



# Case Study

## Uniform Flow around a cylinder using Ansys Fluent

Developed and curated by the Ansys Academic Development Team

Gautham Varma Raja Kochanattu

[education@ansys.com](mailto:education@ansys.com)

# Summary

Ansys Fluent is a Computational Fluid Dynamics (CFD) software that is used to solve fluid flow, heat and mass transfer, chemical reactions etc. It uses a user-friendly interface that helps in modernizing the process of computation from pre-processing to post processing. It uses advanced physical models like turbulent modelling, multiphase modelling battery modelling, combustion, fluid-structure interactions to solve the given problem to high level of accuracy.

In this case study, an understanding, and assumptions of potential flow around a cylinder will be described and we will try to visualize such a flow using Ansys Fluent. Further the real-life implications of such a flow will also be discussed with varying Reynolds numbers and understanding how the flow will be developed for various flow speeds.

# Table of Contents

1. Introduction.....	3
2. Problem Statement .....	3
3. Physics Behind the calculations.....	3
Potential Flow Theory.....	3
4. Geometry and Meshing.....	4
5. Results and Discussion .....	5
6. Conclusions.....	7
7. References.....	7

## 1. Introduction

Reynolds number is a dimensionless quantity that is used to determine the type of flow regime (laminar, transition or turbulent). It can be defined mathematically as the ratio of inertial to viscous forces. The Reynolds number can be calculated using the formula.

$$Re = \frac{(\rho U D)}{\mu}$$

Where,

Re = Reynold's Number

$\rho$  = Density of the fluid

U = Velocity of the flow

D = Hydraulic diameter

$\mu$  = viscosity of the fluid

Depending on the value of this number, the flow can be categorized as Laminar, Transition or Turbulent.

## 2. Problem Statement

In this case study, we will discuss the importance of the Reynolds number in determining the flow regime. We will explore the potential flow theory and its derivations and assumptions. Further, simulations are carried out to understand and visualize the flow. The real-world implications of such a flow will be examined based on the Reynolds Number.

## 3. Physics Behind the calculations

### Potential Flow Theory

Potential flow or ideal flow, is defined by the gradient of the velocity field. Thus, the velocity of the potential flow can be described by the using the following (more details please refer to the [Simple Approximation of Fluid Flows Ansys Innovation Course](#)):

$$v = \nabla \phi$$

Where,  $\phi$  is the velocity potential and  $v$  is the flow velocity. From the vector mathematics, we can see that the

$$\nabla \times \nabla \phi = 0$$

*i.e.*, the curl of the gradient is equal to zero. This suggests that potential flow is always irrotational in nature. This potential flow can be used to describe various external flow parameters, where the vorticity is not prevalent. Now let us consider the super positioning of an inviscid, irrotational, incompressible and steady uniform flow and a doublet (Figure 1). The equations of the uniform flow can be defined as a function of stream and potential flows as

$$\phi = Ux, \quad \psi = Uy$$

Where  $x=r \cos\theta$  and  $y=r \sin\theta$  and the equation of the doublet can be defined by the equation:  
Hence the superimposed flow will have the equation:

$$\phi = \frac{m}{r} \cos\theta, \quad \psi = -\frac{m}{r} \sin\theta$$

Similarly, we also obtain the equations of the velocity component of the flow by finding the derivative of the potential function in each direction which produces:

$$V_r = \frac{\partial \phi}{\partial r} = U \left( 1 - \frac{m}{Ur^2} \right) \cos \theta, \quad V_\theta = \frac{1}{r} \frac{\partial \phi}{\partial \theta} = -U \left( 1 + \frac{m}{Ur^2} \right) \sin \theta$$

From the above equations, the stagnation points will be obtained when the values of velocity will be zero that is  $V_r=0$  and  $V_\theta=0$ ;

These stagnation points imply that from the equation of  $V_r$  and  $V_\theta$ , the following solutions are true:

$$r^2 = \frac{m}{U}; \quad \cos \theta = 0$$

And

$$r^2 = -\frac{m}{U}; \quad \sin \theta = 0$$

If we define  $m/U=R^2$ , the solution for the stagnation point becomes,  $r= +R, -R$  and  $\vartheta=0,\pi$ . Thus, suggesting that streamlines are nonexistent inside the circle with radius  $R$  (Figure 1). This problem then can be redefined by substituting stagnations points by a solid cylinder. Thus, the problem becomes an external flow around a cylinder and the equations that were defined above can be used to define the flow around the cylinder. This analytical solution is a simplified version of the flow and hence might not be an actual representation of the flow. Let us see in the next section how to define this solution using Ansys Fluent and how the results vary from this potential flow theory.

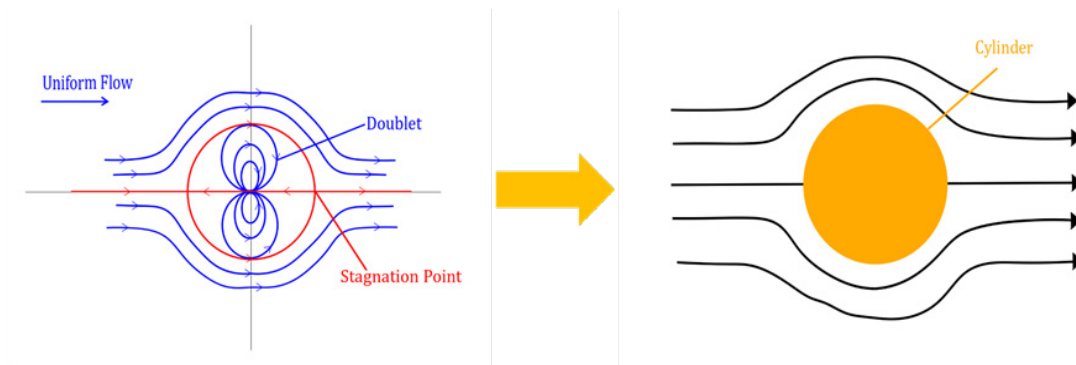


Figure 1 : Depiction of a superposed uniform flow and a doublet.

#### 4. Geometry and Meshing

A simple 2D geometry is used for the simulation, which is defined in Figure 2. The mesh was generated with a total number of nodes of 22301. This was obtained after a mesh sensitivity calculation. This mesh is used to develop the simulation so that the potential flow, defined in the previous section, can be checked. Further details on generating the mesh and mesh sensitivity in Ansys Fluent can be found in this [Ansys Innovation Course Learning Track](#). To understand the actual flow around the cylinder, a 2D transient solution is carried out to obtain the effect correctly, initially for inviscid flow and later for laminar viscous flow at Reynolds number of  $Re = 1, 30$  &  $100$  with corresponding inlet velocities of  $V_{in} = 0.00029, 0.0088$  and  $0.029$  m/s correspondingly. The surrounding fluid is air and the uniform flow velocities are changed to obtain different Reynolds numbers.

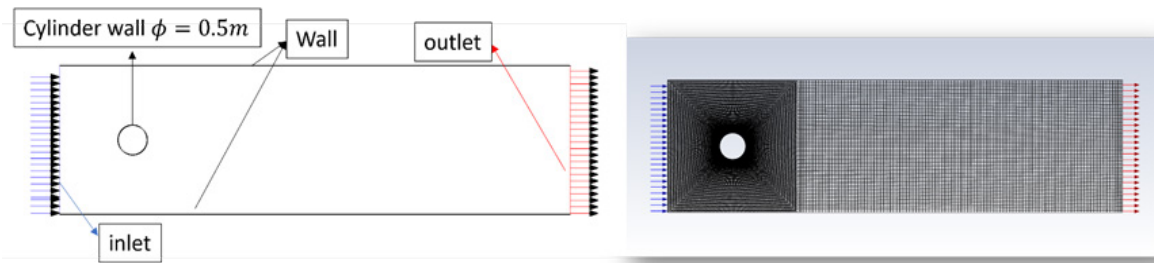


Figure 2: Physics domain with the cylinder for simulation and the mesh used for simulation.

## 5. Results and Discussion

The simulation is carried out for inviscid flow at an inlet velocity value of  $0.00029\text{m/s}$  (corresponding to  $Re=1$ ), see Figure 3. The flow follows a pattern that is expected from the previous section of potential flow theory at this low velocity. To check the values, Figure 4 presents the Coefficient of pressure distribution around the cylinder and is shown to have followed the ideal fluid theory as explained in (Nakayama, 1998). When this pressure distribution is integrated, the total pressure resistance becomes equal to zero, suggesting that there is no force or negligible force acts on the cylinder at this low velocity for an inviscid flow. The streamlines and pressure plots surrounding the cylinder match the predictions of potential flow theory, as explained in the previous section.

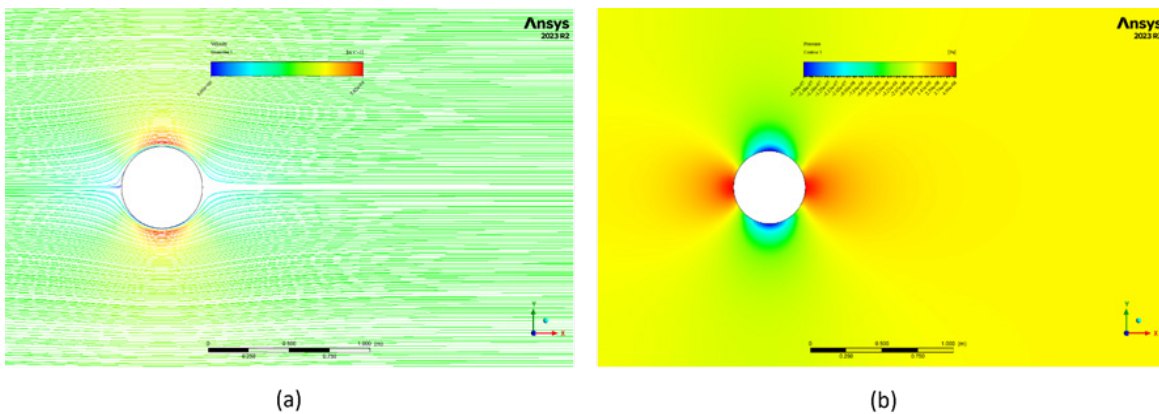


Figure 3 : (a) The streamline plot for an inviscid flow (b) Plot of Pressure for Inviscid Flow at  $0.00029\text{m/s}$

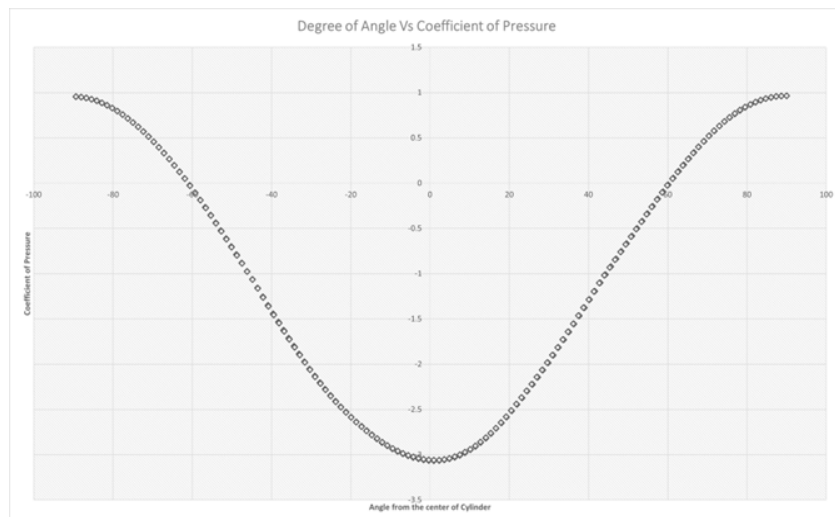


Figure 4: The Coefficient of Pressure distribution around the cylinder

However, in real life, viscosity plays an important role in the pressure distribution. Thus, when we calculate the flow of air around a cylinder for laminar viscous flow, the pressure distribution is quite different, even at this low Velocity corresponding to  $Re=1$  (Figure 5).

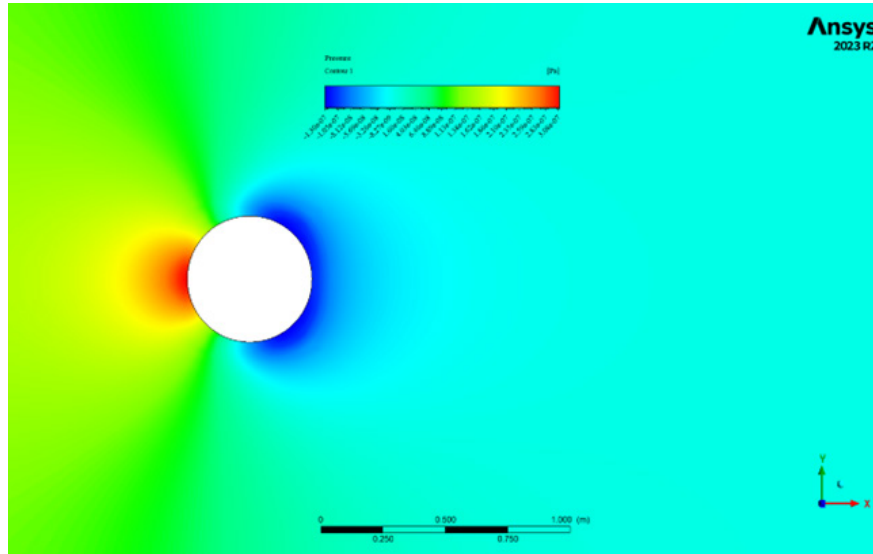


Figure 5 : Pressure distribution for a Laminar flow at  $Re=1$ .

It is worth noting that all the simulations have been carried out in the Transient model (time-dependent), hence the results shown is actually a snapshot of the flow at a particular time in the simulation. Keeping this in mind, Figure 6 shows how the vortices behind the cylinder develops as the Reynolds number increases from  $Re=30$  to  $Re=100$ . At  $Re=30$  Figure 7(a), the streamlines diverge around the cylinder symmetrically. The boundary layer at this speed separates symmetrically from the cylinder leading to two eddies that rotate in opposite directions as shown. Beyond this, the main streamlines will come together. The two eddies' formations are called twin vortices. As velocity increases, these vortices start elongating and will shed alternatively from each end of the cylinder, leading to periodic oscillation of the wake as shown in Figure 7(b). The arrangement of the vortices behind cylinder at  $Re=100$  is called the Von-Karman Vortex Street. These vortices play an important role creating unstable flow behind the cylinder leading to uneven forces acting on the said cylinder. The separation of the flow for the Von-Karman vortices happens at acute angle (close to  $80^\circ$ ) from the front stagnation point of the cylinder (refer b).

Important factors that should be considered while setting up the problem in Ansys Fluent are the Courant number values and the total number of timesteps. Courant number is used in transient flow of computational fluid dynamics to evaluate the timestep required for stable results. The Courant number formula is:

$$C = \frac{U\Delta t}{\Delta x}$$

Where,  $U$  is the velocity of the flow,  $\Delta t$  is the time step for simulation and  $\Delta x$  is the characteristics height of the mesh cell. For an accurate representation of these vortices, ideally, the Courant number should be in the region of 0.4.

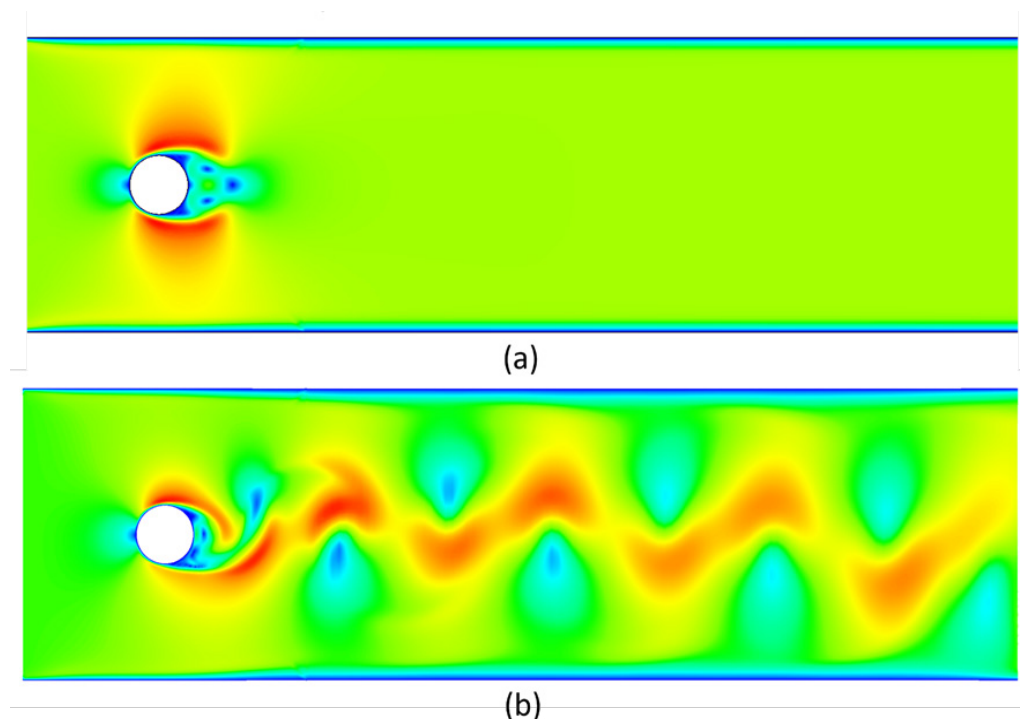


Figure 6 : The velocity contour (a)  $Re = 30$  (b)  $Re = 100$ .

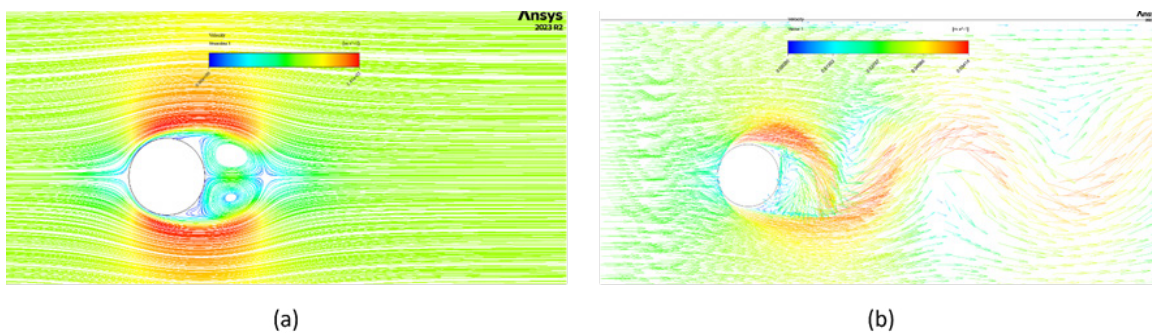


Figure 7: (a) Streamline for a flow around cylinder at  $Re = 30$  (b) Vector plots of flow around cylinder at  $Re = 100$ .

This vortex shedding will continue for the flow till it reaches the critical Reynolds number of  $(1 \times 10^5)$  above which the flow turns Turbulent, which is beyond the scope of the current case study. However, you can learn about it further at [Basics of Turbulent Flows using Ansys Fluent](#).

## 6. Conclusions

In this case study, we discussed the potential flow theory for a flow around a cylinder, and understood the equations that define it, as well as the assumptions. We ran simulations for the same scenario using Ansys Fluent and found that such a theorized solution cannot adequately capture the behavior of real flow. However, a close representation can be observed for a very low velocity inviscid flow around the cylinder. Furthermore, we observed that for most real-life flows, viscosity plays an important role and saw that for laminar flow, a Von- Karman vortex formation is occurring for a flow between Reynolds number 100 and above till the critical Reynolds number. This phenomenon has many practical effects and should be considered while designing real life structures to reduce instabilities.

## 7. References

Nakayama, Y. (1998). Introduction to Fluid Dynamics. Butterworth-Heinemann.



© 2024 ANSYS, Inc. All rights reserved.

## 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.

## Ansyes 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](http://www.ansys.com/education-resources).

## Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at [education@ansys.com](mailto:education@ansys.com).

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.