Case Study

Exact solutions of Navier-Stokes equations - compared with CFD simulations using Ansys Fluent® Software

Ravindra Arjun Shirsath

Developed and curated by the Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This case study uses Ansys Fluent®, the fluid simulation software.

Summary

The Navier-Stokes equations are partial differential equations which describe the motion of viscous fluid substances. They were named after French engineer and physicist Claude-Louis Navier and the Irish physicist and mathematician George Gabriel Stokes. They were developed over several decades of progressively building the theories from 1822 (Navier) to 1842-1850 (Stokes). The Navier-Stokes equations mathematically express momentum balance for Newtonian fluids and make use of conservation of mass. They are sometimes accompanied by an equation of state relating pressure, temperature and density. They arise from applying Isaac Newton's second law to fluid motion, together with the assumption that the stress in the fluid is the sum of a diffusing viscous term and a pressure term – hence describing viscous flow. The difference between them and the closely related Euler equation is that Navier-Stokes equations take viscosity into account while the Euler equations model only inviscid flow. As a result, the Navier-Stokes are parabolic equations and therefore have better analytic properties, at the expense of having less mathematical structure.

The principal difficulty in solving the Navier-Stokes equations arises from the presence of the nonlinear convective term. Since there are no general analytical methods for solving nonlinear partial differential equations, each problem must be considered individually. For most practical flow problems, convective acceleration of fluid particles cannot be ignored. However, in general, exact solutions are possible only when the nonlinear terms vanish identically. There are a few special cases for which the convective acceleration vanishes because of the nature of the geometry of the flow system. In these cases, exact solutions are usually possible, and few such problems are considered here. In the present case study, exact solutions of Navier-Stokes equations for Couette flow and Poiseuille flow are derived first. Followed by steady state simulations using Ansys Fluent software for the same to show the accuracy of the simulations. The investigations are further extended to understand the effect of increasing inlet velocity on all the examples. The results include the flow visualization showing velocity profiles under different conditions for enhanced understanding of basic principles. The present study will serve as a starting point for mechanical, aeronautical, and aerospace engineering students to better understand the fundamental concepts associated with viscous flow.

Table of Contents

1. Introduction

The governing equations for the flow of viscous incompressible fluid are the Navier-Stokes equations. The exact solution of the complete Navier-Stokes equations is a million-dollar problem even today, however under certain simplifying assumptions, it is possible to have the exact solutions. This section briefly explains the possible examples under which exact solutions can be obtained and the associated simplifying assumptions.

The conservation of mass and conservation of momentum equations in cartesian co-ordinate system are represented as follows –

Conservation of mass - $\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0$ (1)

x-momentum equation - $\rho \left[\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right] = \rho g_x - \frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$ (2)

y-momentum equation - $\rho \left[\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right] = \rho g_y - \frac{\partial p}{\partial y} + \mu\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right]$ (3)

z-momentum equation - $\rho \left[\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right] = \rho g_z - \frac{\partial p}{\partial z} + \mu\left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right]$ (4)

The underlying assumptions involved to simplify these equations to be able to solve them analytically are as follows –

- **Steady flow –** Steady flow is defined as that type of flow in which characteristics of fluid like velocity, pressure, density etc. at a point do not change with respect to time, thus mathematically, for a steady flow –

$$\frac{\partial \rho}{\partial t} = 0,\ \frac{\partial u}{\partial t} = 0,\ \frac{\partial p}{\partial t} = 0 \qquad (5)$$

- **Two dimensional flow –** Two dimensional flow in x – y plane (say) is defined as that type of flow in which all the flow parameters are functions of time and two space co-ordinates only, say x and y. The variation in the third direction say z is zero. Thus mathematically, for two dimensional flow in x – y plane,

$$\begin{aligned} u &= f_1(x, y, t) \\ v &= f_2(x, y, t) \\ w &= 0 \\ \frac{\partial(\text{any property})}{\partial z} &= 0 \end{aligned} \qquad (6)$$

- **Fully developed flow –** The flow is said to be fully developed when the mean velocity profiles are independent of position along the direction of the flow.
- **Laminar flow –** If the flow is smooth and if the layers in the flow do not mix macroscopically then the flow is called laminar flow. In laminar flow layers will glide over each other without mixing.

- **Incompressible flow –** Incompressible flow is characterized by the fact that the density of the fluid remains approximately constant as it flows, and it does not change significantly in response to changes in pressure or temperature. It is typically observed at low speed, thus mathematically for incompressible flow –

$$\rho = constant$$

2. Problem Statement

In the present work, the CFD simulations are performed for investigating the flow properties of a laminar viscous flow between infinite parallel plates separated by a small gap as shown schematically in Fig. (1). The investigations are intended to understand the standard exact solutions of the Navier-Stokes equations under the simplifying assumptions as stated above. The flow visualization using CFD is intended for enhanced understanding of the popular basic flows such as Couette flow and Plane Poiseuille flows.

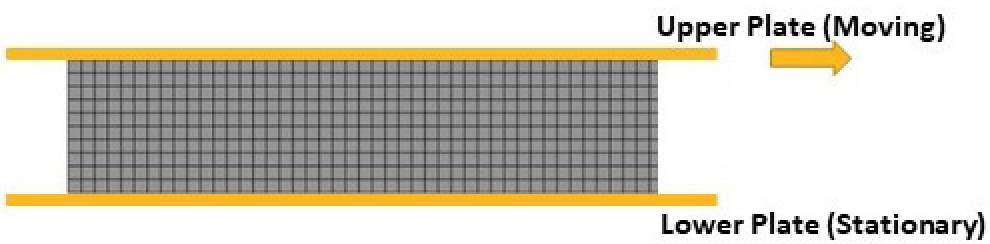


Figure 1. Schematic of flow between parallel plates.

3. Geometry and Mesh

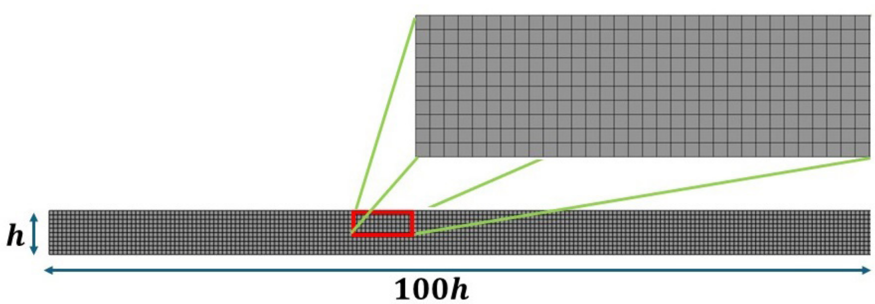


Figure 2. Details of the geometry, mesh and computational domain.

The computational domain considered for present work is 10mm tall and 1000mm long with the fluid region enclosed between upper and lower plates as shown schematically in Fig. 1. The mesh used for present study has **≈11000** nodes resulting in **≈10000** elements. Highly refined mesh is preferred to ensure that near wall phenomenon is captured correctly. The mesh contains only quadrilateral elements resulting in element quality **≈1**, aspect ratio **=1**, skewness **≈0**, and orthogonal quality as seen from Fig. (2) resulting in quick convergence.

4. Solution Methodology

Using the Ansys Fluent tool, the following methodology is used. Numerical solver is set up with pressure-based type, absolute velocity formulation and steady state simulation. The flow is assumed to be viscous laminar hence no turbulence model is required. The working fluid is chosen to have density ρ=1.225 kg/m^3, while the coefficient of dynamic viscosity, μ =1.7894x10^{-4} kg/m s(an order of magnitude higher than air for enhanced viscous effects). Pressure-velocity coupling is dealt with SIMPLE algorithm while least square cell-based method is employed for estimating the gradients and second order upwind scheme is used for discretizing the momentum equations. The inlet is specified to be velocity inlet with the required magnitude of velocity as necessary. The bottom is specified to be stationary wall with no-slip condition while the top boundary is specified to be stationary wall with no-slip condition or moving wall with the desired velocity depending on the type of flow to be simulated. The outlet is set to be a pressure outlet. Zero-gauge pressure is initialized throughout. The criterion for convergence is set to be 10^{-5} and the simulations are run for sufficient number of iterations.

5. Results and Discussion

This section explains the analytical procedure adapted for solving the Navier-Stokes equations under the assumptions discussed in the previous section along with the computational approach. The results from both the approaches are compared and are found to be in good agreement with each other for the three different types of the flows namely – a) Simple Couette flow b) Couette flow and c) Plane Poiseuille flow. Additionally flow visualization from CFD simulations is included for enhanced understanding of the involved flow physics.

5.1 Simple Couette Flow –

The Simple Couette flow is the flow of viscous fluid in the space between two surfaces, one of which is moving tangentially relative to the other as shown schematically in Fig. (3). The relative motion of the surfaces imposes shear stress on the fluid and induces flow.

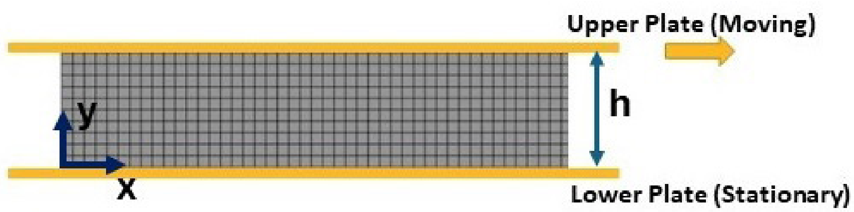


Figure 3. Schematic of simple Couette flow.

The Couette flow is frequently used in undergraduate physics and engineering courses to illustrate shear-driven fluid motion. A simple configuration corresponds to two infinite, parallel plates separated by a distance h, one plate translates with a constant velocity u in its own plane. Neglecting the pressure gradients, the Navier-Stokes equations simplify to –

$$\text{Conservation of mass - } \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (1)$$

$$\text{Conservation of mass - } \frac{\partial \rho}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

$$\Rightarrow \frac{\partial v}{\partial y} = 0 \Rightarrow v \neq f(y) \text{ but since } v = 0 \text{ at the wall, } v = 0 \textbf{ EVERYWHERE.}$$

$$\text{x-momentum equation - } \rho\left[\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right] = \rho g_x - \frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \quad (2)$$

$$\Rightarrow 0 = -\frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial y^2}\right]$$

y-momentum equation - $\rho\left[\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right]=\rho g_y-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]$ (3)

$\Rightarrow 0=-\frac{\partial p}{\partial y}\Rightarrow p\neq f(y)$

z-momentum equation - $\rho\left[\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right]=\rho g_z-\frac{\partial p}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]$ (4)

$\Rightarrow 0=-\frac{\partial p}{\partial z}\Rightarrow p\neq f(z)$

$\Rightarrow$ Thus $\Rightarrow p=f(x)$ only therefore $\frac{\partial p}{\partial x}=\frac{dp}{dx}$

$\Rightarrow 0=-\frac{dp}{dx}+\mu\left[\frac{\partial^2 u}{\partial y^2}\right]$

$$\boxed{\Rightarrow 0=-\frac{dp}{dx}+\mu\left[\frac{\partial^2 u}{\partial y^2}\right] \qquad \textbf{(A)}}$$

In the absence of the pressure gradient, ***dp/dx=0*** , the above equation simplifies to –

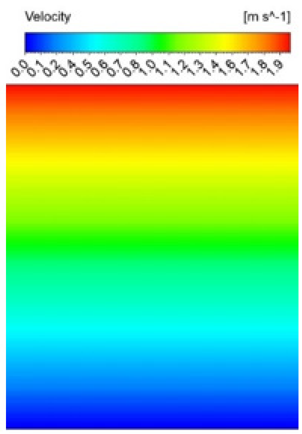
$$\Rightarrow \frac{\partial^2 u}{\partial y^2}=0 \tag{5}$$

Where y is a spatial coordinate normal to the plates and u(y) is the velocity distribution. If y originates at the lower plate, the boundary conditions are

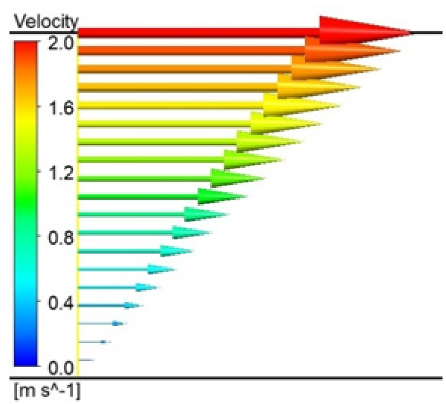
$$At\ y=0, u=0$$
$$At\ y=h, u=u_\infty$$

Integrating equation (5) twice and substituting the above boundary conditions, we get –

$$\frac{\partial u}{\partial y}=C_1$$
$$u=C_1y+C_2$$
$$At\ y=0, u=0\Rightarrow 0=C_1(0)+C_2\Rightarrow C_2=0$$
$$At\ y=h, u=u_\infty\Rightarrow u_\infty=C_1(h)+0\Rightarrow C_1=\frac{u_\infty}{h}$$
$$\Rightarrow u=\frac{u_\infty}{h}y\Rightarrow\frac{u}{u_\infty}=\frac{y}{h}$$

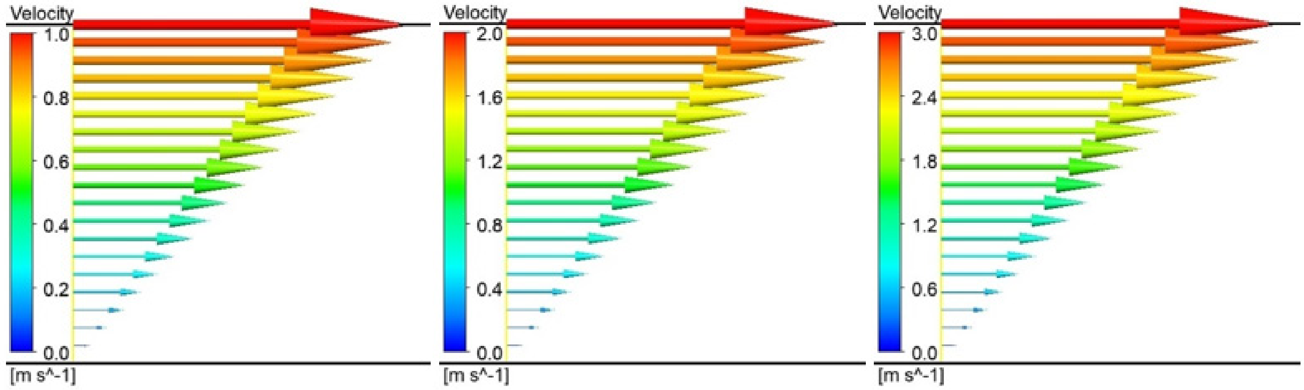


(a) Contours of velocity.



(b) Velocity vectors.

Figure 4. Variation of velocity at u_∞***=2 m/s***.



(a) $u_\infty = 1\,m/s$. (b) $u_\infty = 2\,m/s$. (c) $u_\infty = 3\,m/s$

Figure 5. Velocity variation for different boundary conditions.

As can be seen from Fig. (4), the velocity varies linearly from zero at the lower plate to the plate velocity of the upper plate as also seen from analytical solution. The effect of increasing upper plate velocity is shown in Fig. (5).

5.2 Couette Flow –

A more general Couette flow situation arises when a pressure gradient is imposed in a direction parallel to the plates as seen from Fig. (6). The Navier-Stokes equations, in this case, simplify to -

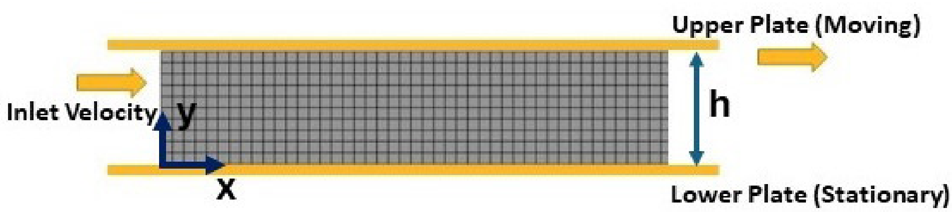


Figure 6. Schematic of Couette flow.

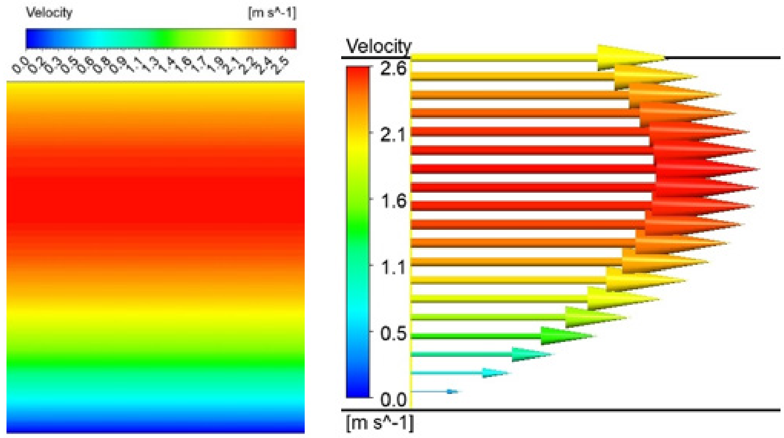
$$\Rightarrow 0 = -\frac{dp}{dx} + \mu\left[\frac{\partial^2 u}{\partial y^2}\right] \qquad \text{(A)}$$

The boundary conditions are

$$At\ y = 0, u = 0$$
$$At\ y = h, u = u_\infty$$

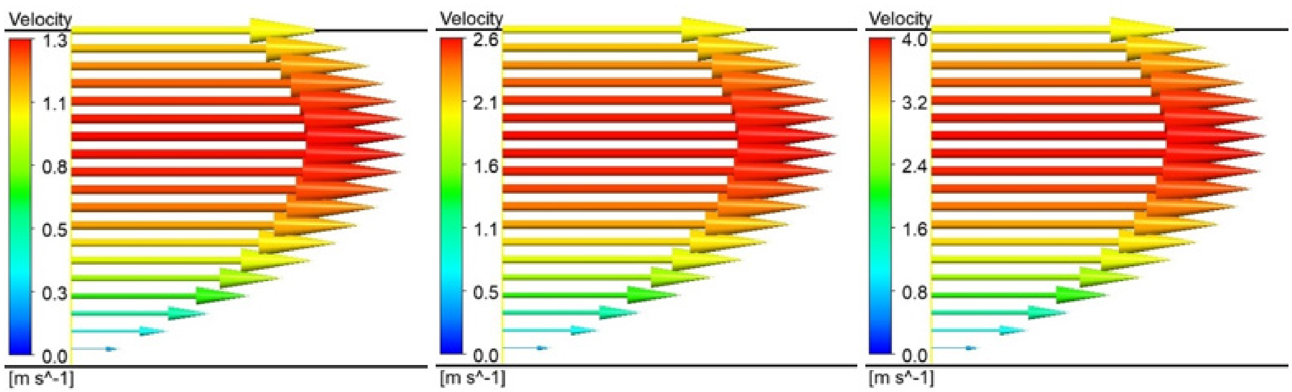
Integrating equation (5) twice and substituting the above boundary conditions, we get –

$$\frac{\partial u}{\partial y} = \frac{1}{\mu}\frac{dp}{dx}y + C_1$$
$$u = \frac{1}{\mu}\frac{dp}{dx}\frac{y^2}{2} + c_1 y + c_2$$
$$At\ y = 0, u = 0 \Rightarrow 0 = 0 + C_1(0) + C_2 \Rightarrow C_2 = 0$$
$$At\ y = h, u = u_\infty \Rightarrow C_1 = \frac{u_\infty}{h} + \frac{1}{2\mu}\left(-\frac{dp}{dx}\right)h$$
$$\Rightarrow \frac{u}{u_\infty} = \frac{y}{h} + \frac{h^2}{2\mu u_\infty}\left(-\frac{dp}{dx}\right)\frac{y}{h}\left(1 - \frac{y}{h}\right)$$



(a) Contours of velocity. (b) Velocity vectors.

Figure 7. Variation of velocity at u_∞=2 m/s.



(a) $u_\infty = 1\, m/s$. (b) $u_\infty = 2\, m/s$. (c) $u_\infty = 3\, m/s$

Figure 8. Velocity variation for different boundary conditions.

While both the shear and pressure gradient being the driving forces for the flow, the velocity varies from zero at the lower plate to the velocity same as that of the upper plate but passes through a maxima in between due to the effect of the pressure gradient as seen from Fig. (7) and also observed from the analytical solution. This maxima depends on the magnitude of pressure gradient. Further effect of increasing the inlet velocity is shown in Fig. (8).

5.3 Plane Poiseuille Flow –

Plane Poiseuille flow is defined as a steady, laminar flow of a viscous fluid between two horizontal parallel plates separated by a distance, h. Flow is induced by a pressure gradient across the length of the plates and is characterized by a 2D parabolic velocity profile symmetric about the horizontal mid-plane as illustrated in Fig. (9).

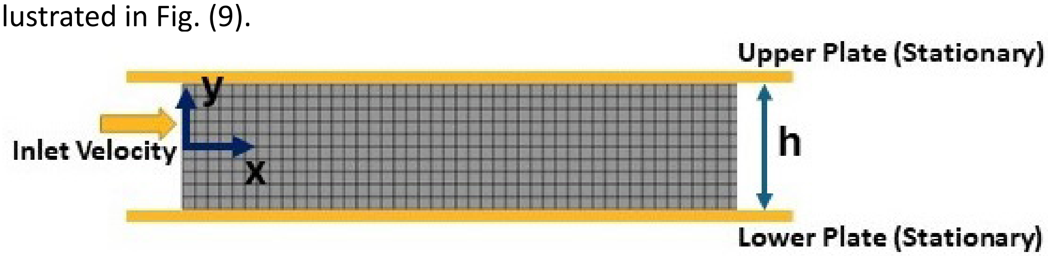


Figure 9. Schematic of Plane Poiseuille flow.

$$\Rightarrow 0 = -\frac{dp}{dx} + \mu\left[\frac{\partial^2 u}{\partial y^2}\right] \qquad \textbf{(A)}$$

In this problem, the Navier-Stokes equations reduce to a second order, linear, ordinary differential equation.

The boundary conditions are

$$At\ y = \pm {}^{h}/_{2},\ u = 0$$

$$At\ y = 0, \frac{du}{dy} = 0$$

Integrating equation (5) twice and substituting the above boundary conditions, we get –

$$\frac{\partial u}{\partial y} = \frac{1}{\mu}\frac{dp}{dx}y + C_1$$

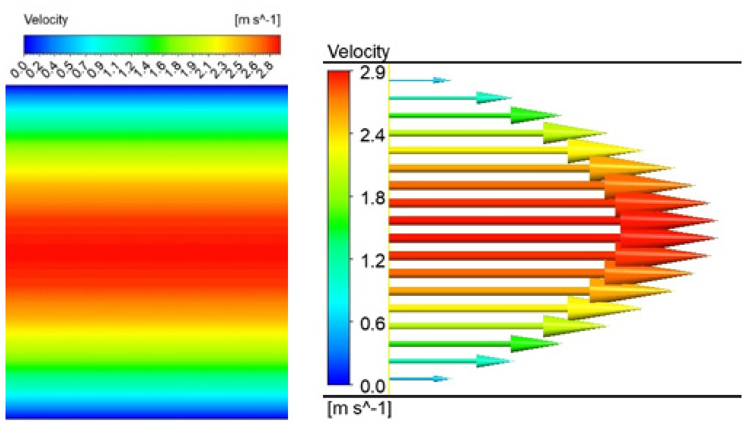
$$u = \frac{1}{\mu}\frac{dp}{dx}\frac{y^2}{2} + c_1 y + c_2$$

$$At\ y = 0, \frac{du}{dy} = 0\ \Rightarrow C_1 = 0$$

$$At\ y = \pm {}^{h}/_{2},\ u = 0 \Rightarrow C_2 = \frac{h^2}{8\mu}\left(-\frac{dp}{dx}\right)$$

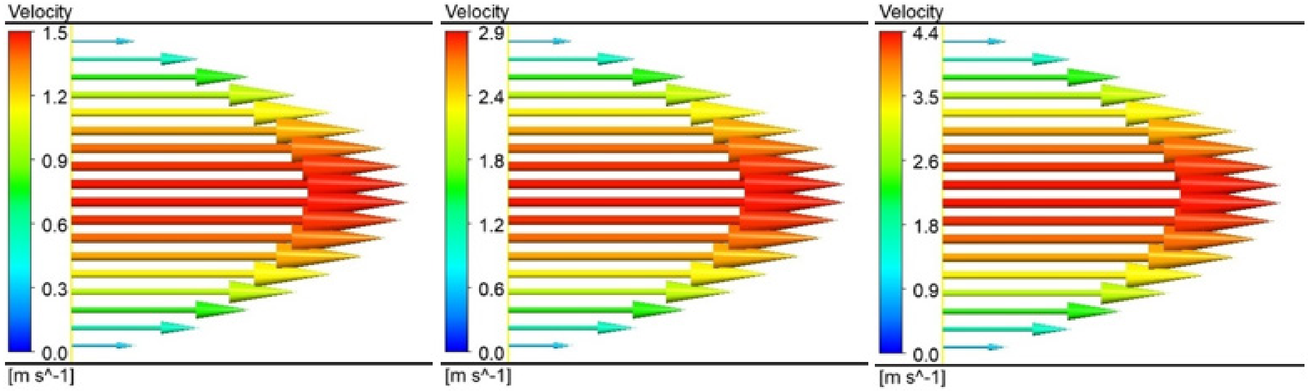
$$\Rightarrow u = \frac{h^2}{8\mu}\left(-\frac{dp}{dx}\right)\left(1 - \frac{4y^2}{h^2}\right)$$

$$\Rightarrow u = \frac{h^2}{8\mu}\left(-\frac{dp}{dx}\right)\left(1 - \frac{y^2}{(h/2)^2}\right)$$



(a) Contours of velocity. (b) Velocity vectors.

Figure 10. Variation of velocity at $\boldsymbol{u_\infty = 2\ m/s}$.



(a) $u_\infty = 1\, m/s$. (b) $u_\infty = 2\, m/s$. (c) $u_\infty = 3\, m/s$

Figure 11. Velocity variation for different boundary conditions.

This is the famous parabolic velocity profile with zero velocity on either of the plates while it is maximum at the centerline as shown in Fig. (10). The magnitude of maximum velocity again depends on the applied pressure gradient. Increasing the inlet velocity only changes the magnitude of velocity throughout the domain while it remains parabolic as seen from Fig. (11).

6. Further Steps

In the present case study, the exact solutions of the Navier-Stokes equations under the listed assumptions are compared with that obtained from CFD simulations using Ansys Fluent software. The results show a good agreement in representing the underlying physics and hence can be demonstrated for enhanced understanding of the fluid mechanics concepts. These steady state investigations show preliminary analysis and the same can be extended to further investigate the effect of different inlet velocities and pressure gradient for both magnitude and directions.

7. References

1. F. M. White, “Fluid Mechanics”, 7th Edition, McGraw-Hill, New York, 2011.

2. R. W. Fow, A. T. McDonald and P. J. Pritchard, “Introduction to Fluid Mechanics”, 6th Edition, John Wiley and Sons, Inc., New York, 2004.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com