

AN EDUCATIONAL SOFTWARE PACKAGE FOR CFD SIMULATION AND VISUALIZATION

H. F. Manesh

**Department of Mechanical Engineering,
Eastern Mediterranean University
Gazimagosa Via Mersin 10, Turkey
hamed.farahani@emu.edu.tr**

M. Hashemipour

**Department of Mechanical Engineering,
Eastern Mediterranean University
Gazimagosa Via Mersin 10, Turkey
majid.hashemipour@emu.edu.tr**

ABSTRACT

The development of the Virtual Reality (VR) techniques for visualization of the computational simulations of complex problem has opened some new avenues for heat transfer and fluid flow research. The importance of data visualization is clearly recognized for to better understanding of the 3-D nature of the flow fields. This work introduces the educational user friendly "VRJET" package designed for teaching fluid mechanics and heat transfer. A Software is developed with C++ standard Programming language using an object-oriented approach to visualize the flow field with high performance computing including advanced support for data presentation and navigation techniques through 3D virtual environment. This work deals with 3-D visualization of the data of impinging laminar single square jet on a heated flat surface, obtained from numerical simulation. This package can be used for research, educational, and engineering.

INTRODUCTION

Computational fluid dynamics (CFD) is currently a powerful tool to solve scientific and engineering problems, involving fluid flow and heat transfer, Because Computational fluid dynamics simulations are capable of accurate simulation of fluid flow.

One of the application of CFD is in simulation of impinging jets. Impinging jets have found a large number of industrial applications because of their highly attractive local

heat and mass transfer characteristics. When a jet strikes a surface, thin hydrodynamic and thermal boundary layers form in the region due to the jet deceleration, as a result pressure is increased (Figure 1). When flow is forced to accelerate in a direction parallel to the target surface, the walled jet or parallel flow zone is formed. Consequently, very high heat and mass transfer coefficients can be obtained at the stagnation zone directly under jet.

According to extensive industrial application of jet, the result data files of Computational fluid dynamics (CFD) computations have to be presented in a way that can be understood by experts as well as by non-expert people such as students.

Traditional visualization methods are not well suited to analyze these phenomena from such large and detailed data sets. Even with the help of plotting software, the amount of the data generated by the numerical simulation can be unmanageable for analysis. More advanced visualization and data management methods are required.

As in any research, the data obtained from the experimental and numerical work are plotted in a 2-D or 3-D static plotter, in order to analyze the result. Due to recent development in computer software and hardware, the data can be visualized dynamically using virtual reality (VR) techniques.

In most of the works, the data used to visualize the flow field is generated by using a simple equation like parabolic

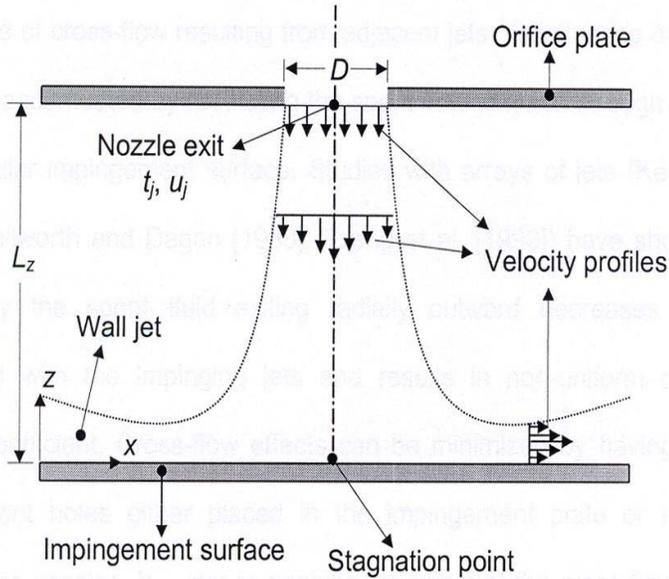


Figure 1 Surface impingement of a single slot gas jet (Aldabbagh, L.B.Y [2002]).

velocity profile in a duct flow. According to our knowledge, most of the visualization works done to date have not used result of the direct numerical simulation.

The major goal of this study is to develop an educational software (VRJET) to visualize a flow field resulting from direct numerical simulation of the laminar single square jet. In order to gaining new insight in the nature of flow, interactive visualization is essential. The implementation of the software touches upon many aspects of high performance computing, including advanced support for data presentation and navigation techniques. The specific tasks of VRJET are as follows:

- To investigate the effect of aspect ratio on a three-dimensional laminar single square jet impinging on a plate.
- To investigate the effect of Reynolds number on a three-dimensional laminar single square jet impinging on a plate.
- To develop a software for visualizing a flow field resulting from direct numerical simulation to analyze 3-D nature of the flow field.

NOMENCLATURE

A_x, A_y, A_z	aspect ratios in x, y and z direction, L_x/B , L_y/B , L_z/B
B	jet width (m)
L_x, L_y, L_z	length of heated surface in x, y and z-directions, respectively (m)
Re	jet Reynolds number ($u_j B/\nu$)
u, v, w	Cartesian velocities
u_j	jet exit velocity (m/s)
U	nondimensional Cartesian velocity, in x-direction (u/u_j)

V	nondimensional Cartesian velocity, in y-direction (v/u_j)
VE	Virtual Environment
VR	Virtual Reality
VRT	Virtual Reality Toolkit
W	nondimensional Cartesian velocity, in z-direction (w/u_j)
x, y, z	Cartesian coordinates
X, Y, Z	nondimensional Cartesian coordinates, x/B , y/B , z/B respectively

Greek symbols

ρ	density (kg/m^3)
--------	-----------------------------

Subscripts

j	jet exit
-----	----------

THE OVERALL METHODOLOGY

The overall methodology of VR based system is shown in Figure 2. Due to inherent complexity, it is convenient to decompose the overall methodology into simpler components, which can be rather assigned to one of the following systems:

- (1) Numerical Engine,
- (2) Database System,
- (3) Peripheral Interfaces
- (4) OpenGL VRT.

A dashed line is used to enclose those modules associated specially with the virtual agent. From the figure, it is clear that the graphical system does not depend on the numerical engine.

NUMERICAL ENGINE

The CFD numerical simulation code is used to produce data set of the velocity fields and temperature profiles for visualization in virtual environment. FORTRAN compiler is used due to its robust compiles designed for high-performance technical computing and the availability of useful subroutines from numerous engineering applications.

COMPUTATIONAL SCHEME

The steady-state, two-dimensional, Continuity and Navier-Stokes equations with constant properties for incompressible flows in Cartesian coordinates are used for this study and can be written as:

continuity:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

x-momentum:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + u \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

y-momentum:

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + v \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

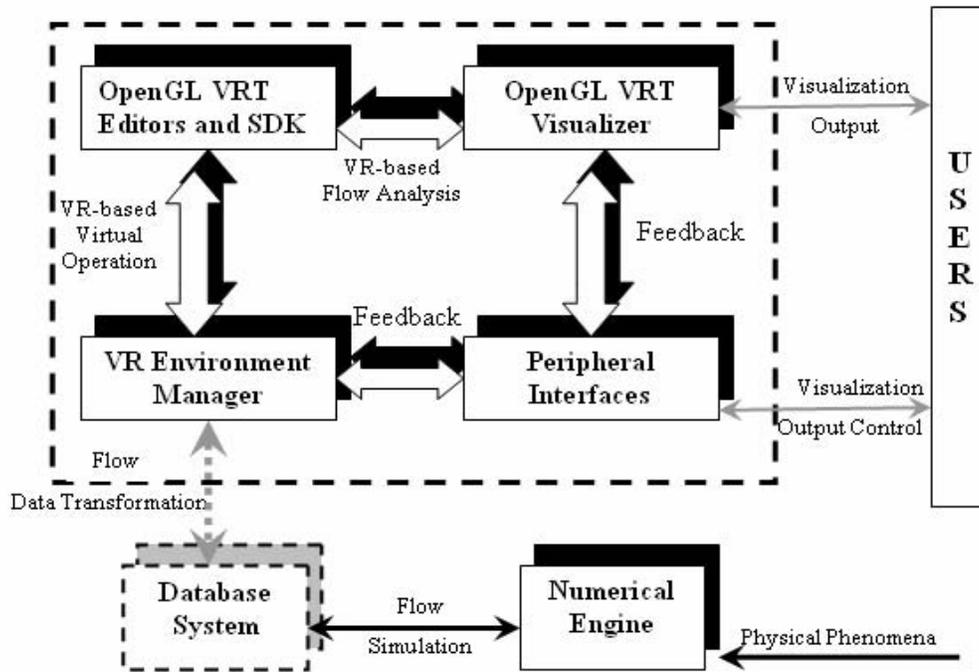


Figure 2. The Overall Methodology

The equations (1-3) can be written in non-dimensional form by normalizing the velocity terms by jet-exit velocity (u_j) and the length terms by jet width (D). Then the non-dimensional continuity and momentum equations for steady laminar flow are:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (4)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{\text{Re}} \left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right) \quad (5)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{\text{Re}} \left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right) \quad (6)$$

The outlet boundary is located far enough downstream for conditions to be substantially developed, accordingly the following conditions are imposed at outlet. Which are open to atmosphere on two sides.

$$\frac{\partial U}{\partial Y} = \frac{\partial V}{\partial Y} = 0 \quad \text{at } X=0 \text{ and } X=A_x \quad (7)$$

$$\frac{\partial U}{\partial X} = \frac{\partial V}{\partial X} = 0 \quad \text{at } X=0 \text{ and } X=A_x \quad (8)$$

All walls are stationary and impervious therefore no slip boundary condition is used for the top and bottom solid walls except for the V velocity at the jet exit cross section at the top

wall, where it assumed to be uniform and was set to be equal to unity. Hence,

$$\begin{aligned} U=V=0 & \quad \text{at } Y=0 \text{ and } Y=A_y \text{ except at nozzle exit,} \\ U=0, V=-1 & \quad \text{at the nozzle exit} \end{aligned}$$

METHOD OF SOLUTION

The governing equations are discretized by using the finite volume method in staggered uniform grids. The solution domain in x -directions has $L_x=40D$, where D is the jet width. The simulation is carried out for jet-to-plate spacing of $L_y=2D$ to $6D$. Grids of size 161×161 are used for all runs. Definitions of geometric parameters and coordinate systems are shown in Fig. 3.

The quadratic upstream interpolation for convective kinetics (QUICK) scheme (Leonard 1979) is used to calculate the convection of a scalar term (U, V field) at a control volume surface. The line-by-line method (H. K. Versteeg and W. Malalasekera [1995]) is used to solve the momentum equation. The velocities are coupled with pressure terms using SIMPLE (Patankar and Spalding [1972]) algorithm. A relaxation factor of 0.3 for the pressure correction equation and 0.7 for velocities are used. Iterations are continued until the second norm of the residuals for all equations reduce below 10^6 where no significant variation are observed at this residual level.

DATABASE SYSTEM

The system typically supported by relational database capable of storing structured data required by the program. A

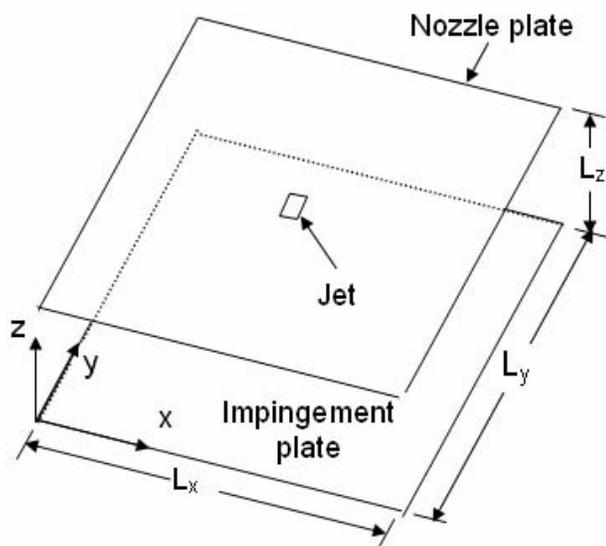


Figure 3 Definitions of geometric parameters and coordinate systems.

relational database places all information in tables. Several database programs exist, including Structured Query Language (SQL) server (Microsoft Corporation), MYSQL (AB Company, Westmonds, Washington), Access (Microsoft Corporation), and Oracle (Oracle Corporation, Redwood Shores, California). While each database program provides relatively similar relation database functions and capabilities, Access (Microsoft Corporation), was selected for use in developed software based primarily on its availability. The database is divided into the number of simulation result, each table accordingly being associated with each simulation result. The DMBS of the database is a dedicated program, with Visual C++ 6.0, integrated with those modules and the OpenGL, VRT, system.

Database characteristics of interest include data storage limits, simplicity of programming for customizable solutions, ease of integration with virtual environment. Improvement of logical design and installation of powerful database provided significantly increased performance.

THE PERIPHERAL INTERFACES

The Peripheral interface is control protocols, mostly provided by OpenGL VRT. The interface that was provided for the system is a link between the VR environment and database that enables the data to transmit between them. Since FORTRAN language lacks sufficient support for directly connecting to database and is not well suited to handling input and output, these problems are easily overcome by creating another interface between Numerical application and an underlying database. The Visual C++ 6.0 has been chosen for developing interfaces.

User-friendly, graphical user interface facilitates information flow between the user and the VE. The interface allows the user to fly-through or walk through the VE from any

perspective. The user can zoom in and out on desired areas. The animation is also controllable. The particle animation can be started, stopped, and continued and the particles can be reset back to the starting positions. After the reset, the animation can be started again from the beginning.

The menu system is constructed which consists of File, Visualization Menu, Controls and Help. A typical menu window is shown in Fig. 4.

A scalar bar has been used to inform the user about the correspondences between the colors of the visualization elements and the actual quantity values. The scalar bar is defined in a way that it will appear as a 3-D object always in front of the user and when the user position is changed it moves accordingly. Thus, it is independent from the rest of the scene. The interaction with the scalar bar is managed with one key. Besides the convenience, it is always stimulating to get something lively in the virtual world and it makes the virtual experience more attractive and thus more immersive. Animated particles were also chosen to the collection of the implemented visualization elements.

The program runs on IBM-compatible PC's under Windows 98/NT/XP. The hardware requirements must be a 586 IBM-compatible PC with a hard disk of 80 MB space and 128 MB RAM normal memory, Graphic Card (For better VR effect the system should be equipped with 3-D Graphic Card) and a color monitor.

Figure 5 displays the projection of the flow lines of the predicted velocity field on the mid vertical x - z plane and the visualization of the flow field by using VR respectively, for $Re=300$ and plate-to-plate distances of 2, 4, 8, and 12 jet widths.

CONCLUSIONS

This paper presented the educational software CFD VRJET to visualize and study complex flow structure of laminar single square jet in three-dimension resulting from direct numerical simulation in 3-D virtual environment using virtual reality. The virtual environment provides a realistically simulated environment of laminar single impinging jet, not only for engineers but also for people with no special background in 3-D modeling and fluid motion.

Human computer interaction and virtual reality are relatively young sciences specially in visualization of numerical result through 3-D Virtual environment. Though a lot of work still has to be done to improve virtual reality part as a three dimensional visualization tool, this work in its current state is an intermediate step to long term goal, visualization in n-Dimensions. The potential benefits of this work in both education and industrial areas are numerous. As the system itself has been made of standalone modules, each can be expanded to make them more generic and global in scale.

The first contribution of this study is to propose a model of virtual reality systems for education. Distance learning is becoming more and more popular. Distance education, especially for lab courses, can be realized without any

compromise. Students and teachers across geographical regions can collaborate effectively for their mutual benefits.

The main objectives in the follow-up project will be the visualization of any kind of numerical and experimental result through virtual environment and to introduce the heat transfer to the numerical code by using energy equation. For instance the full jet in three-dimension by using the three components of the velocity vector, U, V, W , temperature profile and the heat transfer coefficient on the heated plate. and visualization of the array of impinging jets can be done.

REFERENCES

Aldabbagh, L. B. Y., Sezai, I., 2002. Numerical simulation of three-dimensional laminar, square twin-jet with impingement

on a flat plate, flow structure, and heat transfer. Numer. Heat Transfer, Part A, 41, 835-850.

Aldabbagh, L. B. Y., Sezai, I., 2002. Numerical simulation of three-dimensional laminar multiple impinging square jets. Int. Journal of Heat and Fluid Flow, 23, 509-518.

Leonard, B. P., 1979. A stable and accurate convective codelling procedure based on quadratic upstream interpolation. Comput. Method Appl. Mech. Engng., 19, 59-98.

H K Versteeg and W Malalasekera, book, An introduction to computational fluid dynamics, 1995.

Patankar S. V. and Spalding D. B., A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows, Int. J. Heat mass transfer, Vol. 15, p. 1972.

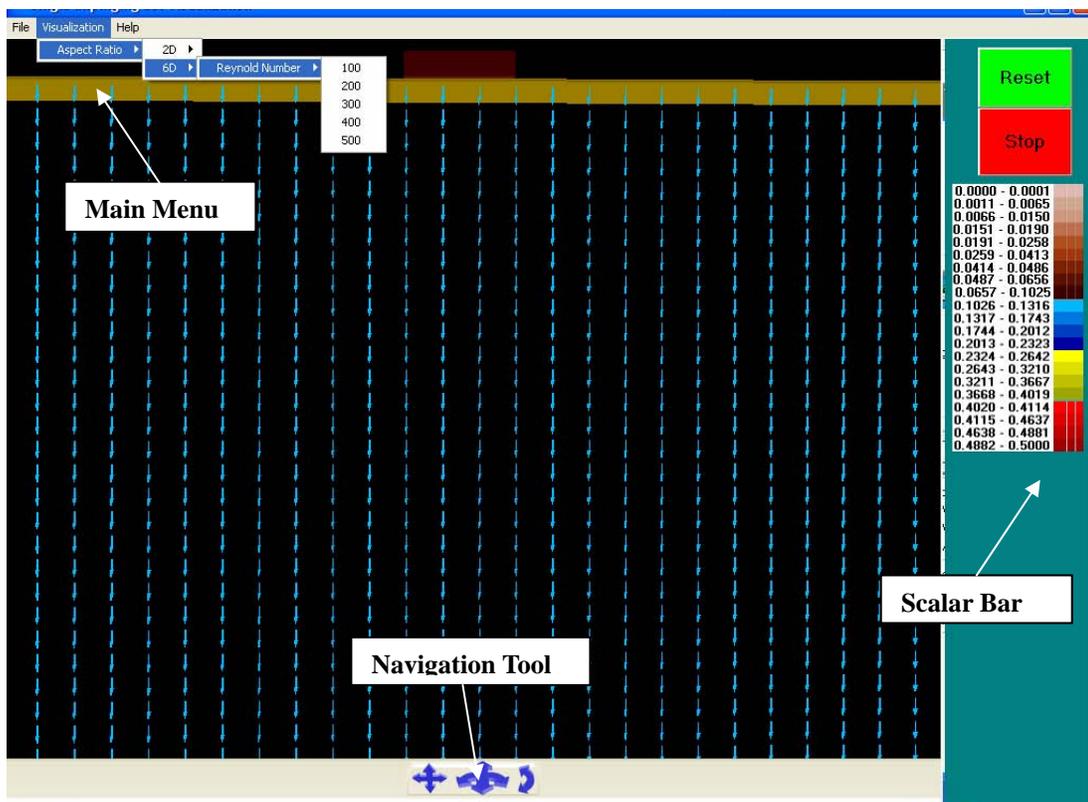


Figure 4. A Typical Menu Window

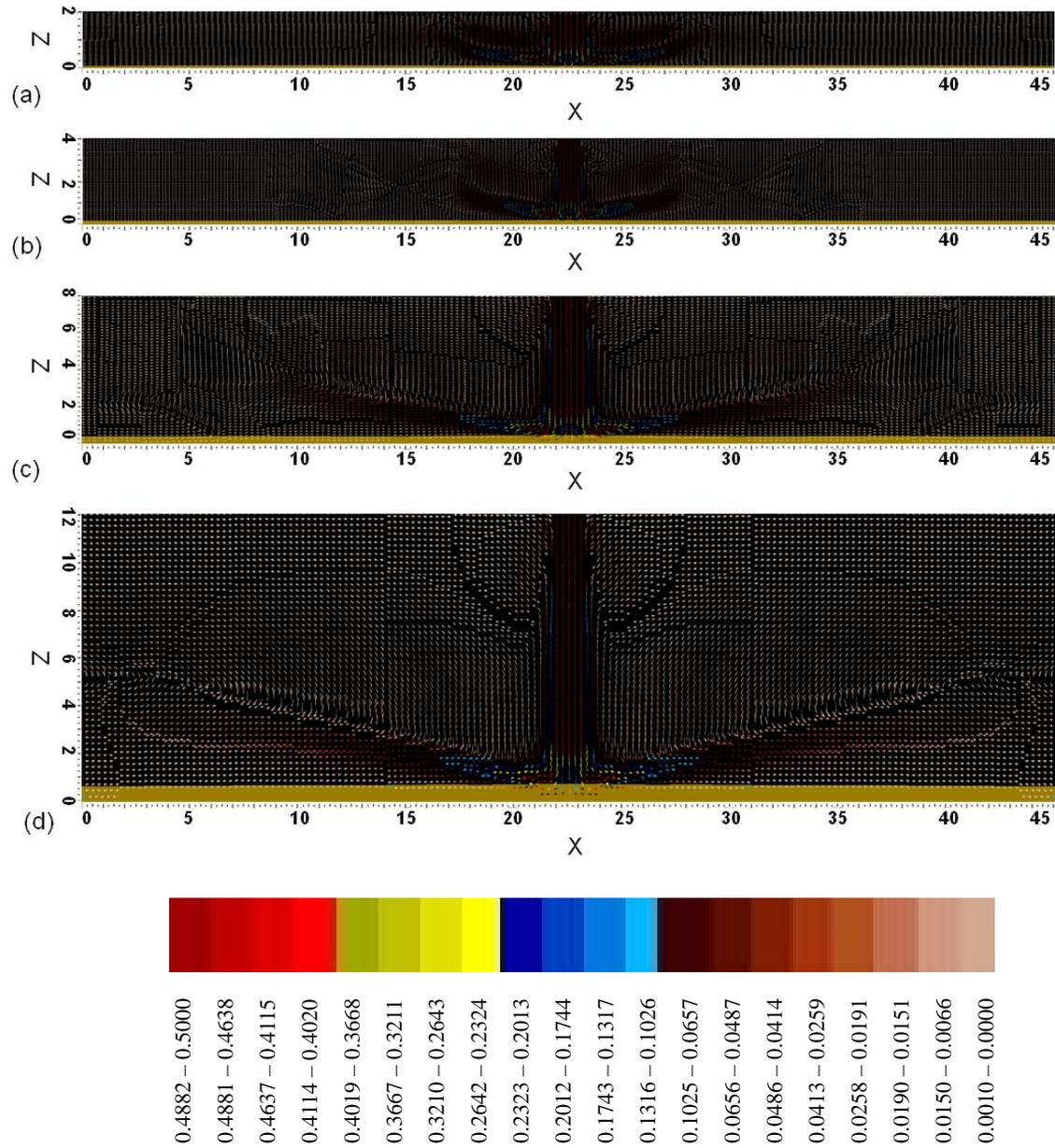


Figure. 5 Visualization of the flow field on mid x - z plane for $Re = 300$ at jet-to-plate spacing (a) $A_z=2$, (b) $A_z=4$, (c) $A_z=8$, and (d) $A_z=12$.