Journal of Fundamental and Applied Sciences

ISSN 1112-9867

Available online at http://www.jfas.info

Research Article

ANALYSIS OF A FULLY DEVELOPED LAMINAR FLOW B/W TWO PARALLEL PLATES SEPARATED BY A DISTANCE BY USING COMSOL MULTIPHYSICS

S. S. Raza

Department of Metallurgy and Materials Engineering, CEET, University of the Punjab, Lahore, Pakistan

Received: 23 August 2016 / Accepted: 28 December 2016 / Published online: 01 January 2017

ABSTRACT

This is a short analysis of flowing water in an infinite channel separated by a distance by using a Simulation Software Comsol Multiphysics. The flow behavior and the interaction with the boundary has been analysed. Wall no slip conditions were set for evaluation purpose. The analysis is a steady state analysis by using Incompressible Navier Stokes Model.

Keywords: Steady state analysis, Velocity profile, Fluid flow.

1. INTRODUCTION

It is often essential in thermal fluid engineering to move a working fluid from one place to another. During this transportation the fluid often comes into contact with the solid surfaces or with the boundaries which gives direction to the fluid. These surfaces may include the walls of the fluid through which fluid is flowing or the surfaces over which the fluid flows like body of an automobile or wings of an aircraft.

An interaction may involve the transfer of energy and momentum b/w the boundary and the fluid and consequently transport specific properties of the fluid like thermal conductivity and viscosity. Gradients of fluid temperature and velocity appear as a result and this appearance is in a direction normal to the surface.

Author Correspondence, e-mail: s_shahzadraza@yahoo.com

doi: http://dx.doi.org/10.4314/jfas.v9i1.36

One can have determination of properties of fluid by a knowhow of these gradients in temperature and velocity as a function of position with respect to the boundary. A good interaction b/w the boundary and fluid generates higher values of the thermal conductivity and viscosity. The most immense interaction occurs in the close contact of the boundary which interrupts the motion of the fluid. As the distance from the boundary increases this influence almost diminishes and fluid flows without any hindrance and its motion can be very well described with the aid of Bernoulli equation. The effects of transport properties near the boundary are of peak importance in regulating the motion of fluid and its efficiency in thermal fluid system. An incompressible fluid is related to an uncoupled system so its mechanical and thermal aspects of flow behavior can be discrete [1, 2].

2. MATERIAL AND METHOD

The current analysis is of a fully developed laminar flow b/w two parallel plates separated by a distance. The aim is to study the outlet velocity field and its values at different points. Comsol Multiphysics is used to simulate and solve the problem. The material of interest is water.

Considering a two dimensional mode and I solve it for velocity field. I selected a steadystate analysis of Incompressible Navier-Stokes model from the Application Mode and doing the following steps.

The geometry is simply created by a rectangle with dimensions 5m*0.05m with equal axis in axes/grid settings. I defined the material properties through the Sub domain Settings dialog box by use of water properties with a density of 1000 Kg/m^3 and viscosity is 0.001 Pa.s.

I opted boundary conditions considering that the velocity gradient b/w 1&4 with inlet and outlet velocity respectively. Wall no slip conditions b/w the surfaces 2 and 3 were set for evaluation [2]. Solve mode was initiated for simulation of the work.

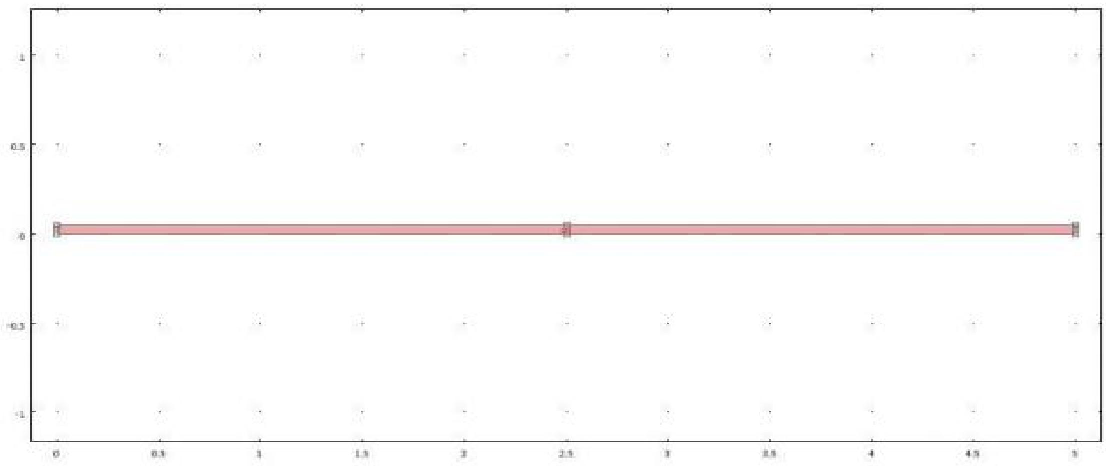

Fig.1. Geometry of the channel with given dimensions

3. RESULTS AND DISCUSSION

Following are the results generated from Comsol Multiphysics for surface and contour plots of velocity flows.

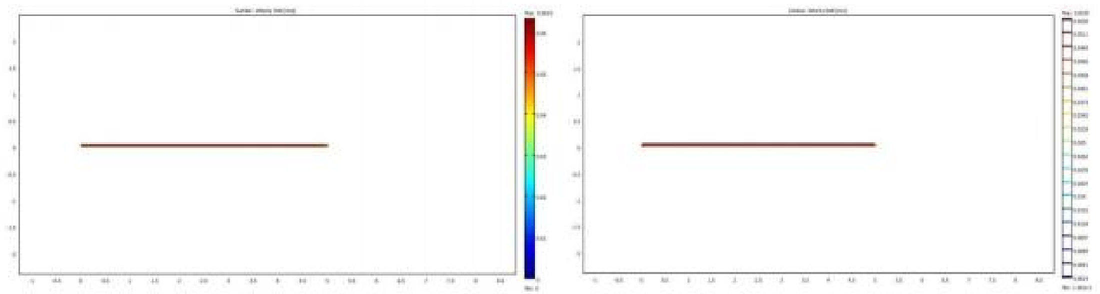

Fig.2. 2-D Surface Plot and Contour Plot for Velocity Flow

The above plots show the velocity profile in the channel with given dimensions after doing the simulation. The colour scheme shows the velocity values. These are the results after complete simulation.

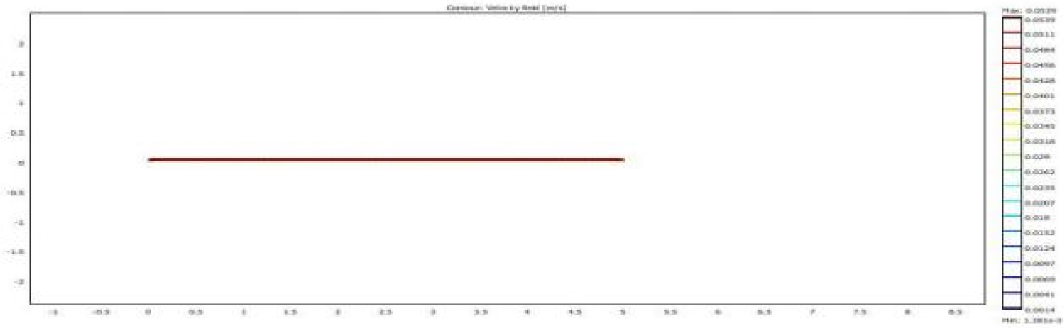

Fig.3. 2D Contour Plot

The outlet velocity profile can be noticed simply by the above 2-D contour plot. The right hand side axes give us estimation about the inner and outer velocity values at different points.

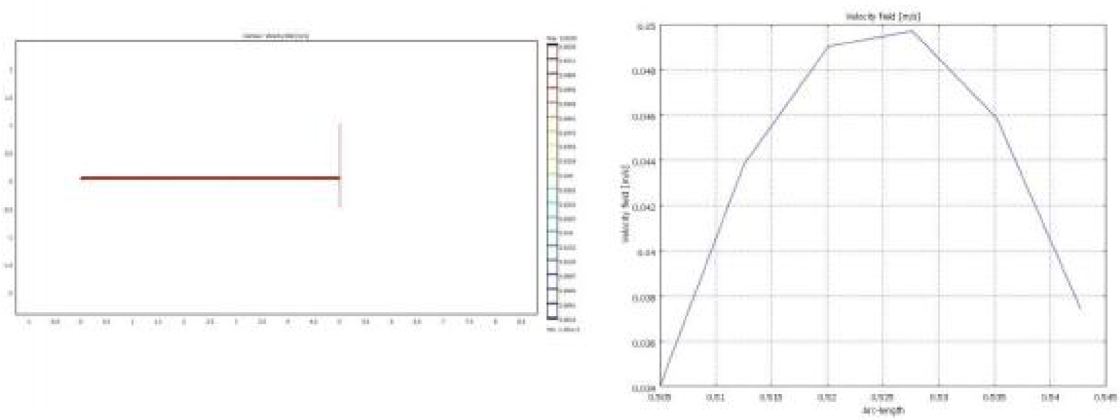

Fig.4. 2D Contour and Cross Section Plot with a line at the outlet

At the start of the flow the flow field lines follow a parabolic path but as the flow progress the flow front becomes a fountain flow the fluid near the center line catches up to the flow front and fountains out toward the walls hence increases its velocity in the centre at the outlet. I have used steady state analysis in the current situation. Since, I was only interested to find the velocity as a function of position and not time. So in current scenario the solution would be time independent. A line was drawn at a distance of 5m on the x-axis which is almost at the outermost contour (outlet). We draw a cross section plot here and take 5 equispaced points here and notice the outlet velocity values.

First point---------0.044 m/sec

Second point-------0.049 m/sec

Third point---------0.0498 m/sec

Fourth point--------0.046 m/sec

Fifth point----------0.038 m/sec

4. CONCLUSIONS

The parabolic curve of the velocity profile shows an agreement with the theory. In the centre there is no hindrance to the fluid with the boundary so it shows a maximum value of the velocity of the fluid.

5. ACKNOWLEDGEMENTS

I am thankful for the support of Simulation and Modelling Lab staff at KTH, Sweden.

6. REFERENCES

1- Date, Anil W. Introduction to Computational Fluid Dynamics. Indian Institute of Technology Bombay. ISBN: 13: 978-0-521-85326-2.

2- mit.edu/snively/www/2_006%20Coursenotes.pdf (Available Online).

3- https://www.comsol.com/ (A licensed Simulation software available at KTH Simulation Lab).

How to cite this article:
Raza S. S. Analysis of a fully developed laminar flow b/w two parallel plates separated by a distance by using comsol multiphysics. J. Fundam. Appl. Sci., 2017, *9(1), 618-622.*