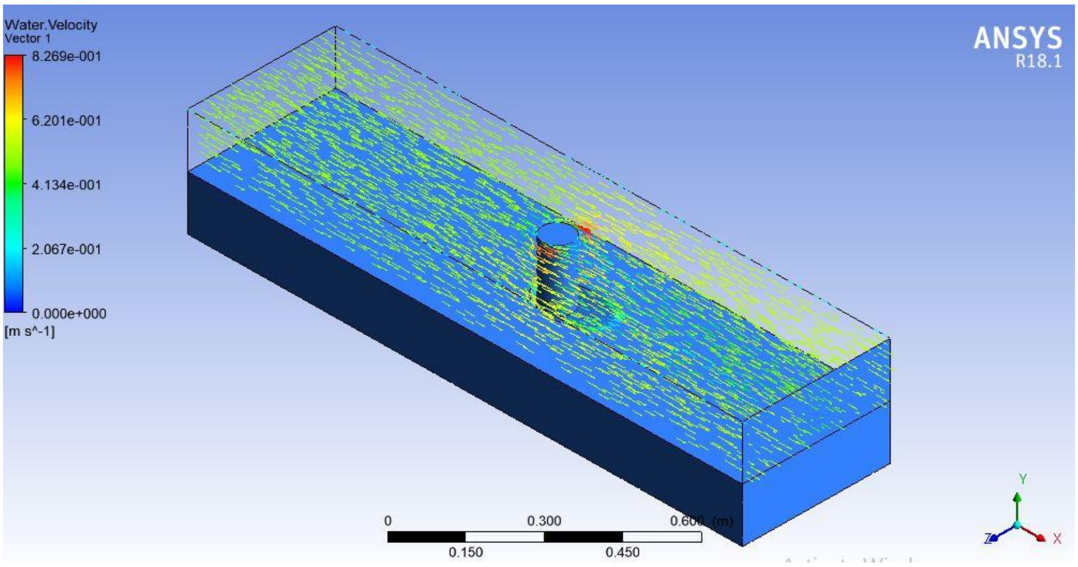

Figure 4.34 Water Velocity Vectors (Circular Pier)

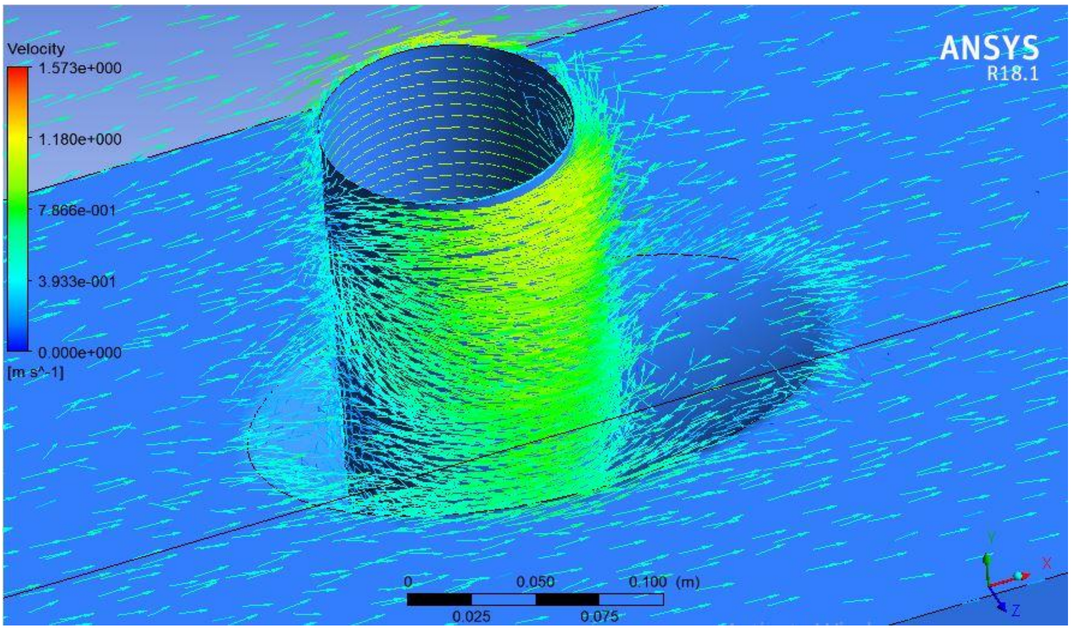

Figure 4.35 Horse Shoe Vortex (Circular Pier)

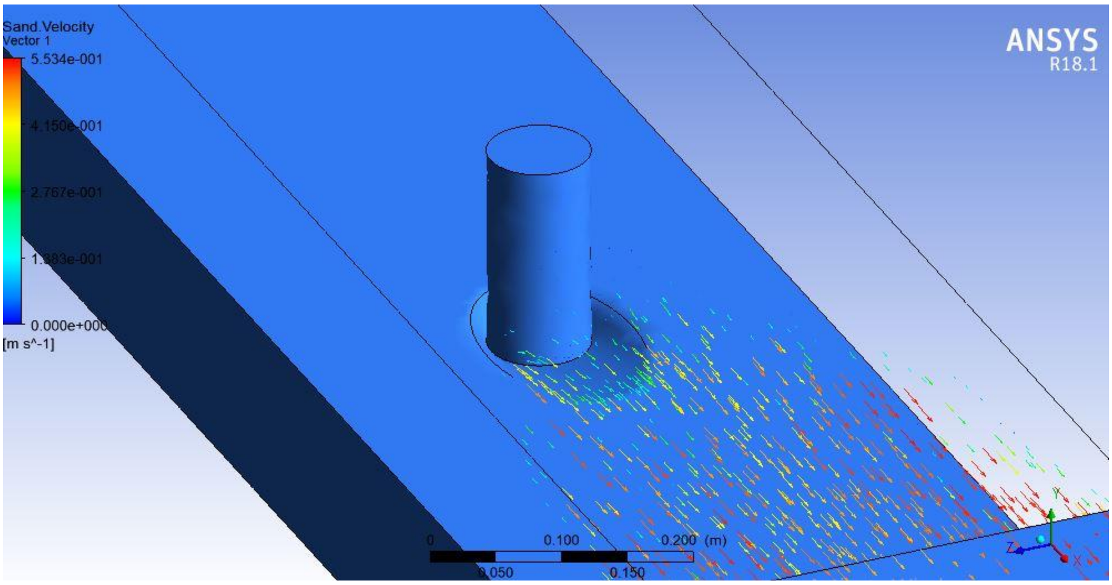

Figure 4.36 Sand Velocity Vectors (Circular Pier)

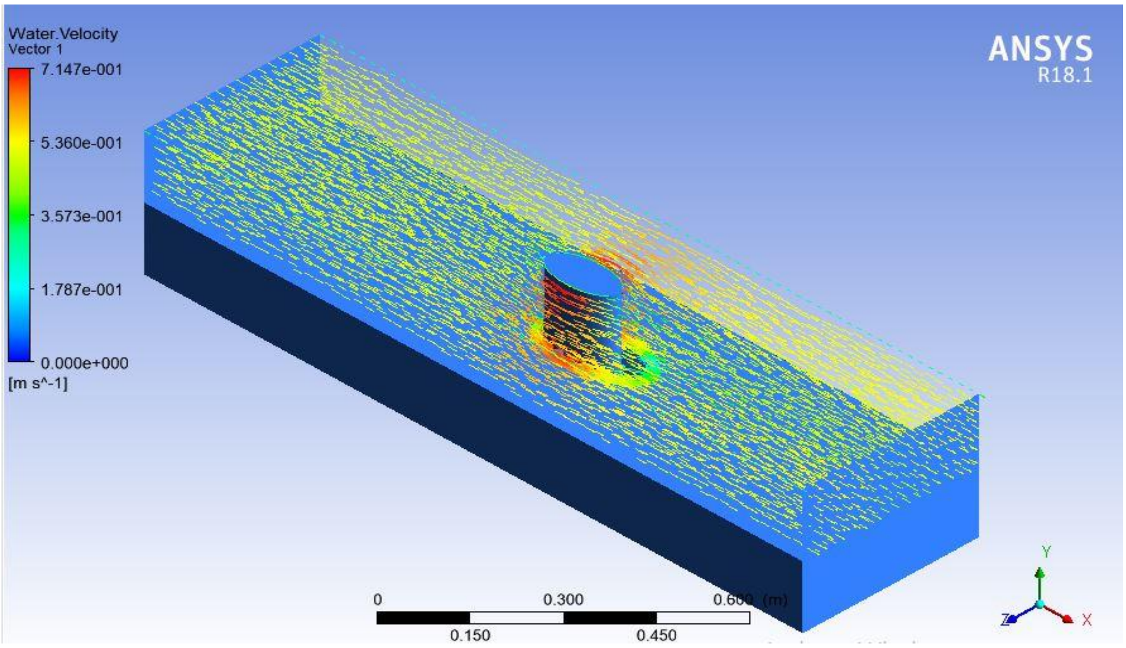

Figure 4.37 Water Velocity Vectors (Elliptical Pier)

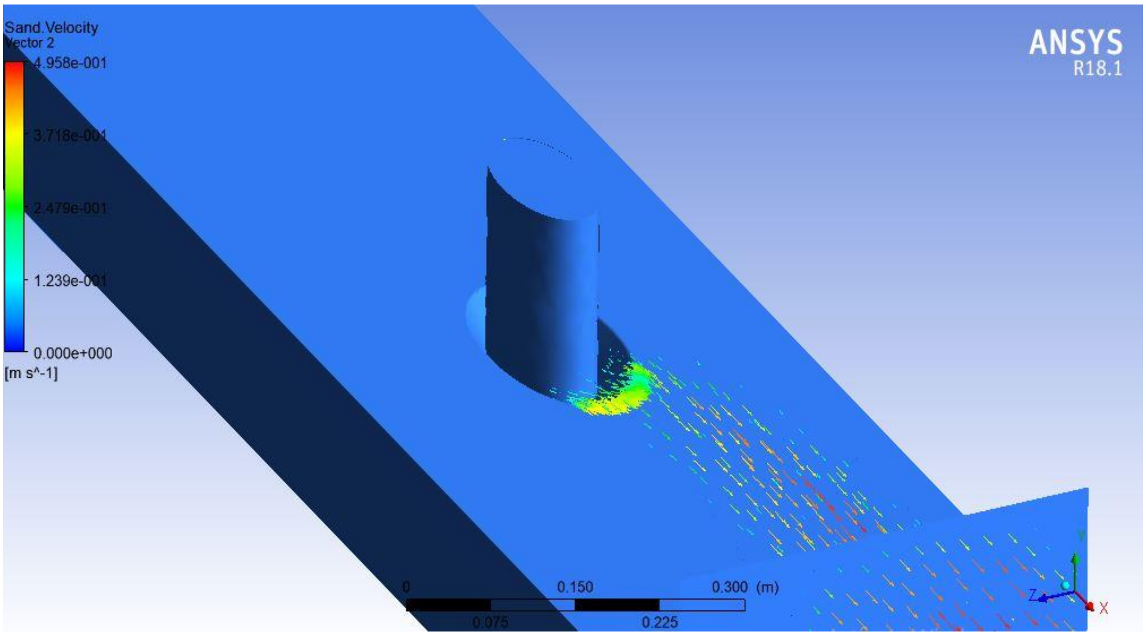

Figure 4.38 Sand Velocity Vectors (Elliptical Pier)

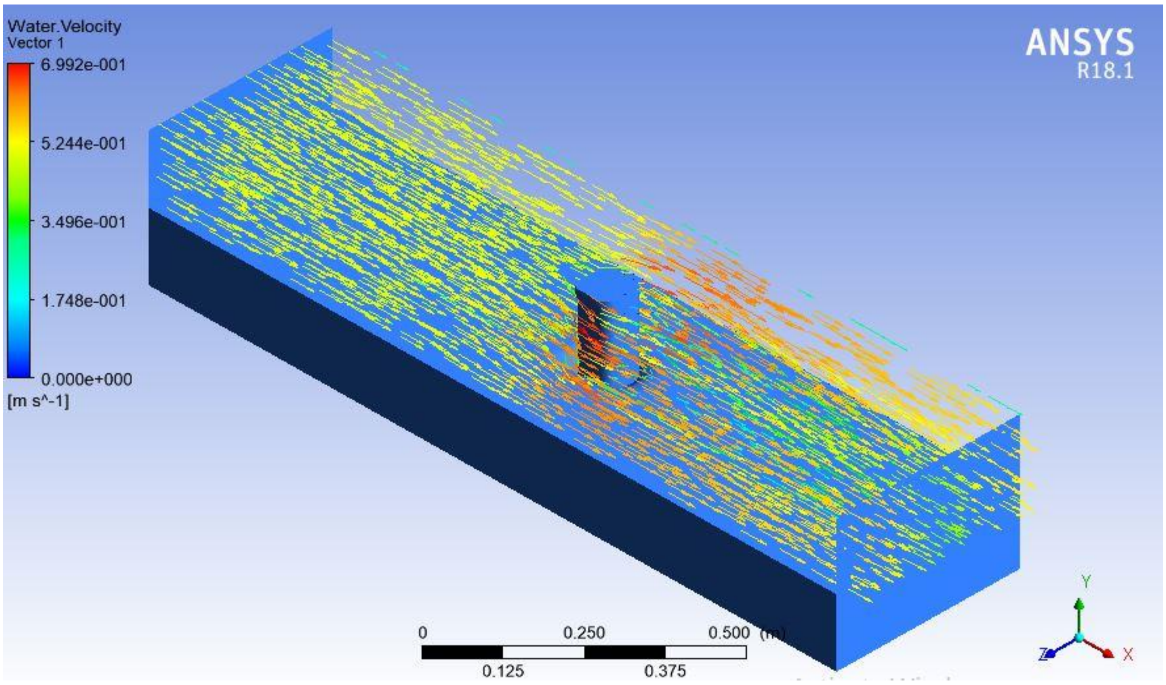

Figure 4.39 Water Velocity Vectors (Square Pier)

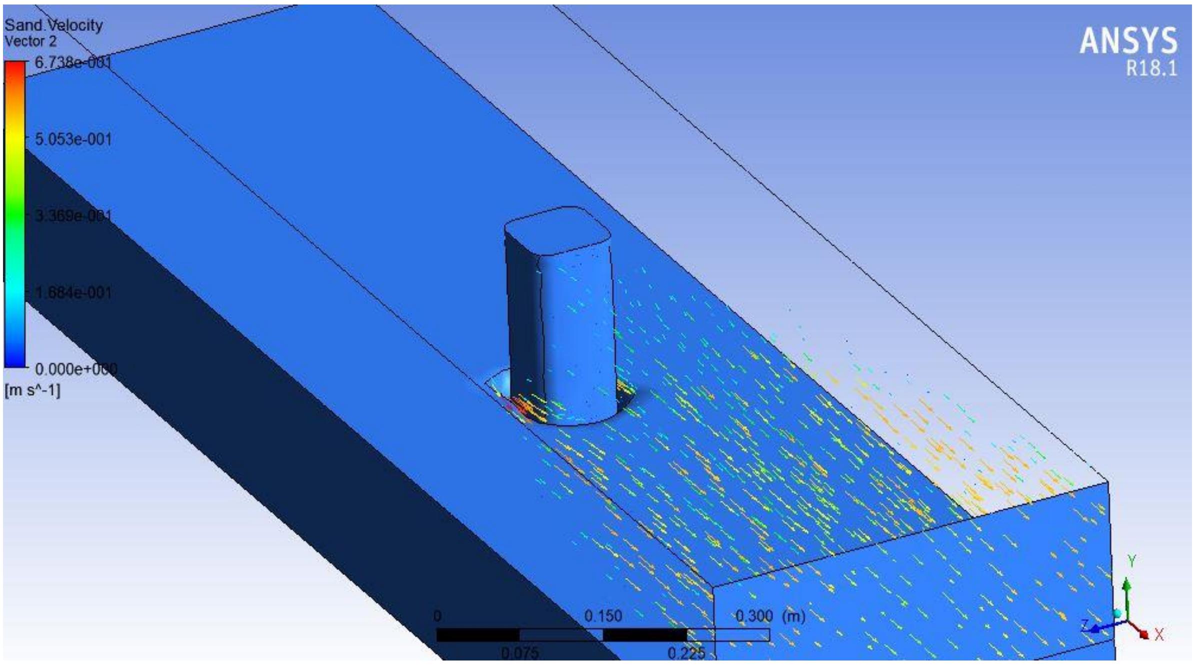

Figure 4.40 Sand Velocity Vectors (Elliptical Pier)

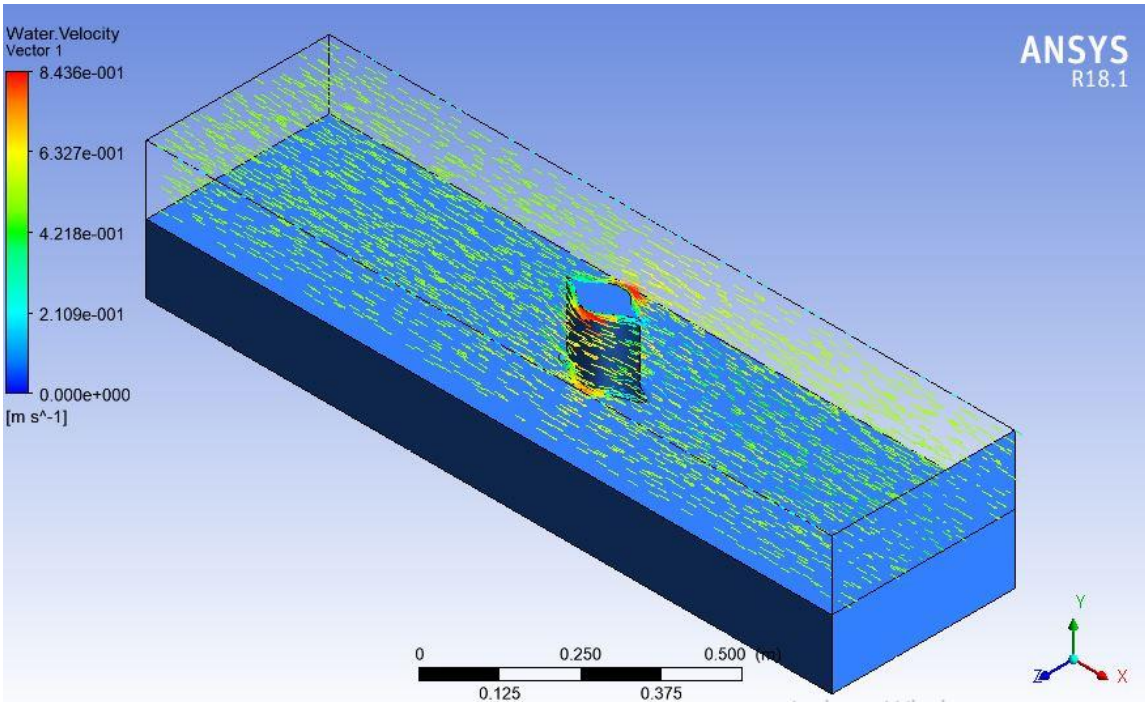

Figure 4.41 Water Velocity Vectors (Streamlined Pier)

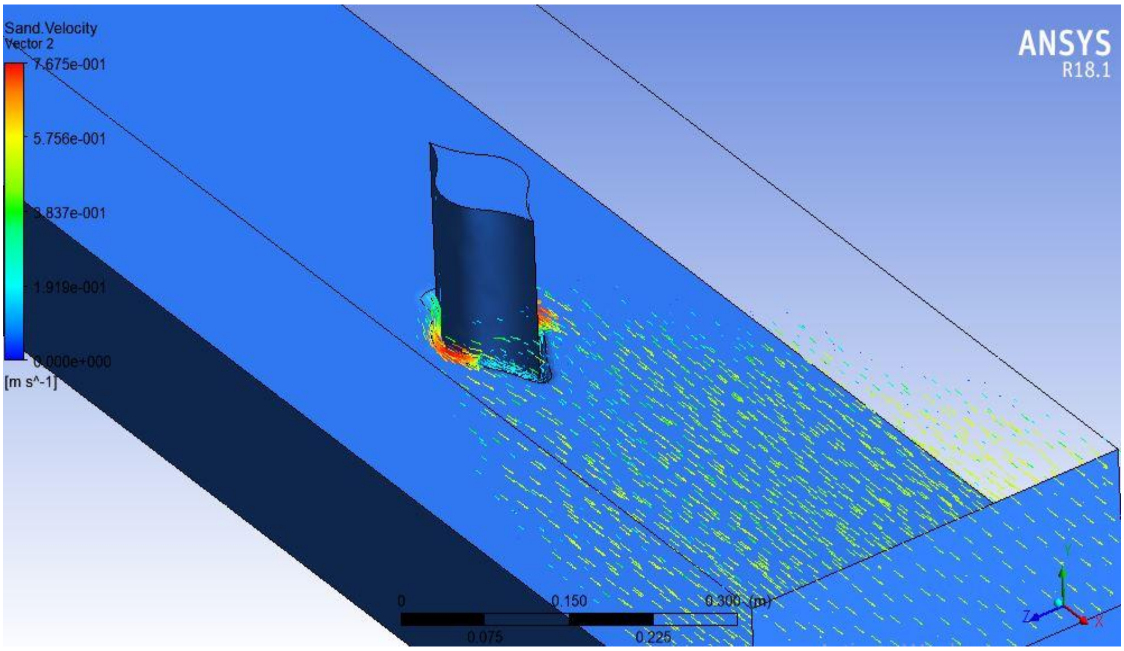

Figure 4.42 Sand Velocity Vectors (Streamlined Pier)

4.1.4.2 Velocity Streamlines

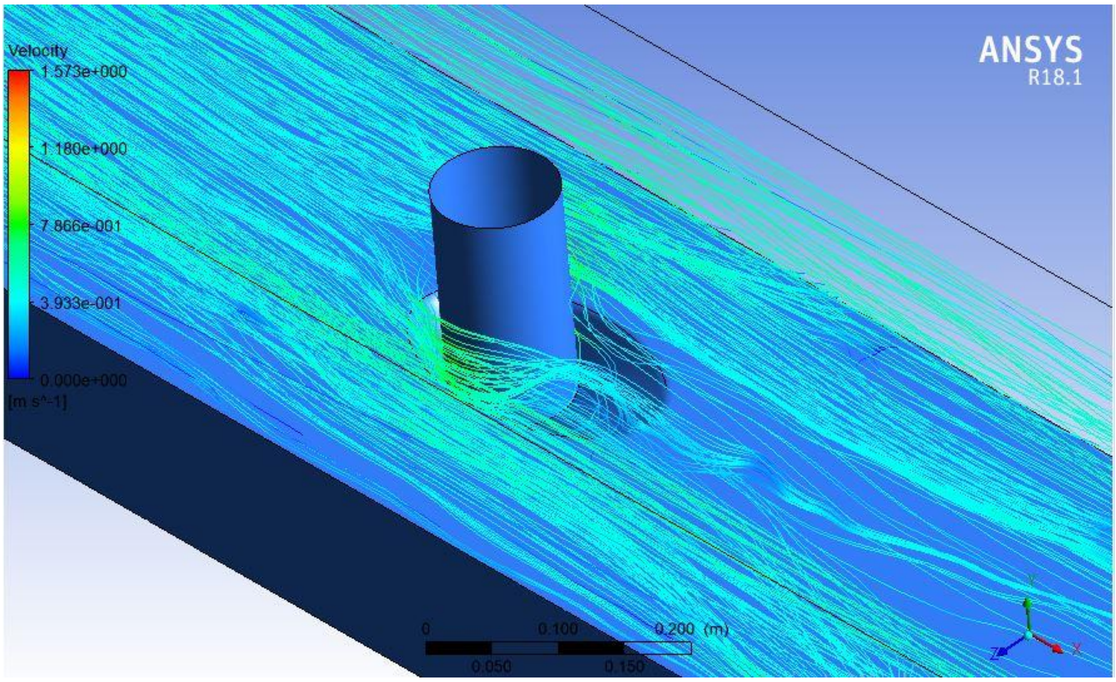

Figure 4.43 Water Velocity Streamlines (Circular Pier)

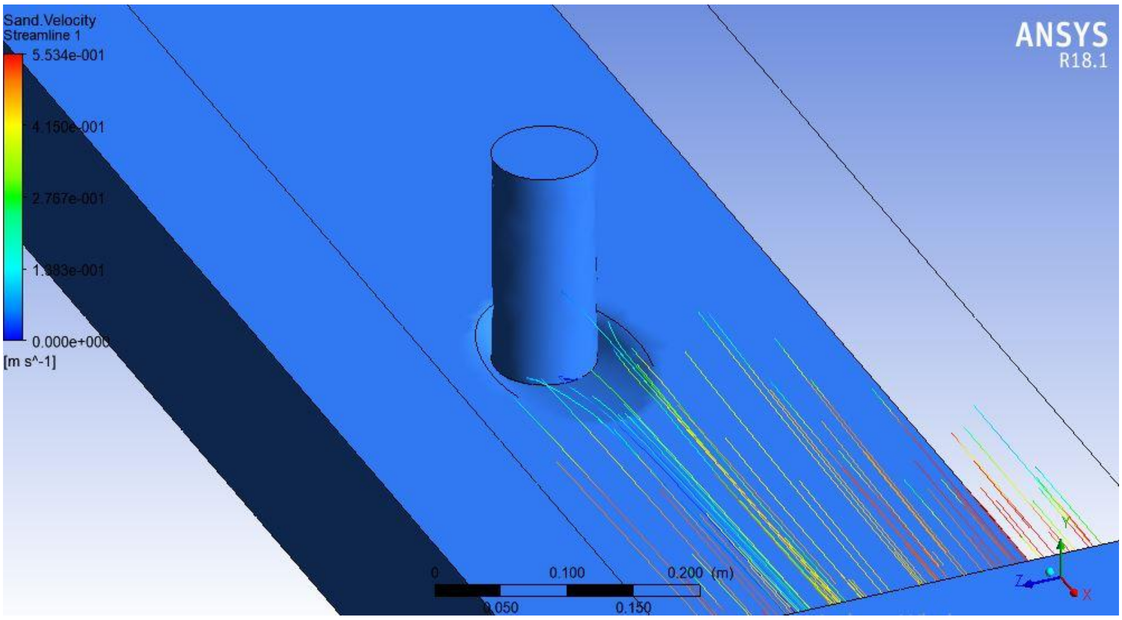

Figure 4.44 Sand Velocity Streamlines (Circular Pier)

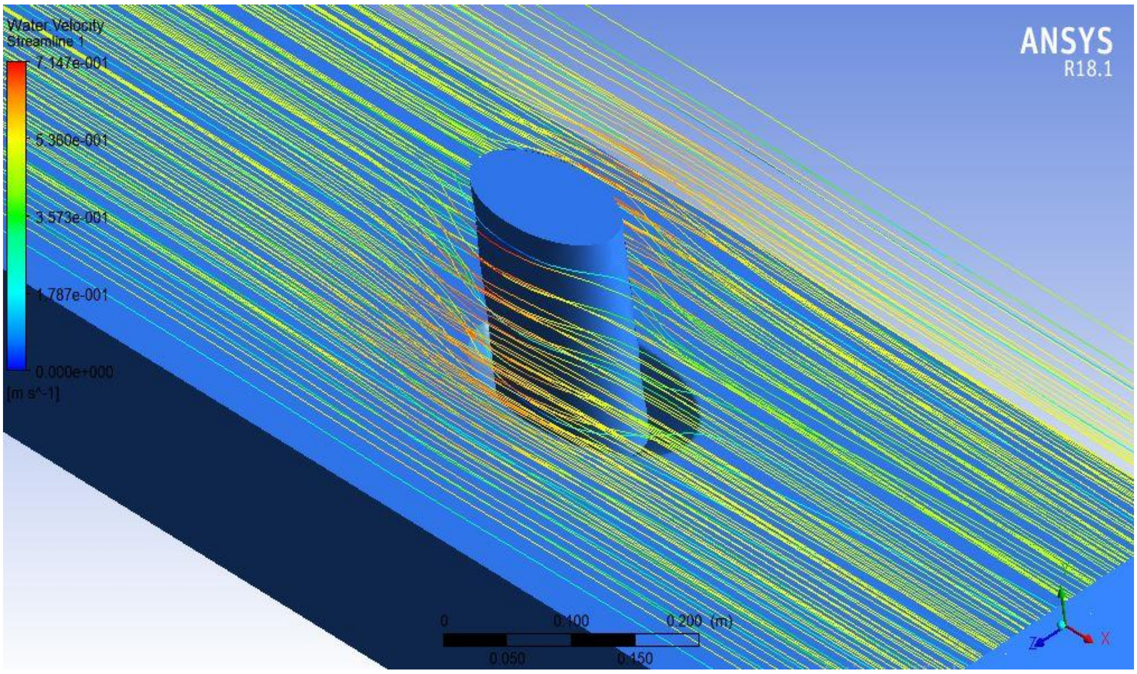

Figure 4.45 Water Velocity Streamlines (Elliptical Pier)

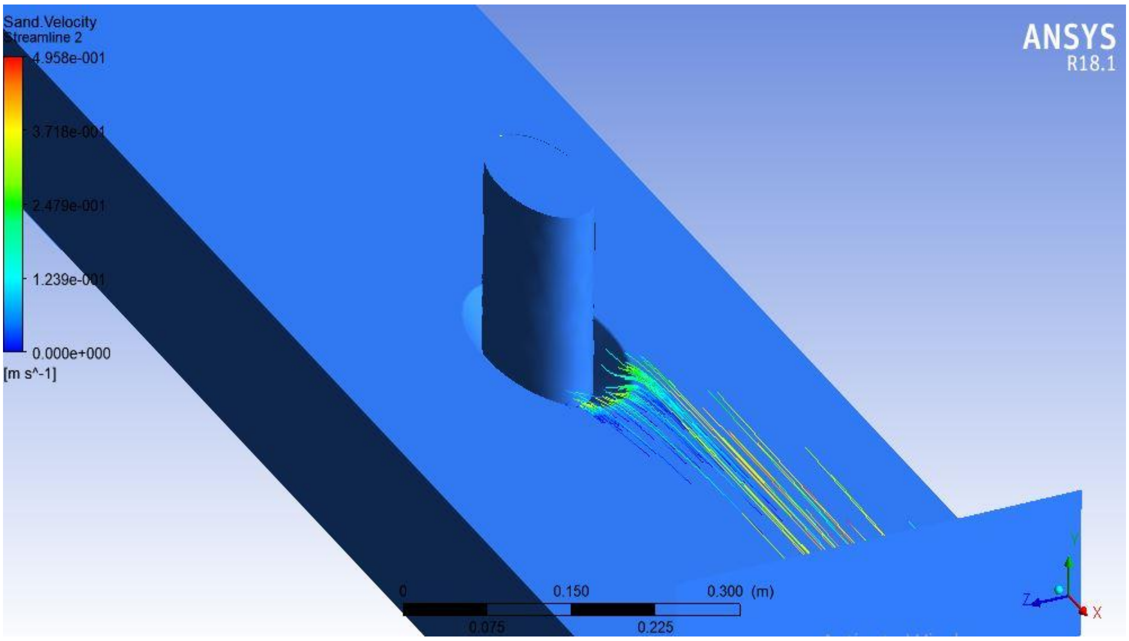

Figure 4.46 Sand Velocity Streamlines (Elliptical Pier)

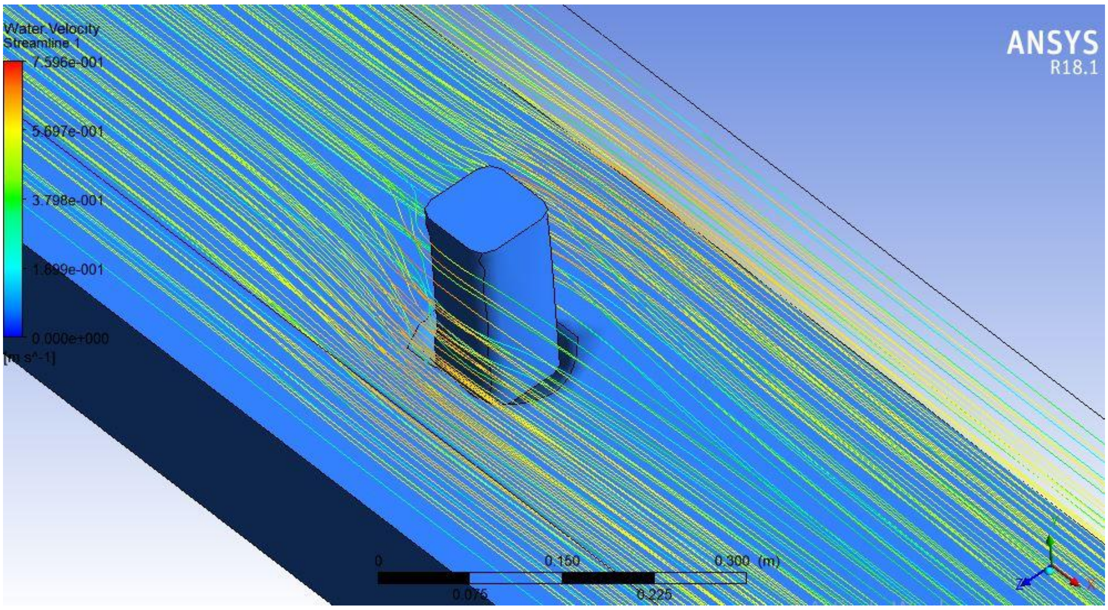

Figure 4.47 Water Velocity Streamlines (Square Pier)

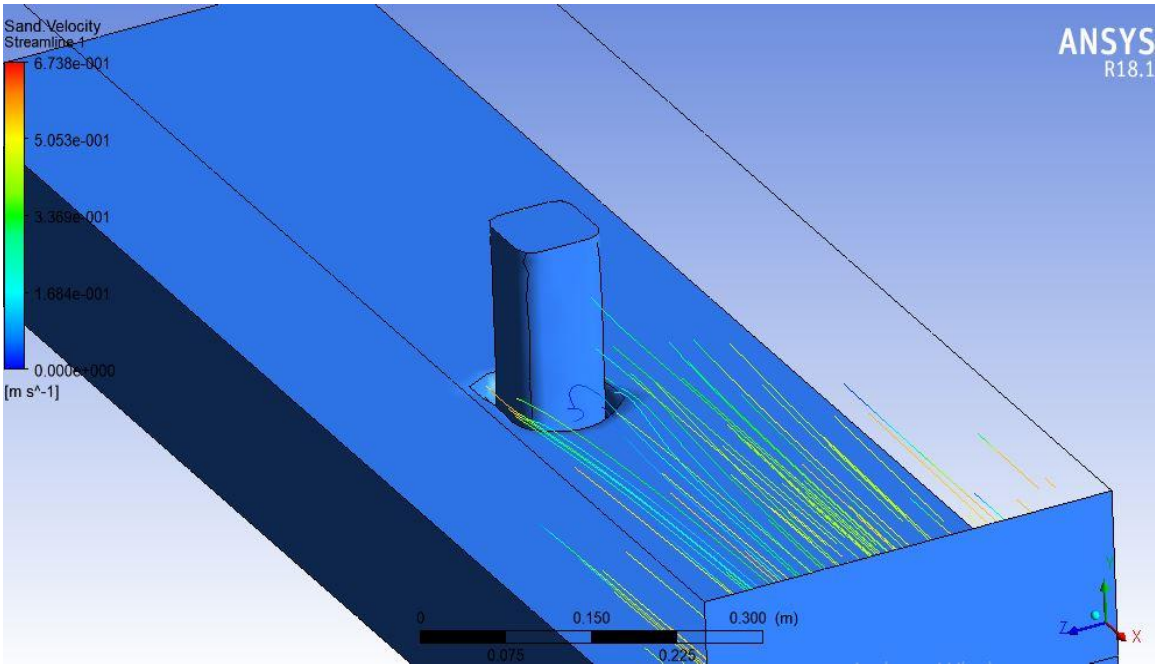

Figure 4.48 Sand Velocity Streamlines (Square Pier)

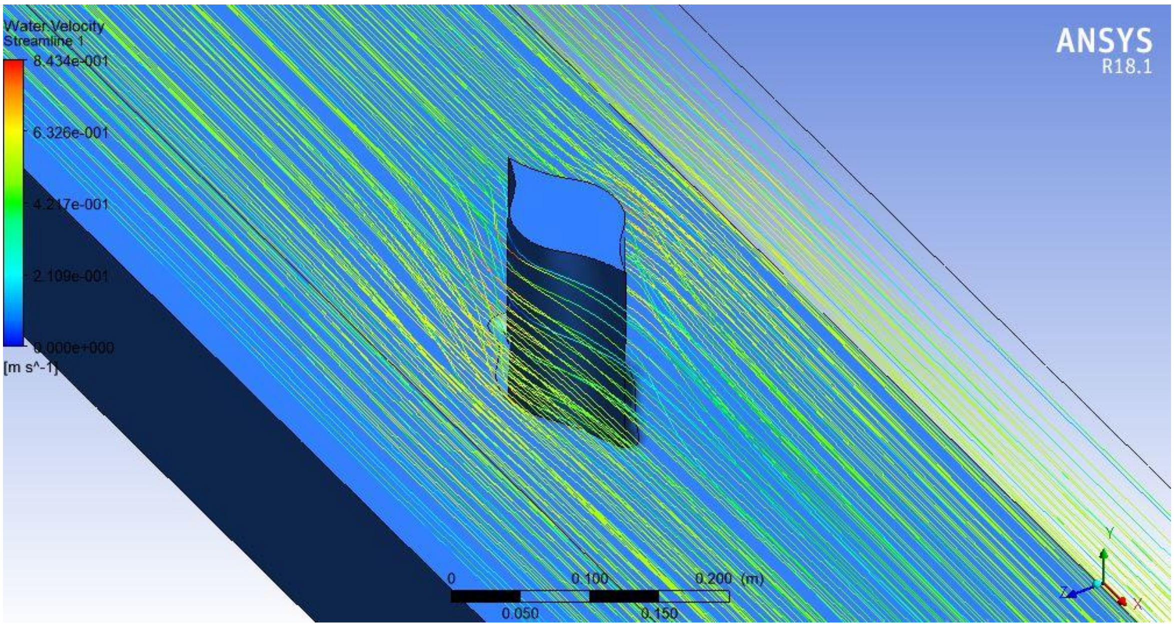

Figure 4.49 Water Velocity Streamlines (Streamlined Pier)

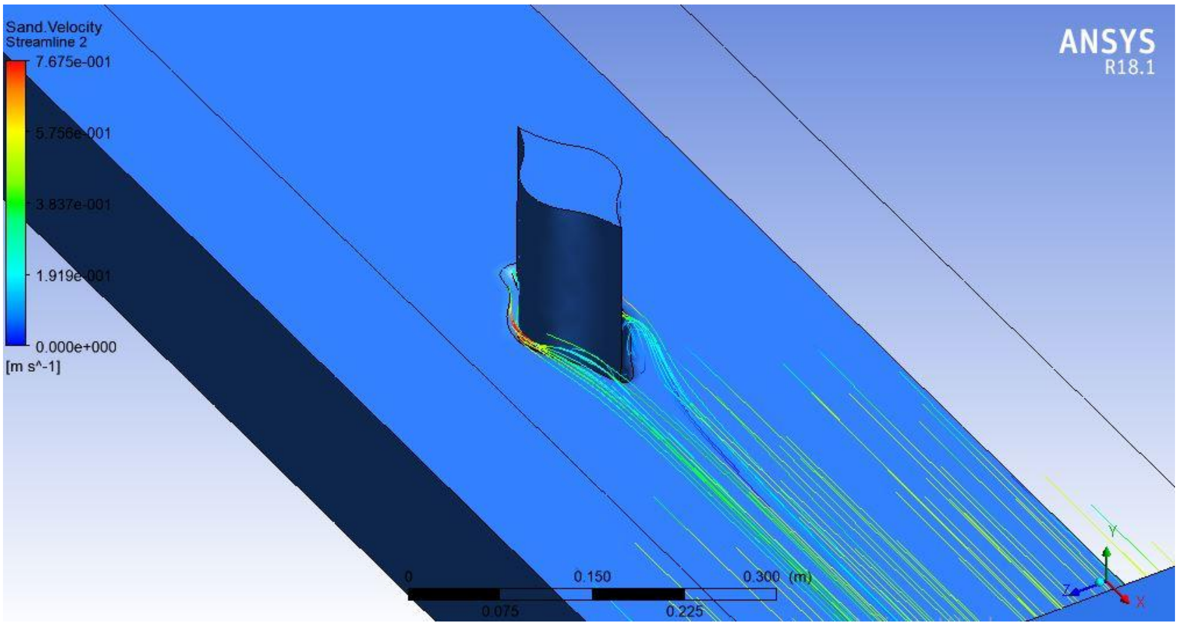

Figure 4.50 Sand Velocity Streamlines (Streamlined Pier)

4.1.5 Turbulence Kinetic Energy Distribution

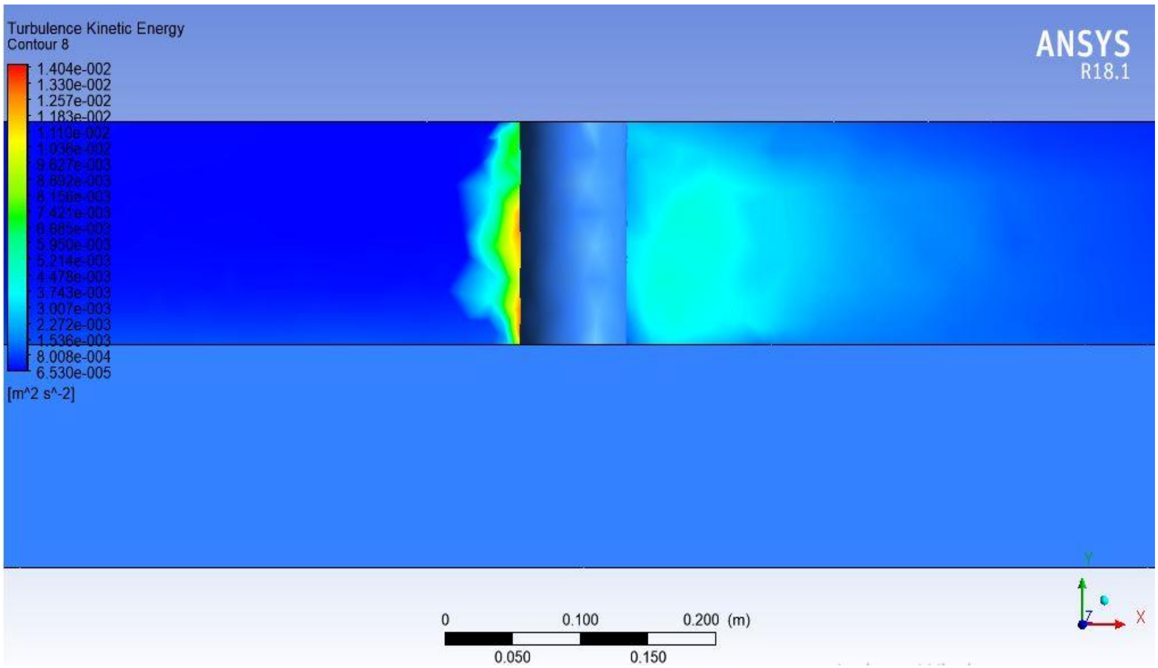

Figure 4.51 Turbulence Kinetic Energy on Longitudinal Plane (Circular Pier)

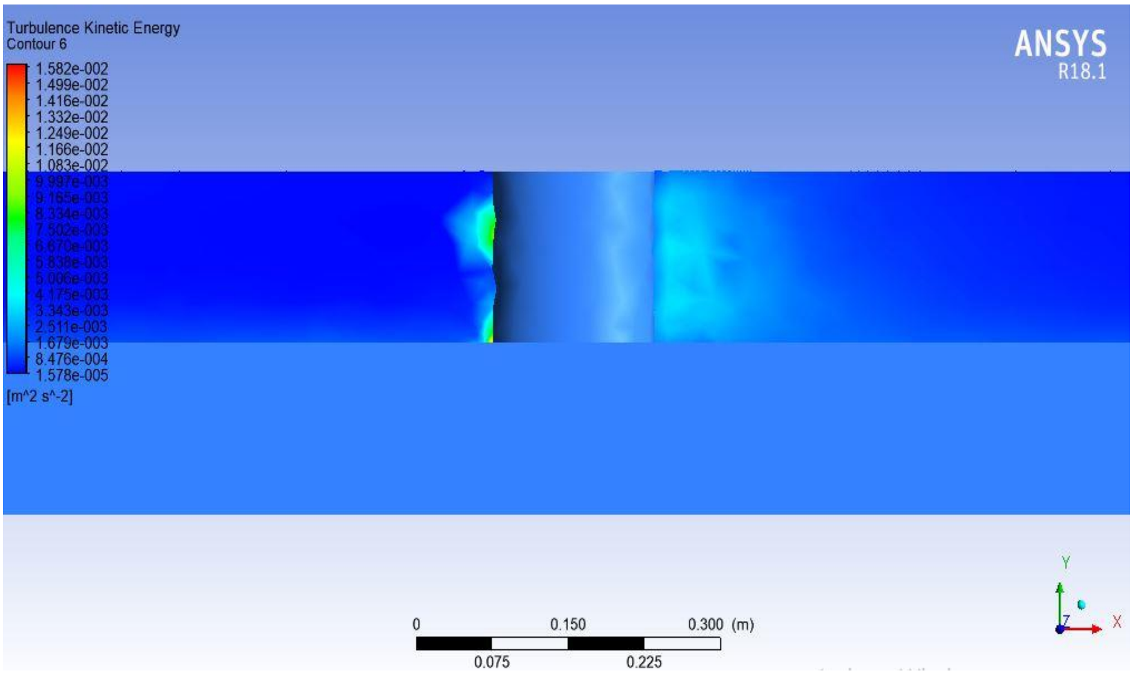

Figure 4.52 Turbulence Kinetic Energy on Longitudinal Plane (Elliptical Pier)

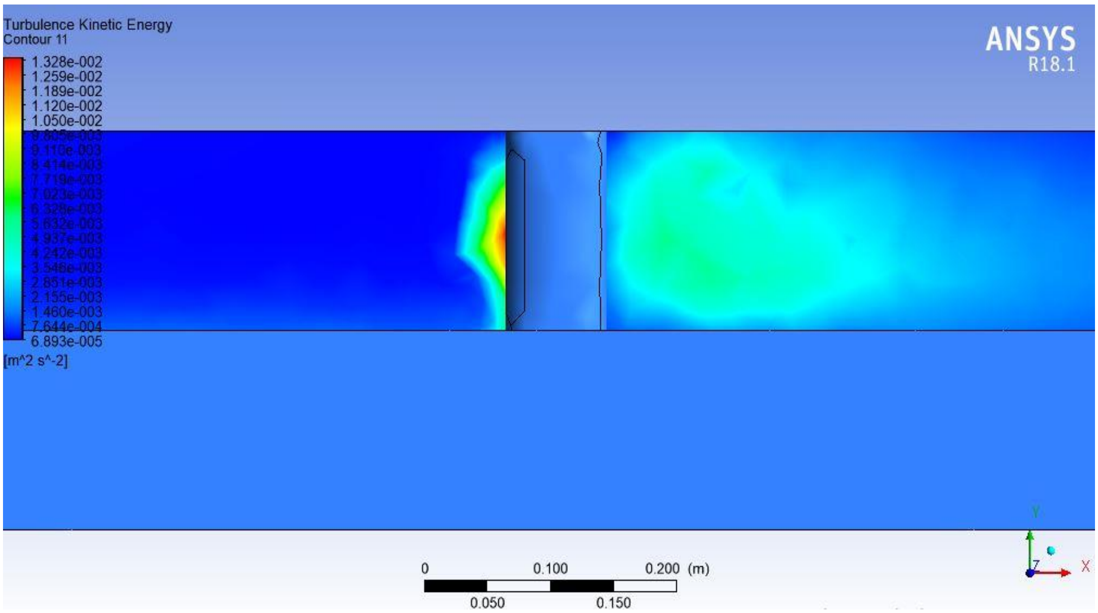

Figure 4.53 Turbulence Kinetic Energy on Longitudinal Plane (Square Pier)

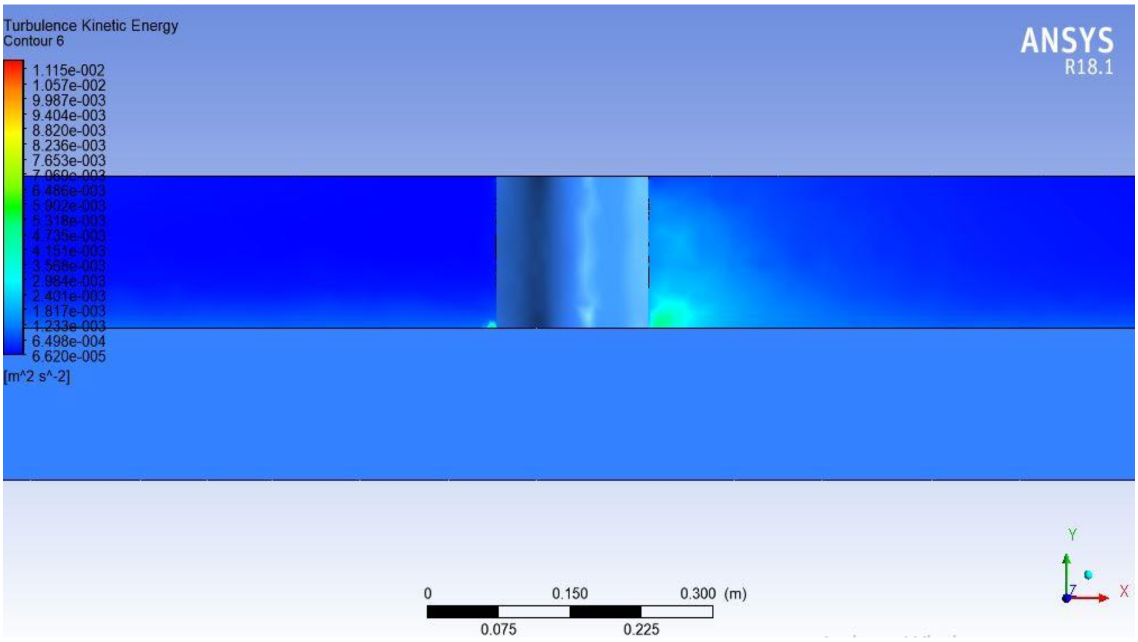

Figure 4.54 Turbulence Kinetic Energy on Longitudinal Plane (Streamlined

The Graphs showing the comparison of turbulent kinetic energy on a longitudinal line upstream and downstream of the pier are as follows,

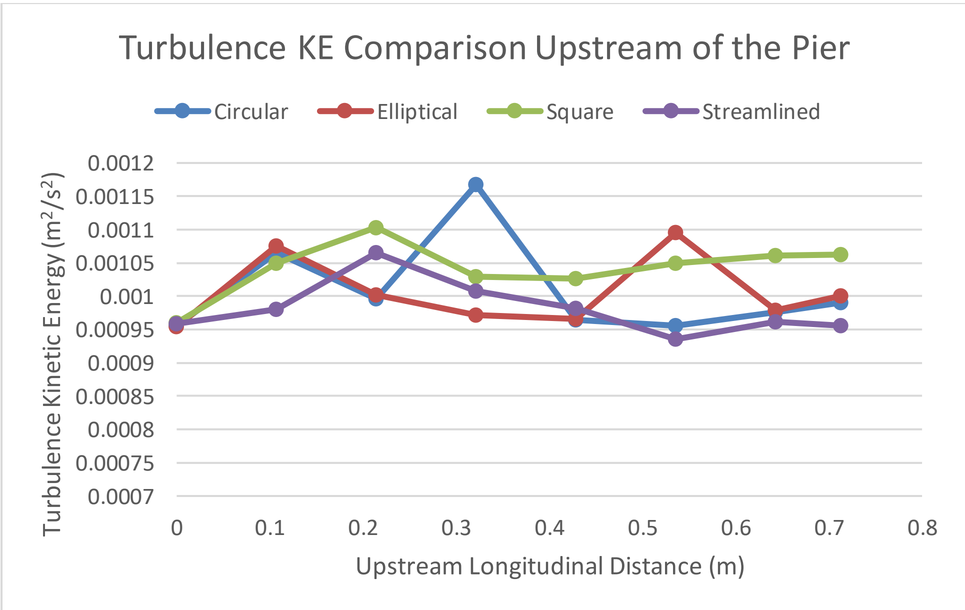

Figure 4.55 Turbulent Kinetic Energy Comparison Upstream of Pier (Sand Bed)

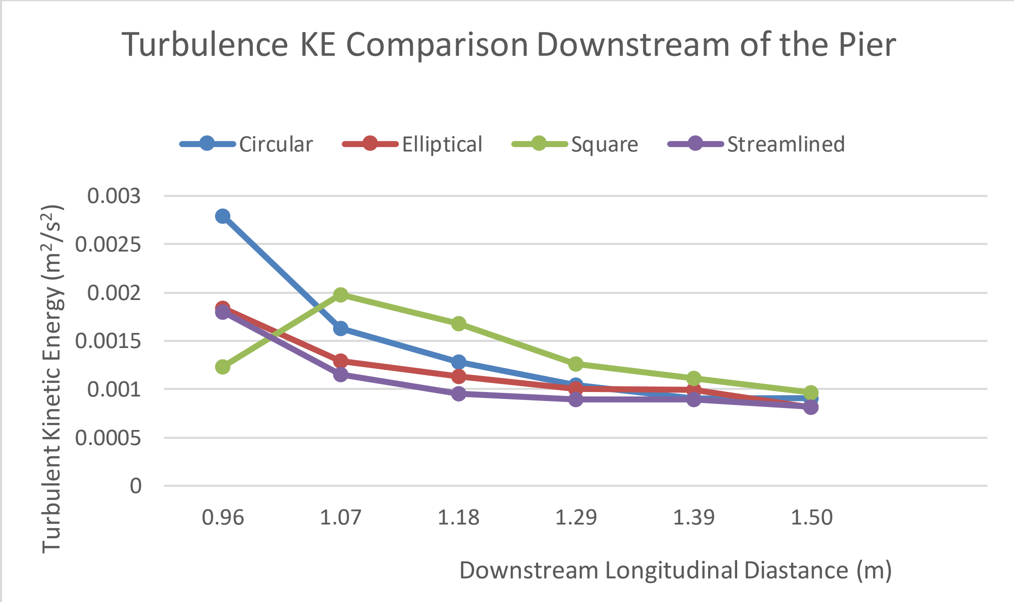

Figure 4.56 Turbulent Kinetic Energy Comparison Downstream of Pier (Sand Bed)

Table 4.10 Turbulence Kinetic Energy Upstream of Pier (Sand bed)

Longitudinal Distance(m)	**Turbulence Kinetic Energy Upstream Side**			
	Circular	**Elliptical**	**Square**	**Streamlined**
0	0.000956	0.000954	0.00096	0.000959
0.107142858	0.001066	0.001075	0.001049	0.00098
0.214285716	0.000996	0.001002	0.001103	0.001065
0.321428567	0.001167	0.000971	0.001029	0.001008
0.428571433	0.000964	0.000966	0.001026	0.000981
0.535714269	0.000956	0.001095	0.001049	0.000935
0.642857134	0.000976	0.000978	0.001061	0.000961
0.713	0.00099	0.001	0.001062	0.000956

Table 4.11 Turbulence Kinetic Energy Downstream of Pier (Sand bed)

Longitudinal Distance(m)	Turbulence Kinetic Energy Downstream Side			
	Circular	Elliptical	Square	Streamlined
0.964285731	0.002794	0.00184	0.001236	0.001802
1.07142854	0.001625	0.001291	0.001978	0.001147
1.17857134	0.00128	0.001129	0.001683	0.000958
1.28571415	0.001044	0.001003	0.001263	0.00089
1.39285696	0.000908	0.000996	0.001111	0.000889
1.49999976	0.000907	0.00081	0.000966	0.000819

4.2 Discussion

- Pressure, Shear Stress, Velocity and Turbulent Kinetic Energy variations are studied for four shapes of pier geometry namely, Circular Pier, Elliptical Pier, Square Pier and Streamlined Pier.
- Pressure variation is analysed on the pier wall by taking 3 lines Line 1, Line 2 and Line 3 on the upstream face of each pier.The graph of pressure comparison on line 1 shows that pressure exerted by the flowing water on the upstream pier wall is minimum in case of streamlined pier. The Pressure acting on the toe of circular, elliptical, square and streamlined pier are 127 Pa, 83.03 Pa, 142.37 Pa and 80.09 Pa respectively. The pressure value at the streamlined pier toe is 43.74%, 3.54% and 36.93% less than the square, elliptical and circular pier respectively.
- Pressure variation is also analysed on the sand bed by taking two lines one longitudinal and other lateral to flume length and passing through the pier.The graph of pressure comparison on longitudinal line shows that pressure acting on sand bed near the pier toe is minimum in streamlined pier. The posible reason for this can be due to the absence of horse shoe structure in case of streamlined pier.The pressure acting on the sandbed near the upstream toe of circular, elliptical, square and streamlined pier are 65.56 Pa, 66.87 Pa, 89.59 Pa and 60.97 Pa

respectively.The pressure in case of streamlined pier is 31.94%, 8.82% and 7% less than the square, elliptical and circular pier respectively.

- Shear Stress variation is also studied on the pier wall and sandbed. The variation on the pier wall is compared by taking 3 lines Linw 1, Line 2 and Line 3 on the upstream face of the pier wall. The graph showing shear stress comparison on line 1 indicates the shear stress values as 1.97 Pa, 0.25 Pa, 0.11 Pa and 0.02 Pa for circular, elliptical, square and streamlined pier.The shear stress in streamlined pier is 81.81%, 92% and 98.98% less as compared to square, elliptical and circular pier respectively.
- Shear stress on the sand bed near the circular, elliptical, square and streamlined pier is 0.50 Pa, 0.51 Pa, 0.54 Pa and 0.49 Pa respectively.Shear stress in streamlined pier is 9.26%, 3.92% and 2% less as compared to square, elliptical and circular pier respectively.
- Velocity variation is represented by velocity vectors and velocity streamlines for both water and sand. The flow velocity gets maximum obstruction from the square pier and minimum from the streamlined pier.
- Turbulence Kinetic energy distribution is also studied on the longitudinal plane passing through the pier. The turbulence kinetic energy value is compared on a longitudnal line on the sandbed .The graph showing the comparison of four piers indicates that the turbulence is maximum in square pier both on upstream and downstream side of the pier.

CHAPTER 5

CONCLUSIONS

5.1 Conclusions

- In the present study, scouring process around the bridge piers was numerically analyzed for four different bridge piers cross sections viz., circular, elliptical, square and streamlined shaped. Three dimensional numerical simulation is done to predict the flow variables around the pier using Ansys Fluent 18.1.
- Formation of horse shoe vortex is first stage of local scouring phenomenon. In the present study, a fixed scour hole is considered around the piers. The scour hole geometry of each pier is constructed by taking information from the general scour hole formation around each pier.
- The present numerical analysis mainly focuses on the variation of pressure, shear stress, velocity and turbulence kinetic energy in the flow domain of the 3-dimensional model.
- The flow field makes direct impact on the upstream face of the pier. Through numerical results, it can be concluded that lesser the projected area of the upstream face of the pier lesser will be the obstruction to the flow and lesser will be the chances of formation of horse shoe vortex. Therefore, Streamlined pier is more effective to use as compared to other pier shapes.
- The shear stress variation on the sand bed shows that the region near the toe of square pier is the most prone to local scouring due to greater value as compared to other pier shapes. This concludes that square pier is the least effective and should be prohibited wherever necessary.
- From the results of turbulence kinetic energy variation on longitudinal plane and longitudinal line on the sand bed, it can be concluded that circular and square shaped piers offer more turbulence as compared to streamlined and elliptical pier.

5.2 Future Scope of the study

- Three dimensional multiphase numerical simulation of flow around four bridge pier cross-sections is being carried out. One can choose other bridge pier cross-sections such as Oblong, Hexagonal, Diamond and Rectangular etc.
- Local scouring depends on various factors like upstream flow velocity, bridge pier

shape, flow depth and dimensions of the pier and channel. In the present study, everything is kept constant except pier shape. The study can also be extended to effects of flow depth and upstream velocity on local scouring and variations of flow variable around the pier.

- Experimental study can also be carried out for the present study and verification of experimental and numerical results can be carried out.
- In the present study, fixed scour hole is considered around the pier with fixed sand bed, numerical simulation can also be carried out for flexible sand bed with the help of dynamic mesh technique. The simulation is with respect to time.
- In addition to the present study, one can also study the effects of change of angle of attack of flow on the pier with local scouring. This can be achieved by changing the orientation of the pier.
- The present simulation study is done with eulerian multiphase, k-ε turbulence model. The CFD studies of local scouring can also be carried out with other multiphase and turbulence model like Volume of fluid (VOF), mixture model and k-ω model.

REFERENCES

- Prasanna S V S N D L, and Suresh Kumar, N.(2018) *Simulation of Flow Behaviour around Bridge Piers Using ANSYS – CFD.*
- Wen Xiong, Pingbo Tang, Bo Kong, C. S. Cai (2016) *Reliable Bridge Scour Simulation using Eulerian Two-Phase Flow Theory.*
- Mohammad Vaghef, Hamed Dashtpeyma, Arash Adib, Javad Roohian (2011) *Numerical Analysis of Flow Pattern around square Bridge Piers using New Ansys Software.*
- Zaid Hadi Obeid, Dr. Abdul-Hassan K. Al-Shukur (2016) *3D Numerical Simulation of Local Scouring and Velocity Distributions around Bridge Piers with Different Shapes.*
- J. S. Antunes do Carmo (2005) *Experimental Study of Local Scour around bridge piers in rivers*.
- J. A. Vasquez, B. W. Walsh(2018) *CFD simulation of local scour in complex piers under tidal flow.*
- Aghaee Y and Hakimzadeh H.,(2010) *Three Dimensional Numerical Modeling of Flow around Bridge Piers Using LES and RANS*: River flow, Dittrich, Koll, Aberle & Geisenhainer (eds) Bundesanstalt für Wasserbau, , 221-218.
- May Than Zaw, Cho Cho Thin Kyi, Win Win Zin (2018) *Comparison of Bridge Pier Scour by Using Numerical simulation with FLUENT and Field Observation*
- P. X. Ramos, R. Maia, L. Schindfessel, T. De Mulder, J. P. Pêgo (2016) *Large Eddy Simulation of the water flow around a cylindrical pier mounted in a flat and fixed bed.*
- Zhu Zhi-wen, Liu Zhen-qing (2012) *CFD prediction of local scour around bridge piers*.
- Deepika Bhulla, Rajendra Magar (2017) *Scouring Around The Bridge Pier-A State of Art.*