

Comparison of the flow characteristics with Two-Equation K- ϵ Model Simulation in presence of different shapes of structures in a particular domain

Md Shike Bin Mazid Anik¹, Mohammad Kaosain Akbar²

¹Department of Building, Civil and Environmental Engineering, Concordia University, Montreal, QC, Canada

²Concordia Institute for Information Systems Engineering, Concordia University, Montreal, QC, Canada

¹md_anik@live.concordia.ca

²kaosain.makbar@gmail.com

DOI: 10.29322/IJSRP.13.08.2023.p14013

<http://dx.doi.org/10.29322/IJSRP.13.08.2023.p14013>

Paper Received Date: 05th July 2023

Paper Acceptance Date: 05th August 2023

Paper Publication Date: 16th August 2023

Abstract— The article is about the flow characteristics analysis of a particular domain in presence of different shapes of structures by performing a two-equation [K- ϵ] model simulation. Three different types of structures (rectangular, triangular, and ellipse) are used to make the blockage of the flow. After the creation of the geometry through Ansys Fluent Software in presence of the structures, the domain is prepared by meshing the water phase with centimeters and the air phase with 70 centimeters of cell size. Then the model is set up for a two-equation K- ϵ model to run by applying proper boundary conditions. With a successful run, the model shows the flow contour, which indicates velocity, vorticity, pressure, turbulence intensity, and turbulence kinetic energy inside the domain. The distance vs dissipation rate, kinetic energy, and velocity graph shows the various circumstances of a flow particle with the change of distance. Also, it reflects the effect of the different shapes of structures on the flow behavior. Finally, considering the limitation of computer storage and run time, the model still shows a good and quite expected result which can be related to practical life experience. For further studies, there is a scope to run K- ω model with the same parameter and shape of structures and compare the result and key findings of the study with the current model.

Keywords—Keyword are your own designated keyword which can be used for easy location of the manuscript using any search engines. It includes at least 5 keywords or phrases in alphabetical order separated by comma.

INTRODUCTION

K- ϵ model Model is commonly used in most of the computational fluid dynamics (CFD) to simulate the mean characteristics and behavior in turbulence condition of the flow. It is a two-equation model that gives a general description of turbulence by means of two transport equations [1]. This model starts with two major equations of Boussinesq approximation and transport equation of turbulence kinetic energy and eventually

derives exact partial differential equation of kinetic energy dissipation.

The main reason for choosing the K- ϵ model model over the K- ω model is K- ϵ model predicts flow behavior well in the region far away from the wall where K- ω is suitable for flow in the viscous sublayer [2]. Besides, the K- ϵ model is reliable for free shear flow as well as belongs to high accuracy and fast convergence rate compared to the other models.

The main objective of the study is to observe and analyze the flow characteristics in a domain in presence of different shapes of structures. First, the comparison of the flow behavior by its velocity, vorticity, pressure, turbulence kinetic energy, turbulence intensity, etc in presence of three different shapes of structures: rectangular, triangular, and ellipse by two-equation model simulation [K- ϵ model]. Secondly, a brief discussion of the change of flow characteristics in the existence of different shapes of structures.

The article is divided into five major sections. The initial section is a general overview of the 'Two Equation Model [K- ϵ model]' simulation with the objectives of the study and preparations for the report. Next, we discuss the model setup process including the creation of geometry in Ansys Fluent Software as well as meshing of the geometry in presence of the different shapes of structures. The third section describes the process of calculation of the model with initialization and patch and the fourth section elaborates the performance of the model that comes with the results of pressure, velocity magnitude, vorticity magnitude, turbulence intensity, and kinetic energy contour of the model in presence of different types of structures. It is seen that the model flow behavior in the existence of different types of structures. This section also includes a discussion about the distance with dissipation rate as well as

velocity graph along with the comparison of the outcoming results. The last section highlights the conclusion and summary of the study made in this article.

MODEL SETUP PROCESS

For the comparison of different structures in a particular domain, Ansys Fluent Software [Version 2022, R1] is chosen to run the Two Equation model [K-ε model] simulation.

A. Geometry

Geometry of the model domain is sketched from the ‘Geometry’ tab of the software through ‘Line’ command. Dimensions of the geometry are followed as below in Table 1.

TABLE 1
DIMENSIONS OF THE MODEL DOMAIN

Name	Symbol in Figure 1	Length in meter
Air Inlet	-	0.5
Air Outlet	V16	2
Water Inlet	V13	0.5
Water Outlet	V9	1
Air (Top Surface)	-	10
Water (Bottom Surface)	H10	10
Air Wall (Vertical)	V3	1
Water Wall (Vertical)	V12	1
Water Wall Boundary (Horizontal)	H1	1

After sketching the geometry of the domain, it is converted to surface from ‘Surface from Sketch’ toolbar of the software as seen in Figure 1.

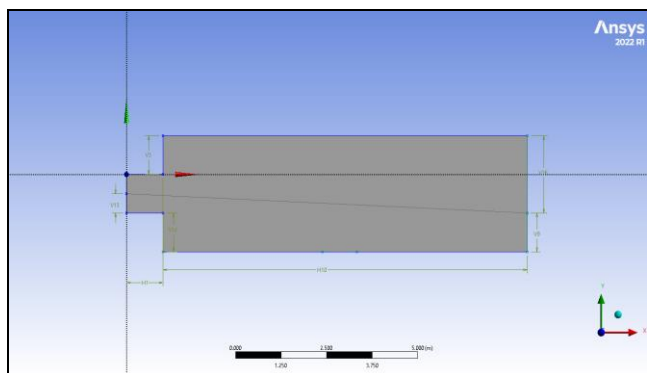


FIGURE 1
MODEL DOMAIN

Three different types of structure are chosen and sketched inside the model domain to observe the change of flow in the domain while running the model as seen in Figure 2. They

are: (i) Rectangular [1 meter x 1 meter]; (ii) Triangle [3 arms 1 meter each] and (iii) Ellipse [2 meters large diameter and 1-meter small diameter]. Ellipse was chosen in place of a circle mostly considering the practical life scenario of a blockage inside a pipe that is not completely circular in shape.

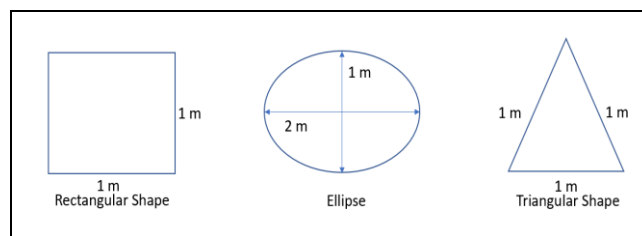


FIGURE 2
DIFFERENT SHAPES OF STRUCTURES USED IN THE MODEL

After the creation of the domain, the inlet, outlet, bottom wall, and top air portions are indicated and renamed under the domain. Also, the domain is split into two phases: air and water as well as renamed the different phases.

B. Meshing

Meshing is the method of generating a 2D or 3D grid over geometry in order to discretize it and analyze it with simulation. The grids are defined based on the complexity of geometry [3]. Meshing helps to divide the geometry into small volume parts of the cell in different irregular shapes that helps to predict the characteristics of the particle by analyzing the small part of the cell.

For the model geometry analysis, there are different size of mesh which is applied in the domain. In the air and water phase, the mesh size is 7 and 5 centimeters accordingly. Also, the intersection edge of both phases is considered as 6 centimeters. The size of the mesh is shown in Table 2.

TABLE 2
MESH SIZE OF DIFFERENT PHASES USED IN THE MODEL

Phase	Mesh Size (meter)
Air	0.07
Water	0.05
Edge	0.06

All the domains with different shapes of structures/obstacles are followed by the same size of mesh. Table 3 reflects the total nodes and elements of the mesh in presence of different types of structures along with the skewness and element quality. The average of element quality of the mesh is 0.94 for all the domains of the different structures which are above 90%. So, it is considered as a good quality mesh [4]. Also, the maximum skewness of the mesh is below 0.85 as well as average skewness is below 0.33. This also indicates the good quality of the mesh in all domains [5]. Inflation of the mesh with 4 maximum layers and a growth rate of 1.0 is also created in all of the domains to observe the effect of inflation on flow characteristics. Finally, the

mesh is updated from the Ansys Fluent Software to complete the procedure.

TABLE 3
TOTAL NODES, TOTAL ELEMENTS, SKEWNESS, AND ELEMENT QUALITY OF THE MESH IN DIFFERENT SHAPES OF STRUCTURE

Structure Shape	Total Nodes	Total Elements	Skewness	Average Element Quality
Rectangular	13091	12799	0.52714 (Max)	0.94508
			0.03459 (Avg)	
Triangle	13263	12958	0.52983 (Max)	0.94138
			0.03914 (Avg)	
Ellipse	12994	12685	0.55772 (Max)	0.9307
			0.05098 (Avg)	

C. Setup

After completing the mesh generation, the ‘Setup’ tab is selected from the Fluent software to set up the parameters for the model. The model is considered as ‘Pressure Based’ and ‘Transient’ time of the flow with a gravity of -9.81 m/s².

In order to add water-liquid as a material, the ‘Fluent Database’ tab is selected under the ‘Materials’ tab and connected to the ‘water-liquid’ option along with its properties. After that, air and water are chosen as the primary and secondary phases of the model. The surface Tension Coefficient is taken as a constant value of 0.0728 N/m in the phase interaction of the air and water.

Under the ‘Viscous’ tab, the standard ‘K-ε model’ is chosen as the method for the model. At the ‘Boundary Condition’ tab, all the regions of the model are arranged under inlet, outlet, interior, and wall zones. Air and water inlet is considered as ‘Velocity Inlet’ and their outlet is chosen as ‘Pressure Outlet’. The bottom side of the water is considered as ‘Wall’ as well as the top of the air is chosen as ‘Pressure Outlet’ to consider an open space at the top of the domain.

The whole domain is considered under atmospheric pressure in both phases. Also, a velocity of 5 m/s is applied in the water inlet inside the domain to observe the behavior of the flow in the model.

CALCULATION PROCESS

In the calculation process, the turbulence kinetic energy and turbulence dissipation rate are considered as zero. Also, those are considered as ‘Second Order Upwind’ under the solution method along with momentum and level-set function.

A. Initialization and Patch

This publication is licensed under Creative Commons Attribution CC BY.

<http://dx.doi.org/10.29322/IJSRP.13.08.2023.p14013>

Before running the calculation, it’s important to complete initialization and patch the model. For the initialization method, ‘Standard Initialization’ is selected, and the reference frame is ‘Relative to Cell Zone’. Then the patch is done with the water phase where the value of the volume fraction is set up as 1.

B. Time Step Size and Number of Time Steps

There are several numbers of time steps and time step size has been tried out for trial to run the calculation for the model. As there is a limitation of computer memory and time to run the model, the number of time steps of 1000 and time step size of 0.01 is a good fit for the model for all the domains with different structures. Also, ‘Maximum Iteration/Time Step’ is taken as 20. After that, the model is started to run by clicking the ‘Calculate’ tab.

RESULT AND DISCUSSION

After a successful run of the model, the contour is selected to observe the condition of the phase, pressure, turbulence intensity, velocity, and vorticity inside the domain in presence of the different structures.

A. Result

1 Contour of Volume Fraction of Water Phase

Figure 3 shows the contour of the volume fraction of water in a range between 0 to 1 in presence of a rectangular shape of the structure. The three consecutive images reflect the situation of the flow in the domain when the model is around 30%, 60%, and 90% completed. In the existence of the rectangular structure, more eddies are observed in the flow before the obstacle because of the backflow of the water.

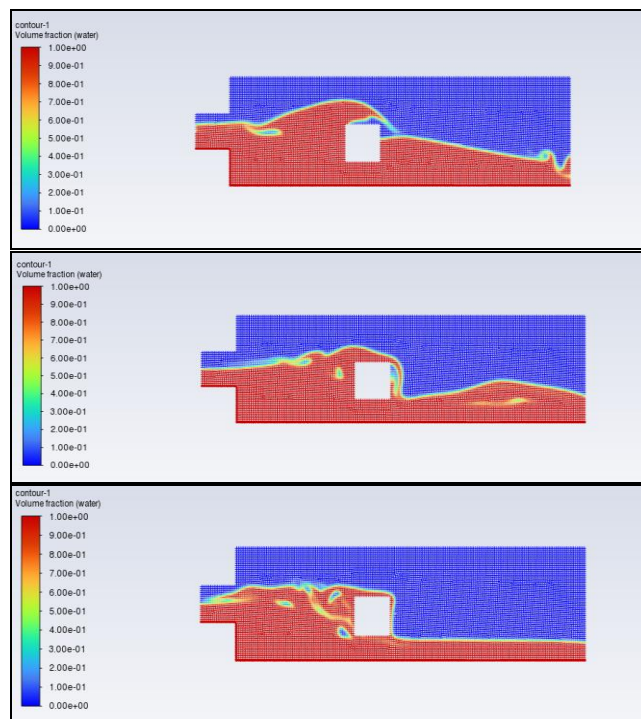


FIGURE 3
CONTOUR OF THE MODEL IN DIFFERENT PERCENTAGES OF PROGRESS [RECTANGLE STRUCTURE]

With the presence of the structure with ellipse shape, there is less viscosity found compared to the rectangular shape structure [Figure 4]. Flow is seen to pass more smoothly in the domain with the existence of an ellipse shape obstacle rather than a rectangular one.

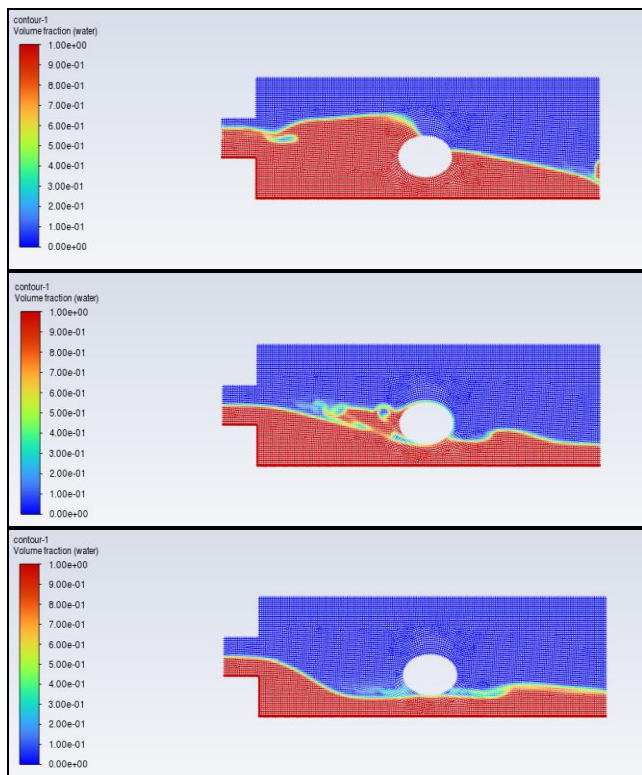


FIGURE 4

CONTOUR OF THE MODEL IN DIFFERENT PERCENTAGES OF PROGRESS [ELLIPSE STRUCTURE]

Again, with time passing (in 60% progress of the model), some of the eddies are visible before the structure due to the backflow of the water. For the triangular shape obstacle, eddies are observed to create on both sides of the structure as seen in Figure: 5.

2 Contour of Velocity Magnitude

Velocity is high under the ellipse and triangular shape structure, but under the rectangular structure, it becomes low as seen in Figure 6. It's because the rectangular structure has a sharp corner facing vertically towards the upstream which hinders the velocity to flow smoothly.

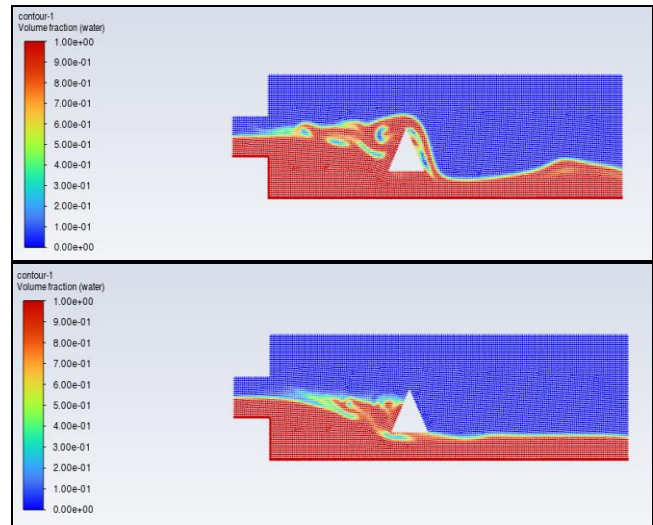
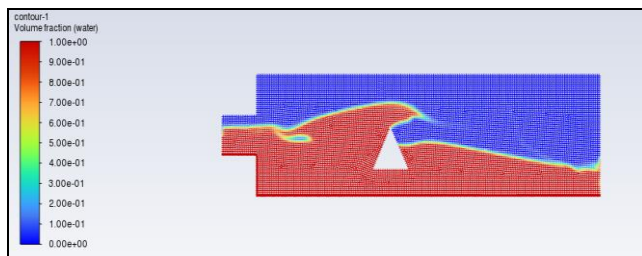


FIGURE 5

CONTOUR OF THE MODEL IN DIFFERENT PERCENTAGES OF PROGRESS [TRIANGLE STRUCTURE]

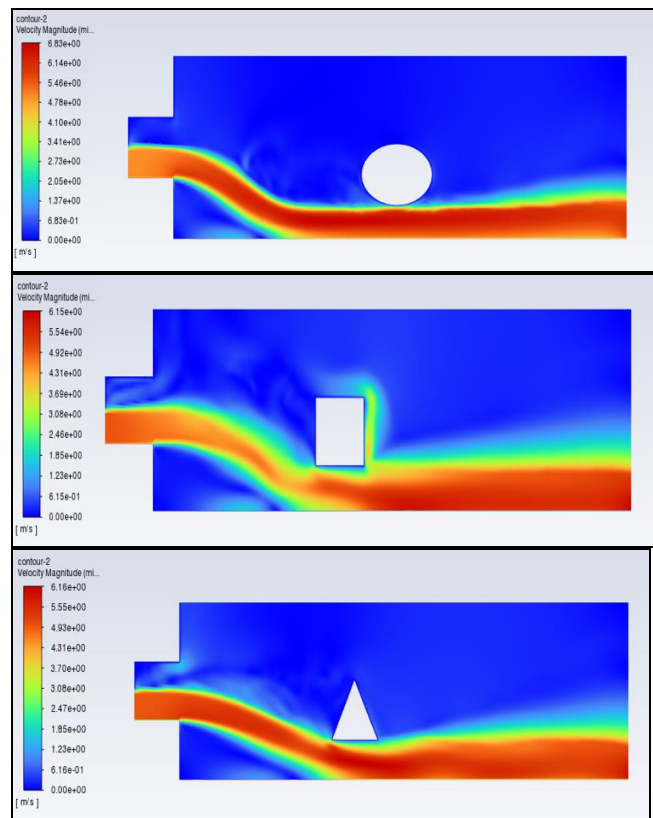


FIGURE 6

VELOCITY MAGNITUDE IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES

3 Contour of Vorticity Magnitude

For all types of different shapes of structures, vorticity is commonly visible in upstream near the air and water flow intersection inlet [Figure 7]. Vorticity is low at the bottom of the rectangular structure compared to the bottom side of the other two structures.

4 Contour of Pressure

Pressure is always high at the bottom of all shapes of structures near the bottom wall due to the high velocity of the flow seen in Figure 8.

5 Contour of Turbulence Intensity

The intensity of turbulence is observed to be very high near the rectangular shape of the structure in a comparison of the other two structures [Figure 9]. Also, turbulence intensity is high at upstream near the air and water inlet intersection in all three types of different structures.

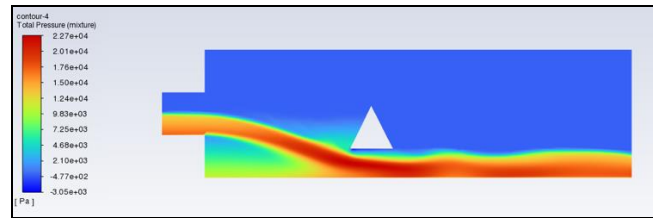


FIGURE 8
 PRESSURE DIFFERENCE IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES

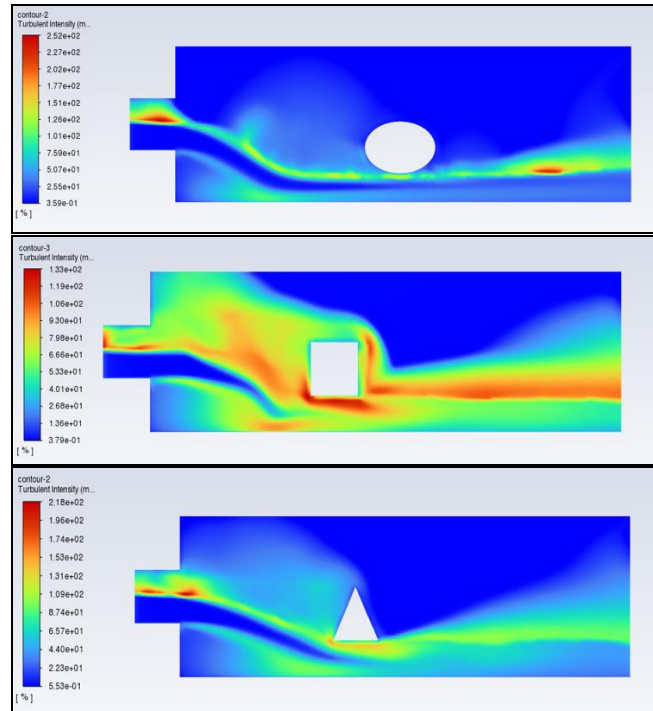


FIGURE 9
 TURBULENCE INTENSITY DIFFERENCE IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES

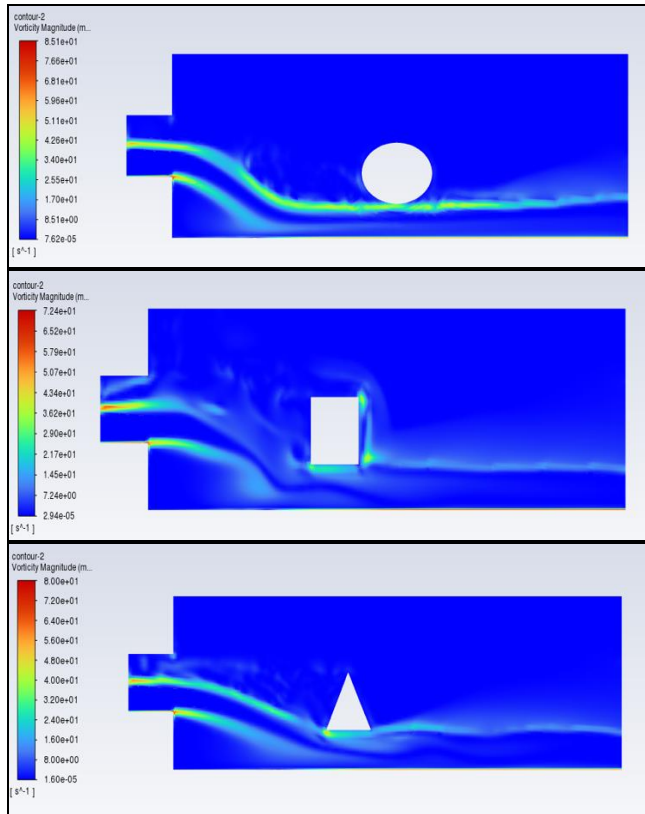
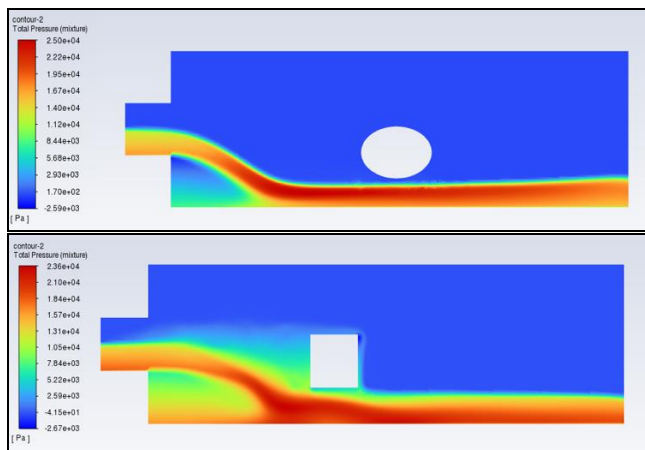
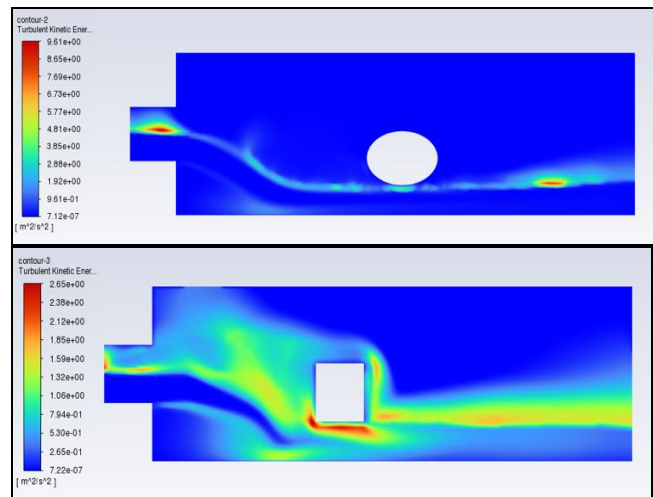


FIGURE 7
 VORTICITY MAGNITUDE IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES



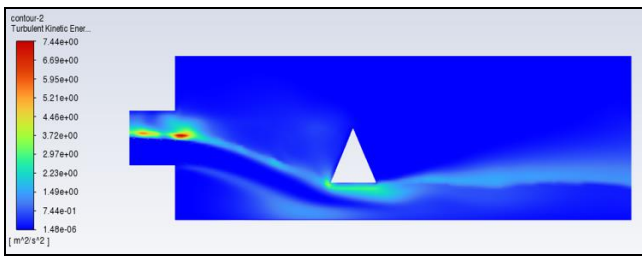


FIGURE 10
 KINETIC ENERGY DIFFERENCE IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES

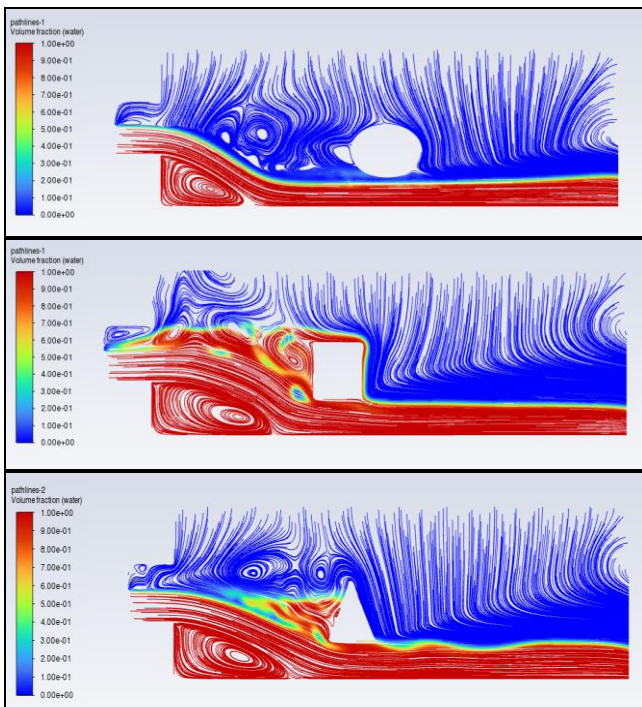


FIGURE 11
 FLOW PATHLINES IN PRESENCE OF DIFFERENT SHAPES OF STRUCTURES

6 Contour of Turbulence Kinetic Energy

Like turbulence intensity, turbulence kinetic energy also seems to be high near the rectangular shape of the structure compared to the other two structures [Figure 10]. Turbulence kinetic energy is also visible near the air and water intersection inlet at upstream in all different types of structures.

7 Flow Pathlines of Volume Fraction of Water Phase

Figure 11 shows the pathlines of the flow. In all three different types of structures, a vortex is visible to be produced near the bottom wall where flow falls from upstream to downstream.

8 Distance vs Dissipation Rate Graph

The following graphs [Figure 12 to Figure 14] reflect the turbulence dissipation rate with respect to the change of distance inside the domain. From the graph, it is seen that the dissipation rate only varies within 0 to 2 meters of the domain for the interior air phase (the black dots). For the interior water phase (the green dots), the dissipation rate is almost the same for

triangular and rectangular shapes of structures where it's comparatively high at downstream [in between 8 to 10 meter] for the ellipse shape of structures. Figure 12 describes that the dissipation rate of turbulence rises up to $50 \text{ m}^2/\text{s}^3$ for the interior water phase and $75 \text{ m}^2/\text{s}^3$ for interior air phase in existence of ellipse shape of structure. In water phase, the dissipation rate mostly varies in between 0 to $20 \text{ m}^2/\text{s}^3$.

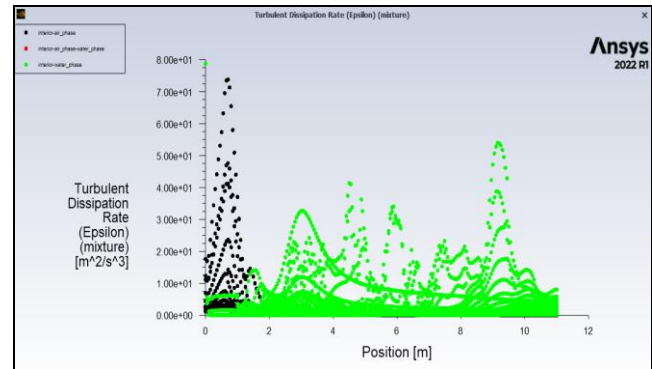


FIGURE 12
 TURBULENCE DISSIPATION RATE IN PRESENCE OF ELLIPSE SHAPE OF STRUCTURE

In presence of rectangular shape of the structure, the dissipation rate does not go up to $30 \text{ m}^2/\text{s}^3$ for both air and water phases as can be seen in Figure 13. Rather, it stays in between 0 to $10 \text{ m}^2/\text{s}^3$ for most of the particles. Dissipation rate in air phase is as almost as same as the dissipation in water phase. For air phase, dissipation is high in between 0 to 2 meter zone of the domain and for water phase, it is high in between 4 to 6 meter of the domain.

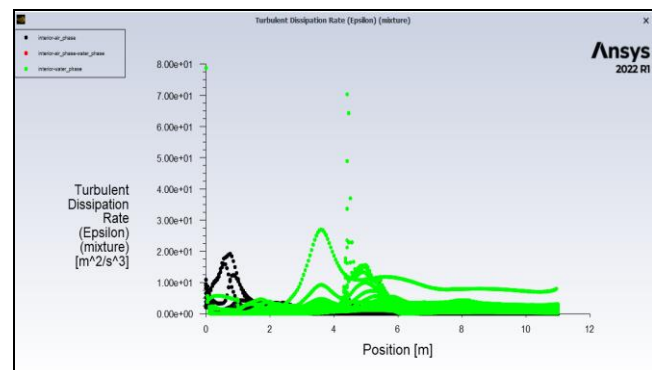


FIGURE 13
 TURBULENCE DISSIPATION RATE IN PRESENCE OF RECTANGULAR SHAPE OF STRUCTURE

Turbulent dissipation rate in presence of triangle shape structure shows nearly similar dissipation rate like the rectangular shape structure in the water phase as it varies in between 0 to $20 \text{ m}^2/\text{s}^3$ for most of the particles [Figure: 14]. But dissipation rate is higher in the air phase with triangular structure in comparison with the rectangular structure as it goes to $70 \text{ m}^2/\text{s}^3$ as a maximum value. The value of the dissipation rate in between interior air and water phase is about $40 \text{ m}^2/\text{s}^3$ which is indicated as a red dot in the Figure 14.

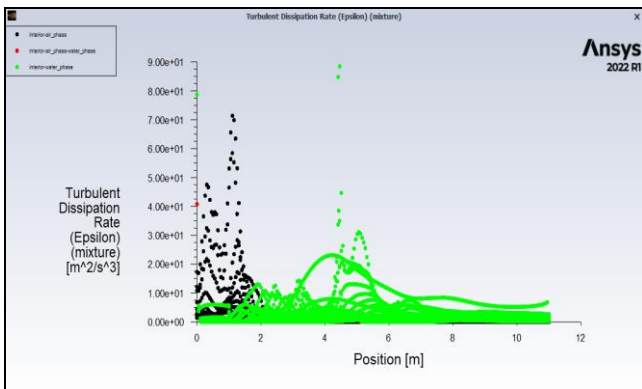


FIGURE 14
TURBULENCE DISSIPATION RATE IN PRESENCE OF TRIANGLE SHAPE OF STRUCTURE

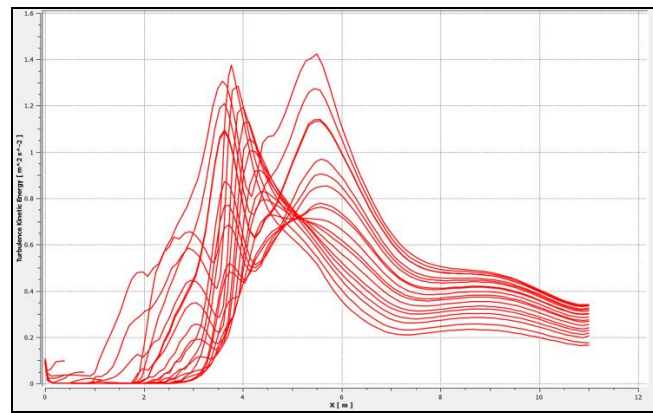


FIGURE 16
DISTANCE VS KINETIC ENERGY GRAPH IN PRESENCE OF RECTANGLE SHAPE OF STRUCTURE

9 Distance vs Turbulence Kinetic Energy Graph

From the distance vs turbulence kinetic energy graph, it is observed that turbulence kinetic energy is highest from 2 to 6 meters of the domain in presence of all types of structures.

The kinetic energy is seen to be highest at $1.5 \text{ m}^2/\text{s}^2$ in presence of the ellipse shape of structure [Figure 15]. A rise of the value of kinetic energy for most of the particles can be observed in between 2 to 5 meter distance. At downstream, near the outlet, all the lines of kinetic energy seem to converge at a certain value of about $1.5 \text{ m}^2/\text{s}^2$.

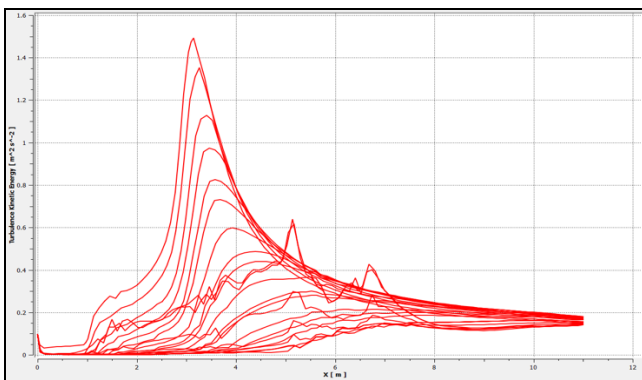


FIGURE 15
DISTANCE VS KINETIC ENERGY GRAPH IN PRESENCE OF ELLIPSE SHAPE OF STRUCTURE

For the existence of rectangular structure, kinetic energy of turbulence shows an increasing value from $0 \text{ m}^2/\text{s}^2$ to around $1.4 \text{ m}^2/\text{s}^2$ in between the distance of 2.5 to 5.5 meter inside the domain [Figure 16]. At downstream, near the outlet, all the lines indicating kinetic energy moves almost parallel showing a value in between 1.5 to $3.5 \text{ m}^2/\text{s}^2$.

Like the presence of rectangular shape structure, the triangular structure also shows an increasing line near the place of the structure in the domain. It clearly proves that with the presence of structure, turbulence kinetic energy increases significantly.

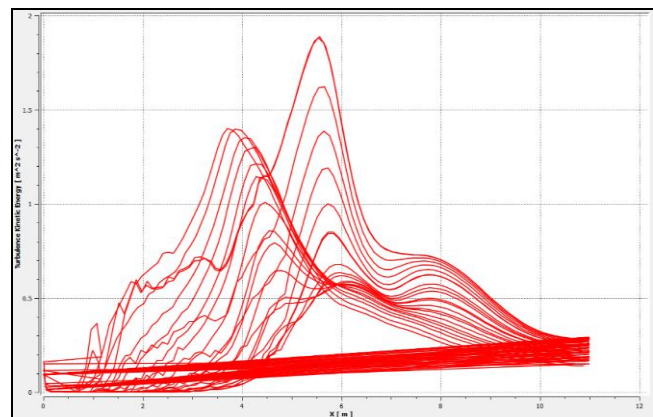


FIGURE 17
DISTANCE VS KINETIC ENERGY GRAPH IN PRESENCE OF TRIANGLE SHAPE OF STRUCTURE

In the existence of a triangular shape structure, some of the particles of the flow show an increasing straight line in the graph which indicates that those particles are not affected much by the kinetic energy of turbulence and remain below the value of $0.3 \text{ m}^2/\text{s}^2$ [Figure 17]. Also, the graph indicates that the kinetic energy of turbulence shows the highest value of about $1.9 \text{ m}^2/\text{s}^2$ in presence of triangular shape structures in comparison with other shapes of structures.

10 Distance vs Distance vs Velocity Graph

From the distance vs velocity graph, it is found that velocity becomes low when it meets with any type or shapes of structure. Some particles of the flow show an increasing straight line in presence of ellipse and triangular shape of structures. It's because those particles are not affected by the existence of the structures and reasonably, shows no curvature in the graph.

Figure 18 indicates that in presence of ellipse structure, the minimum velocity decreases to 0.5 m/s around. Most of the downfall of the velocity is visible in between 2 to 5 meter where the structure is situated. A bundle of straight lines of the graph shows that some of the particles inside the domain has no effect of the structure and therefore, their velocity increases with

distance and time. Maximum velocity in presence of ellipse structure is found to be around 6.2 m/s.

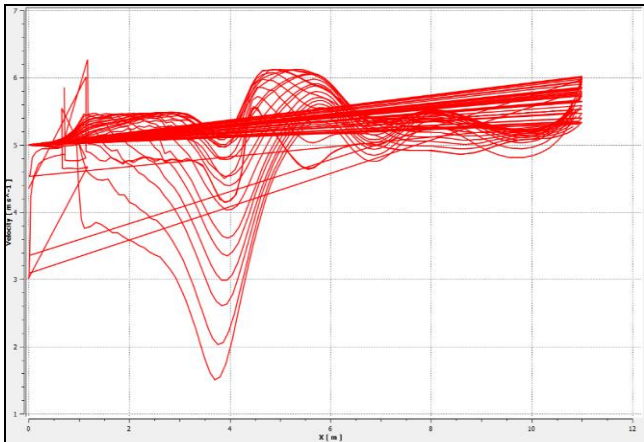


FIGURE 18
DISTANCE VS VELOCITY GRAPH IN PRESENCE OF ELLIPSE SHAPE OF STRUCTURE

In presence of the rectangular structure, the velocity seems to increase near the outlet at the downstream [Figure 19] and becomes the maximum value of about 6 m/s. Also, there is a downfall of the velocity in between 3.5 to 4.5 meter of the domain because of the presence of the structure. The minimum velocity is observed as 0.6 m/s around.

In existence of rectangular structure, no slightly increasing straight line is visible in the graph which indicates that all of the particle inside the domain is somehow affected by the rectangular shape of obstacles.

In Figure 20, it is observed that the minimum velocity of the graph is found to be about 0.5 m/s where the maximum velocity is around 6.2 m/s in presence of triangular shape of structure.

Like the existence of ellipse graph of Figure 18, this graph also has some increasing straight line that indicates some of the velocity particles are not affected by the triangular structure.

Likewise other distance vs velocity graphs from Figure 18 to Figure 20, it is clear that the value of velocity decreases with the presence of structure.

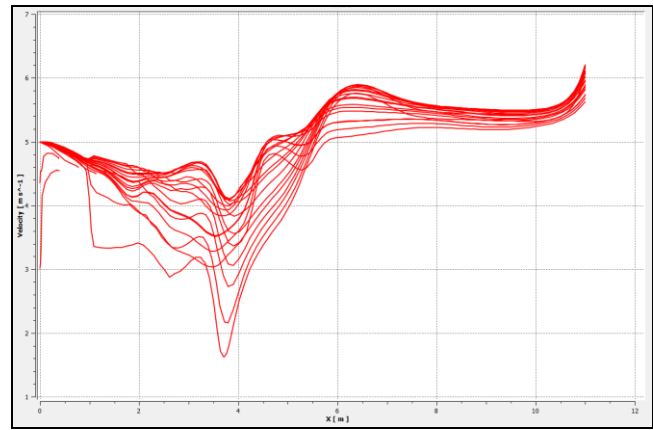


FIGURE 19
DISTANCE VS VELOCITY GRAPH IN PRESENCE OF RECTANGLE SHAPE OF STRUCTURE

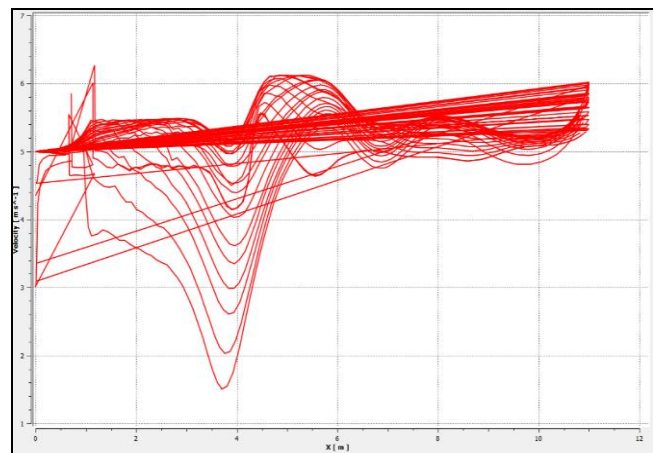


FIGURE 20
DISTANCE VS VELOCITY GRAPH IN PRESENCE OF TRIANGLE SHAPE OF STRUCTURE

B. Discussion

The model works well with small meshing, but as the length of the bottom channel in downstream is 10 meters, less than 50 centimeters of meshing takes a long time to process. That's why the mesh size of water is taken as 50 centimeters so that it may have a smooth run-on time.

The mesh does not fulfill Zang's first Guideline of the mesh where it is mentioned that at least one y^+ value of the nodes (distance from the bottom wall) must be in the interval $0 < y^+ < 1$ and at least three nodes in the interval $0 < y^+ < 10$ [6]. To maintain this restriction, the velocity of the flow has to be very less, and the mesh size has to be very small [0.0001 meter approximately]. It is a time-consuming process, and the computer also needs a high memory and processor speed. To avoid this circumstance, the minimum mesh size is taken as 50 centimeters and it is processed with a significant amount of velocity of 5 m/s.

Results from the model and graph analysis are fine and as expected. Velocity becomes slow when it passes the structure. The rectangular structure holds back more velocity and increases

turbulence intensity as well as kinetic energy because of its larger surface area compared to the other two structures. Vorticity is seen to be highest at upstream near the air and water interaction phase for all three types of structures.

The turbulence dissipation rate in presence of an ellipse is found to be high in comparison to the other two structures. Therefore, from the distance vs turbulence kinetic energy and velocity graph, it is clear that in presence of structures of any shape, turbulence kinetic energy increases and velocity decreases at the place of the obstacle.

CONCLUSION

With the two-equation model [K- ϵ] simulation in presence of different shapes of structures, the flow of water shows quite expected behavior as it appears in the real life. As the surface area of the structure increases, the flow becomes more turbulent by creating different sizes of eddies.

For further studies, this model can be used to perform with a different two-equation model like K-Omega Model or one-equation model like Spalart-Allmaras Model and compare the outcomes from those models with this current one.

Also, the limitation of resources of the computation process is a factor to keep in mind as running a turbulent model with very small mesh sizing and different shape and size of structures can cause a long time to complete.

References

- [1] Wikipedia, K-epsilon turbulence model, the free encyclopedia, https://en.wikipedia.org/wiki/K-epsilon_turbulence_model
- [2] Kobeissi, Raif, Difference between of K-epsilon and K-omega model, Ansys Forum, URL: <https://forum.ansys.com/forums/topic/difference-between-k-epsilon-and-k-omega-turbulence-model>
- [3] Simscale, What is a Mesh?, URL: <https://www.simscale.com/docs/simwiki/preprocessing/what-is-a-mesh>
- [4] Dassults Systems, Solidworks Help, Mesh Quality Check. URL:https://help.solidworks.com/2021/english/SolidWorks/cworks/c_Mesh_Quality_Checks
- [5] Simscale, Documentation, Mesh Quality, URL:<https://www.simscale.com/docs/simulation-setup/meshing/mesh-quality>
- [6] Wilcox DC. (2006), "Turbulence Modeling for CFD", DCW Industries, La Canada, California.