Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 2, Number 4; January-March, 2015, pp. 293-298

http://www.krishisanskriti.org/jbaer.html

Effect of MHD on Turbulent Flow Past Over a Hemispherical Body Placed in an Open Channel

Nishant Singhal[1], Sayahnya Roy[2] and Koustuv Debnath[3]

[1]Department of AE & AM IIEST, Shibpur Howrah-71110, India
[2]Department of AE & AM IIEST, Shibpur Howrah-711103, India
[3]Department of AE & AM IIEST, Shibpur Howrah-711103, India
E-mail: [1]nishantsinghal2012@gmail.com, [2]sayahnya1110@gmail.com, [3]debnath_koustuv@yahoo.com

Nomenclature

B = Magnetic field magnitude, Tesla
$\vec{B}$ = Magnetic field vector
$\vec{j}$ = Current density vector
$\vec{u}$ = Velocity vector
μ_0 = Magnetic permeability, $4\pi \times 10^{-7}\ N/A^2$
l = Turbulence length scale, m
d = Depth of flow, m
y = Distance from bed, m
h = Hemisphere radius, m
ρ = Water density, kg/m^3
ν = Kinematic viscosity, m^2/s
μ = Dynamic viscosity, $Pa\text{-}s$
σ = Electrical conductivity, $ohm-m$
∇p = Pressure gradient
Re = Reynolds number
Ha = Hartmann number
Pr = Prandtl number
$\vec{j} \times \vec{B}$ = Lorentz force
Q = Flow discharge, m^3/s
x, y, z = Streamwise, spanwise and transverse
i = Free index

Abstract—*An observation of MHD effect on 2-D turbulent channel flow around a hemispherical bluff body is presented. Turbulence closure is achieved with the use of $k-\varepsilon$ model for incompressible fluid flow. The Implicit formulation and second order discretization is employed for RANS equations and the $k-\varepsilon$ equations. Increasing magnitude of Lorentz force systematically reduce the velocity gradients near the bluff body. The mean flow field in wake region is examined and compared with the results applying MHD on the flow field. The flow Reynolds number is Re=9950and Hartmann numbers are 5, 8, 10 with a channel bed relative roughness of 0.0001. Computational simulations are performed to investigate how turbulence is affected by the influence of magnetic field.*

Keyword: *MHD, Hartmann number, $k-\varepsilon$ model, Open channel*

1. INTRODUCTION

Magnetohydrodynamics (MHD) is the study of the dynamics of fluid flow field with magnetic field. Study of Turbulence in MHD flow is not a new subject. For research and industrial purpose both experimental and simulation works were done since 1942. The study on MHD has wide applications in liquid metals flow, casting, power generation and many more. A Direct Numerical Simulation (DNS) has been employed by Zikanov and Thess [1] to study the turbulent heat transfer under the influence of constant magnetic field. In 2008 Yamamoto et al. [2] presented a study, the MHD pressure loss and heat-transfer characteristics for higher Reynolds number. Other numerical studies have been done for MHD duct flow by Krasnow et al. [3]. The investigation about reduction of Joule dissipation due to MHD flow by Smolentsev et al. [4] incorporated with $k-\varepsilon$ model. Further in this manner Young et al. [5] described the mechanisms which are responsible for instabilities and transition to turbulence in a liquid metal duct flow.

A experimental study for turbulent wake generated due to presence of cylinder by Rhoads et al. [6] carried for Reynolds number of order 10^4. A numerical solution for steady 2D incompressible flow past an infinite row of cylinders [7], discussed about recirculation regions grow linearly with Reynolds number. Moreover, the characteristics of turbulent flow in open channel for 2D dune shaped structure were examined by Majumder and Sarkar [8].

There are several numerical and experimental studies done on MHD as mentioned above. MHD study was not much applied in open channel flow. In present work we placed a hemispherical bluff body in an open channel and descritized by $k-\varepsilon$ model for electromagnetic flow and compiled a UDF worked as source term that creates magnetic field throughout the fluid volume.

2. COMPUTATIONAL SCHEME

A 18.0m long and height of 0.4m 2D open channel flow with 0.2m depth of water is employed as the inlet boundary condition. A hemispherical bluff body of diameter 0.06m has been placed in the channel. The velocity near the free surface is u = 0.05m/s. The Reynolds number is $\mathrm{Re} = ud/\nu$ based on water depth d and u. For simulation, $k-\varepsilon$ model was employed as the turbulence model.. The $k-\varepsilon$ model in ANSYS14.0 FLUENT software is incorporates with a user defined function as a source term to generate magnetic field.

We have tested several schemes of meshing for channel model. However, with use of unstaggered scheme with quadrilateral elements, we achieved convergence with reasonable computational effort. Boundary conditions were applied to simulate our present study by using two separate inlets for air and water and uniform distributions were given for all the dependant variables. The turbulence length scale was taken 0.014, channel bed, wall of the roughness were given wall boundary using no-slip boundary condition to set the velocity zero. The wall boundary condition requires specification of wall roughness parameters; roughness constant which we have specified as 0.5. FLUENT14 manual reports that choosing a roughness constant value of 0.5 when used with $k-\varepsilon$ models, we have specified the roughness height 0.0001 as per material property. in our present study we kept the bed as net finished. For the water inlet boundary (Fig.1.), pressure inlet boundary condition was chosen with and turbulence intensity was specified as 5.05 % and velocity magnitude was specified as 0.05m/s depending on the run case. For the free surface (Fig.1.), symmetry boundary condition was chosen. But only one outlet was specified to let the solver decide the flow level at the outlet as calculated from inside the domain. For the outlet boundary, pressure outlet boundary condition was chosen and backflow turbulence was specified as 0, the bottom level was specified as 0 m and free surface level as 0.2 m. Using FLUENT14, the models were run using unsteady second order upwind implicit finite difference formulation.

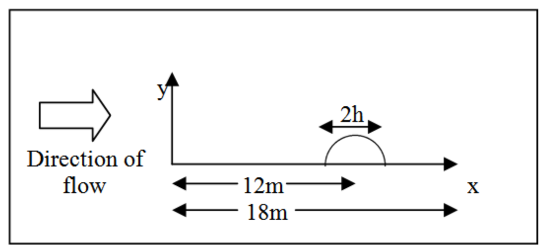

Fig. 1: Layout of plain channel and placed hemisphere

3. GOVERNING EQUATION

Flow field results are obtained using CFD to solve the Navier-Stokes equations. This Navier-Stokes equation containing a body force term that has been used as Lorentz force by developing a UDF(User defined function). The following equations are used-

The continuity equation-

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Navier-Stokes equation-

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \upsilon \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}$$

After the time averaging the above equations we get-

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{3}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \upsilon \frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{4}$$

The body force term f_i is used as Lorentz force by applying the Magnetic field through our flow domain.

The Reynolds-average Navier-Stokes equations (RANS) are time averaged equations of the motion for fluid flow. The idea behind the equations is Reynolds decomposition, whereby an instantaneous quantity is decomposed into its time-averaged and fluctuating quantities. The RANS equations are primarily used to describe turbulent flows. These equations can be used with approximations based on the properties of flow turbulence to give approximate time-average solutions to the Navier-Stokes equations.

4. PROBLEM FORMULATION

We consider the flow of an incompressible electrically conducting fluid in an open channel. The Flow is driven by a pressure gradient ∇p in x direction and a constant magnetic field is applied throughout the flow field, i.e.,.

$$\nabla p = \frac{\partial p}{\partial x}\hat{i} + \frac{\partial p}{\partial y}\hat{j} + \frac{\partial p}{\partial z}\hat{k} \tag{5}$$

In x direction-

$$\nabla p = \frac{\partial p}{\partial x}\hat{i} \tag{6}$$

The induced current density for fluid velocity u is obtained by Ohm's law-

$$\frac{\partial \vec{B}}{\partial t} + (\vec{u}.\nabla)\vec{B} = \frac{1}{\mu\sigma}\nabla^2\vec{B} + (\vec{B}.\nabla)\vec{u} \tag{7}$$

From solved magnetic field $\vec{B}$, the current density $\vec{j}$ can be calculated using ampere's relation as-

$$\vec{j} = \frac{1}{\mu}\nabla \times \vec{B} \tag{8}$$

The governing equations reduce to Navier-Stokes system with the additional Lorentz force-

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u}.\nabla)\vec{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\vec{u} + \frac{1}{\rho}(\vec{j}\times\vec{B}) \tag{9}$$

and

$$\nabla.\vec{u} = 0 \tag{10}$$

5. FLUENT-THE CFD TOOL

The CFD tool that we have used is a robust CFD code – FLUENT14 for simulating and solving open channel and open channel contractions under different test conditions. We have tested some cases in open channels for their flow structure. Below we provide summary of the code and its specifications. FLUENT14 is a simulator to solve the CFD problems and the solving methodology is by finite difference method. FLUENT14 is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT14 provides complete mesh flexibility, including the ability to solve our flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral/tetrahedral/hexahedral/pyramid/w-edge, and mixed (hybrid) meshes. FLUENT14 also allows refining or coarsening the grid based on the flow solution.

5.1 The $k-\varepsilon$ model

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k-\varepsilon$ model in FLUENT falls within this class of turbulence model. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism (Launder 1972).

The standard $k-\varepsilon$ model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε).The model transport equation for k is derived from the exact equation, while the model transport equation for k was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In general, when the turbulence kinetic energy is needed for modeling a specific term, it is obtained by taking the trace of the Reynolds stress tensor:

$$k = \frac{1}{2}\overline{u_i'}\overline{u_i'} \tag{11}$$

In the derivation of the $k-\varepsilon$ model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard $k-\varepsilon$ model is therefore valid only for fully turbulent flows.

5.2 Introduction to UDF

A user-defined function (UDF), is a function that is programmed which can be dynamically loaded with the FLUENT solver to enhance the standard features of the code. For example, we can use a UDF to define our own boundary conditions, material properties, and source terms for associate flow regime, as well as specify customized model parameters(e.g., DPM, multiphase models), initialize a solution, or enhance post-processing. UDFs are written in the C programming language using any text editor and the source code file is saved with a .C extension (e.g., mhdudf.c). One source file can contain a single UDF or multiple UDFs, and we can define multiple source files. UDFs are defined using DEFINE macros provided by Fluent Inc. They are coded using additional macros and functions also supplied by Fluent Inc. that access FLUENT solver data and perform other tasks. Every UDF must contain the udf.h file inclusion directive (#include "udf.h") at the beginning of the source code file, which allows definitions of DEFINE macros and other Fluent-provided macros and functions to be included during the compilation process.

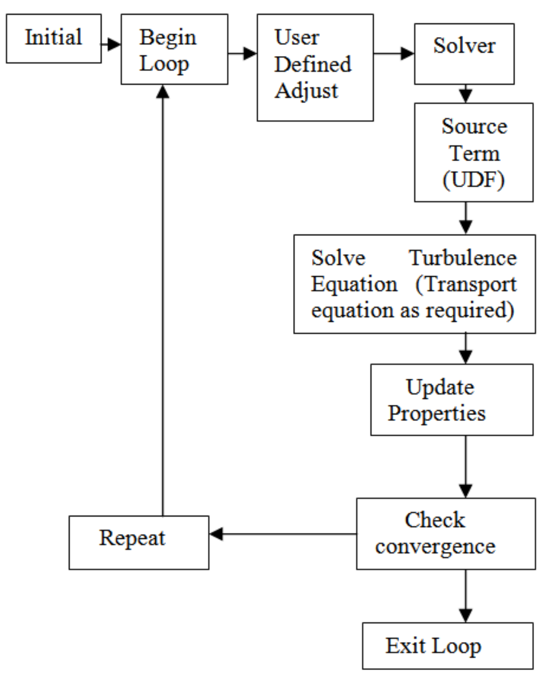

Fig. 2: Flow chart of the compiling the UDF

The values that are passed to a solver by a UDF or returned by the solver to a UDF are specified in SI units. Source files containing UDFs can be either interpreted or compiled in FLUENT. For interpreted UDFs, source files are interpreted and loaded directly at runtime, in a single- step process. For compiled UDFs, the process involves two separate steps. A shared object code library is first built and then it is loaded into FLUENT. Once interpreted or compiled, UDFs will become visible and selectable in FLUENT graphics panels, and can be hooked to a solver by choosing the function name in the appropriate panel.

6. RESULT

In order to ensure the fully developed flow, we have to validate our result with Nezu and Nakagawa experimental data for flow depth of 0.20 m and mean velocity 0.295m/sec. The flow Reynolds number 58,000.

From figure.3(a)-(c) shows the normalized stream wise mean velocity, log law and turbulent intensity. The stream wise mean velocity profile is quite good as comparing with Nezu and Nakagawa result. The standard log-law from plot (b). The turbulence intensity in stream wise direction is linearly decreased from bed to free surface.

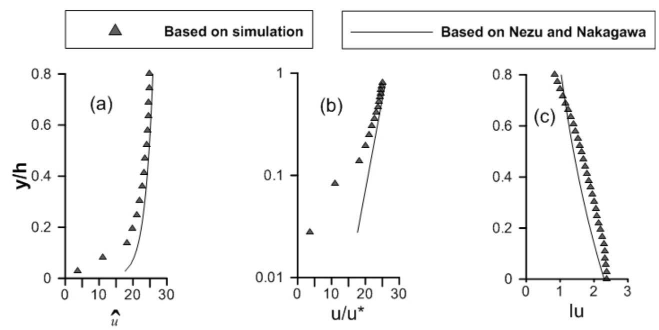

Fig.3: Streamline velocity, log-law and turbulence intensity

Now we are focusing on roughness of hemispherical bluff body of 0.03m diameter placed at 12m apart from the inlet of the open channel. We keep our focus on mean velocity profile, turbulence intensity and turbulence kinetic energy.

6.1 Mean Velocity

From figure.4 plot(a) we have seen that the plot of stream wise mean velocity as a function of vertical distance (y) for plane rigid bed at fully developed region (0.05m downstream from bluff body) is in good agreement with the previous investigations (e.g., Nezu and Nakagawa 1993). By placing the bluff body at 12m and measuring the stream wise velocity profile, the result was that the velocity increases in viscous sub layer as compared to that of plane bed. Further taking the velocity profile at 0.05m upstream from bluff body, a wake region is formed due to roughness (hemisphere).

6.2 Turbulence Intensity

Turbulence intensity (I_u) profile for open channel case is linearly decreased from bed to free surface, but placing the bluff body it results slightly different. From figure.4 plot (d) & (f), the I_u increased up to y = 0.03m and then linearly decreased up to free surface. At the bluff body position, profile starts from the upper surface of the curved face. The free surface value is approximately same as for open channel flow.

6.3 Turbulence Kinetic Energy

Turbulence Kinetic Energy (k) profile from figure.4 (g) & (h), upstream and downstream positions from the bluff body, we have seen for open channel case the profiles are linear at all three positions. Introducing the bluff body, the profiles first increased to y = 0.028m and 0.04m for downstream and upstream position (from bluff body) respectively, then decreased up to free surface.

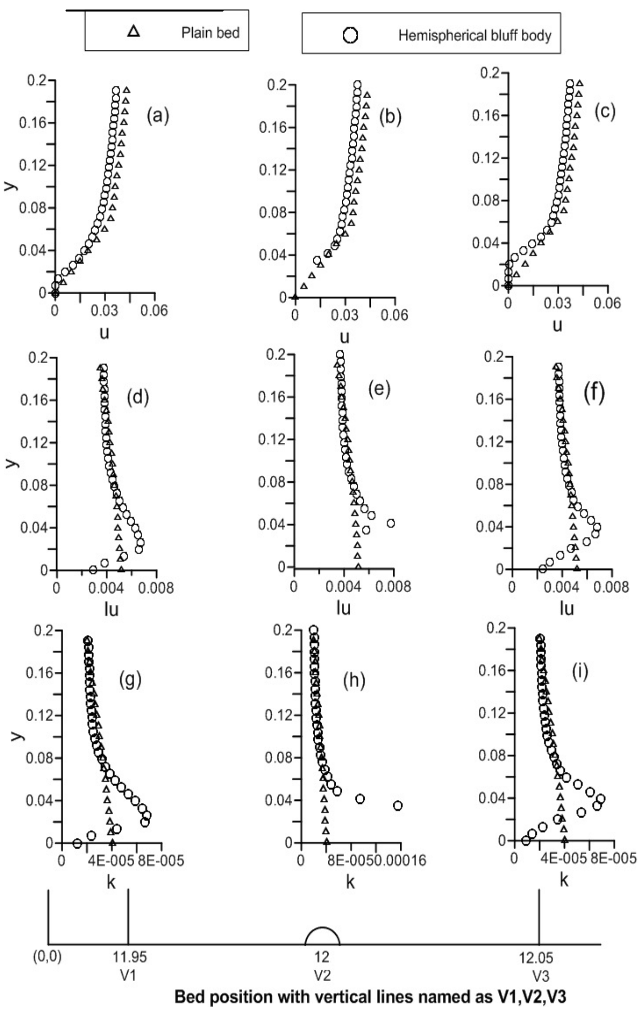

Fig. 4: Comparison between Mean velocity, TI & TKE at various positions of plain bed

6.4 Hartmann number

Hartmann number (*Ha*) is the ratio of electromagnetic force to the viscous force.

$$Ha = Bl\sqrt{\sigma/\mu_0}$$

For present study we have done simulation for three Hartmann number Ha = 5, 8, 10. In figure.5 three different positions V1, V2, V3. The hemisphere is placed at V2 position. The mean velocity, turbulence intensity and turbulent kinetic energy comparison with applied magnetic field is shown in figure.5.

We observed from figure.5 (a, b, c) for all positions(V1, V2, V3) the mean velocity magnitude gets reduced as the Hartmann number increases. Similarly from Turbulence intensity (d, g, e) and Turbulence kinetic energy (g, h, i) also gets reduced by increasing the magnetic field magnitude.

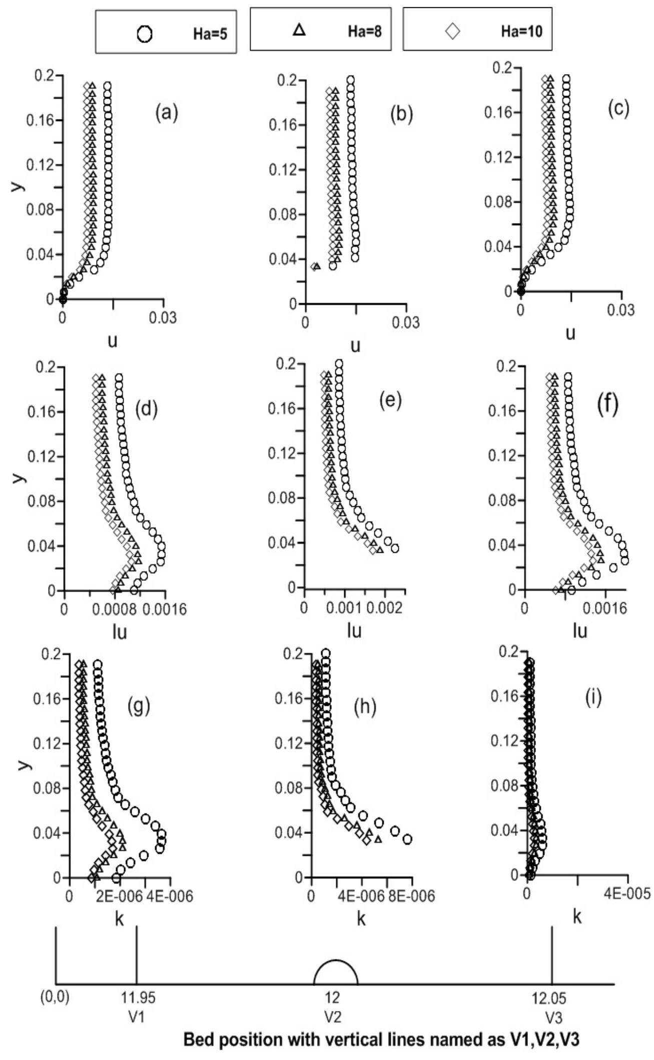

Fig.5: Comparison at different positions of plain bed for Ha=5, 8, 10

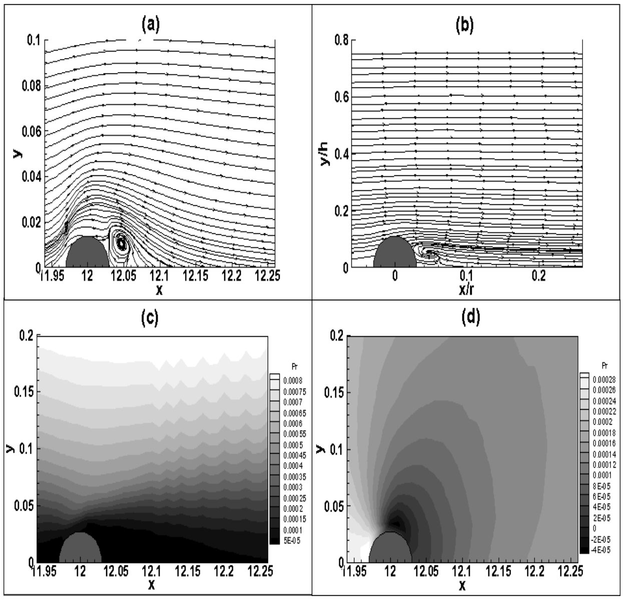

Fig.6: Flow Streamline, Normalized streamlines, Dynamic pressure & Static pressure contours

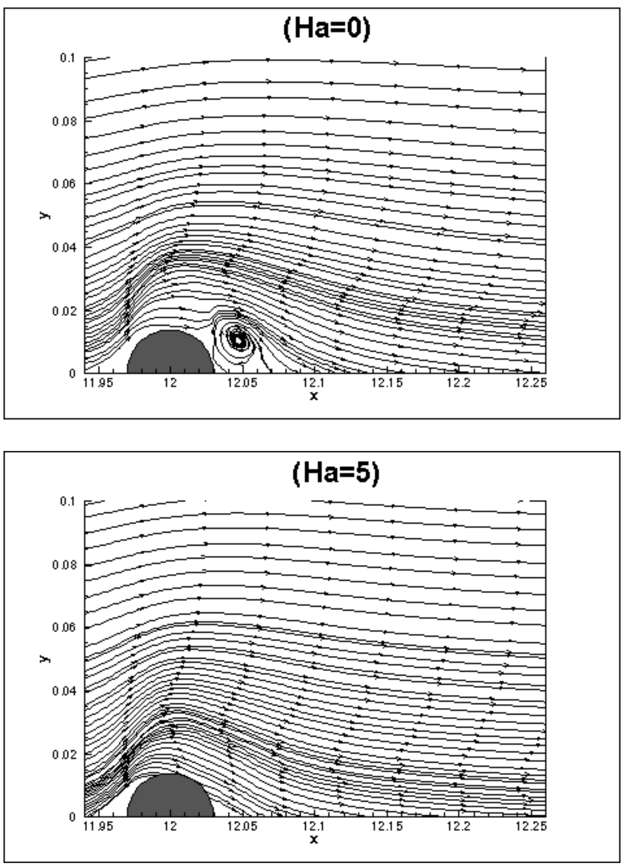

Fig.6: (b) Streamline for Ha=0 and 5

A Numerical study for incompressible flow over cylinder by Badami et al. [9] results that the length of wake can be reduced with the use of splitter plates. Another investigation by Mittal and Kumar [10] for the flow past over a rotating cylinder. But in our study the bluff body is stationary and from figure.6, it is seen that placing a bluff body in a channel flow creates wake near it at *Re* = 9950. As we applied the magnetic field the wake is quietly reduced as shown above in figure.6(b). The mean velocity at upstream position is negative due to the wake generation. As moving ahead from the bluff body, the flow streamlines are linear. The free surface is not influenced by the wake.

7. ACKNOWLEDGEMENT

In performing this study, we had to take the help and guideline of some respected persons, who deserve a greatest gratitude. The completion of this paper gives me much pleasure. After a time span we are able to present the study. We would like to show my gratitude to Mr. Santosh Kumar Singh and Mr. Krishnendu Barman for their support.

REFERENCES

[1] Thess, A.; Zikanov, O. “”Direct numerical simulation as a tool understanding MHD liquid metal turbulence”, Elsevier Applied Mathematical Modelling, 2004, 28, 1-13

[2] Yamamoto, Y.; Kunugi, T.; Satake, S.; Smolentsev, S. “DNS and $k-\varepsilon$ model simulation of MHD turbulent channel flows with heat transfer”, Elsevier Fusion Engineering and design. 2008, 83, 1309-1312

[3] Krasnow D.; Zikanov O.; Boeck T. “Numerical study for magnetohydrodynamics duct flow at high Reynolds and Hartmann numbers”, J. Fluid Mech. 2012, 704, 421-446

[4] Smolentsev, S. et al. “Application of the $k-\varepsilon$ model to open channel flows in a magnetic field”, IJES, 2002, 40, 603-711

[5] Young, J.; Smolentsev, S.; Abdou, M. “Study of instabilities in a quasi-2D MHD duct flow with an inflectional velocity profile”, Elsevier Fusion Engineering and design. 2014, 89, 1163-1167

[6] Rhoads, J.R.; Edlund, E.M.; Ji, H. “Effects of magnetic field on the turbulent wake of a cylinder in free-surface magnetohydrodynamic channel flow”, J. Fluid Mech. 2014, 742, 446-465

[7] Fornberg, B. “Steady incompressible flow past a row of a circular cylinders”, J. Fluid Mech.. 1991, 225, 655-671

[8] Mazumder, B.S.; Sarkar, K. “Turbulent flow characteristics and drag over 2-D forward-facing dune shaped structures with two different stross-side slopes”, Environ Fluid Mech., 2013,

[9] Badami P. et al. “Numerical Analysis of Flow past circular Cylinder with Triangular and Rectangular Wake Splitter”, World Academy of Science, Enginnering and Technology., 2012, 6, 9-20

[10] Mittal, S.; Kumar, B. “Flow past a rotating cylinder”, J. Fluid Mech. 2003, 476, 303-334