

# Study of Forced Convection over Heated 2D Cylindrical Body

Yeshas Sudharshan

## Abstract

The primary objective of this study is to observe the effect of exposing a 2-D heated cylindrical body to an airflow of 3 different Reynolds numbers (20, 100, 200). To visualise their respective velocity and temperature behaviours using contours and understand the physics behind the nature of air flow and its influence on the convection of a heated cylindrical body. The study is performed entirely in OpenFoam environment from using BlockMesh for creation of geometry and mesh to post-processing results using paraFoam software.

## 1 Introduction

The flow of air as a fluid around a heated cylindrical body has long been of interest to thermal and aeronautical engineers who are involved in designing re-entry vehicles, high speed aero-thermal effects and establishing chimneys, heat exchangers, HVAC systems and many more applications. While intuition provides a basic level of understanding, higher level of understanding of the behaviour of air with respect to velocity and temperature variations around the cylindrical body and after-effects observed downstream of the body, visualising the fluid behaviour with the help of CFD provides a deeper insight which will be helpful in designing and optimizing the product further beyond basic understandings.

The Reynolds numbers are selected for a general case of airflow over a 2-D cylindrical body in the low Reynolds number region involving laminar flow in Reynolds number 20 and 100 and in the case of Reynolds number 200 having just started the transition from laminar to turbulent flow and the case considered is not specific to any particular industrial application. Hence this study is an exercise to develop a better understanding of the physics behind similar real-world problems with the help of CFD results.

## 2 Problem Statement

This case study considers a 2-D cylindrical body of diameter 0.1 meters which is enclosed in a domain having dimensions as mentioned in table 1.

Boundary name	Dimensions in meters
Inlet	0.35
Outlet	0.35
Wall1	2.25
Wall2	2.25

Table 1: Table consists of boundary edge dimensions used in mesh

The inlet, wall 1, wall 2 are at a distance of 0.15 meters from the center of 2-D cylindrical body that is 1.5 times the diameter of the body whereas the outlet boundary is 20 times the diameter of the cylindrical 2-D body at 2 meters from the center of the cylindrical body as shown in figure 1.

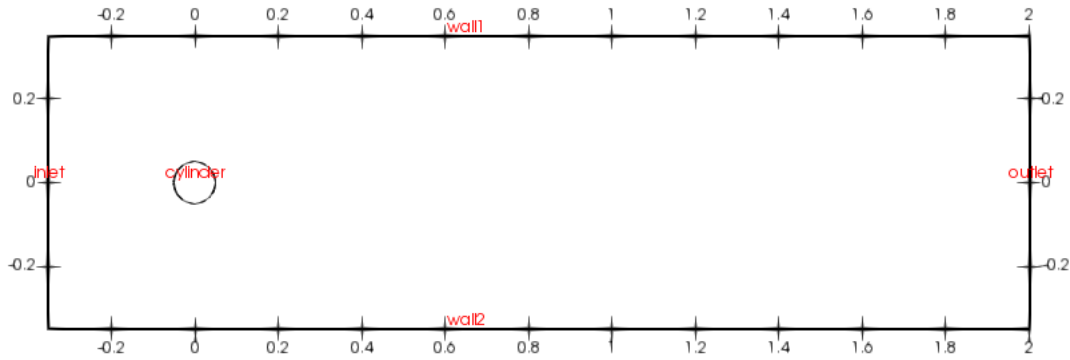


Figure 1: display of control volume encasing the circular cylinder geometry

### 3 Governing Equations and Models

For this case, thermal modelling is required in addition to the governing equations required for modelling fluid flow physics. Hence energy equation is necessary along with continuity equation and navier-stokes momentum equations. Here, only x and y directions of navier-stokes momentum equations are considered since the case study is regarding 2-D flow. All cases except for Reynolds number 200 can be considered as steady, whereas the fluid will obtain a transitional and unsteady flow at Reynolds number 200 the KEpsilon turbulence model has been chosen to help capture the turbulent behaviour of the flow properties more accurately. The governing equations for the case study are as follows.

- Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0. \quad (1)$$

- Navier-Stoke momentum equation X- direction

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

- Navier-Stoke momentum equation Y- direction

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

- energy equation for heat transfer

$$\rho c_p (\mathbf{U} \cdot \nabla T) = \nabla \cdot (k \nabla T) + \Phi. \quad (4)$$

- turbulent kinetic energy dissipation rate equation  $\epsilon$

$$\frac{D}{Dt} (\rho \epsilon) = \nabla \cdot (\rho D_\epsilon \nabla \epsilon) + \frac{C_1 \epsilon}{k} \left( P + C_3 \frac{2}{3} k \nabla \cdot \mathbf{u} \right) - C_2 \rho \frac{\epsilon^2}{k}. \quad (5)$$

- turbulent kinetic energy equation  $k$

$$\frac{D}{Dt} (\rho k) = \nabla \cdot (\rho D_k \nabla k) + P - \rho \epsilon. \quad (6)$$

## 4 Simulation Procedure

### 4.1 Geometry and Mesh

As per the geometry detailed above in problem statement. BlockMesh utility of openfoam which is one of the two built-in mesh generators of openfoam, blockMesh is chosen for it's ease in developing grids for simple geometries such as in our case of a 2-d cylindrical body.

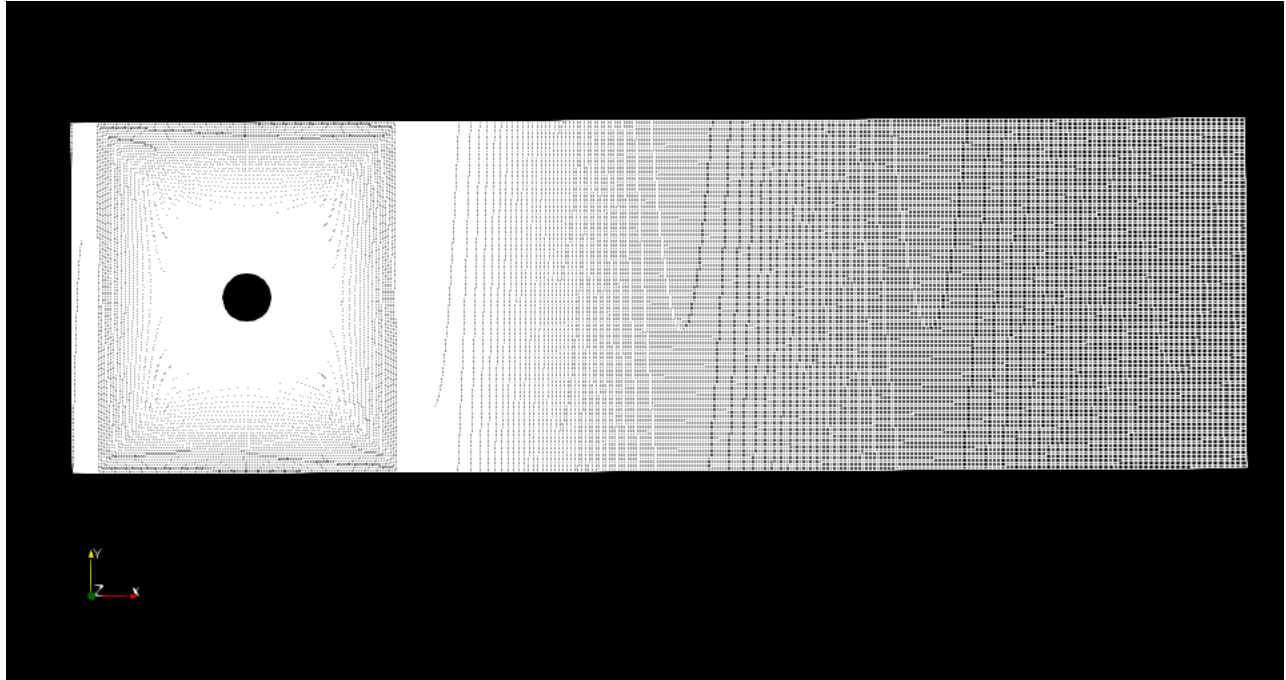


Figure 2: image of the mesh developed for the whole control volume

All dimension values and mesh parameters are detailed in the blockMeshDict file and running the command will generate a mesh as mentioned in blockMeshDict file and shown in fig2. In our case, the domain is such that the line connecting the diagonal ends have coordinates  $(-0.35 - 0.350)$  and  $(20.350.01)$ . There are a total of 6 blocks generated to increase the refinement of cells closer to the cylindrical body to better capture the changes in flow properties and capture the temperature changes close to the cylindrical body. To concentrate the cells close to the 2-D cylindrical body's boundary, a grading scheme called simpleGrading is used with a factor of 10 to achieve the above mentioned concentration towards boundary of cylindrical body.

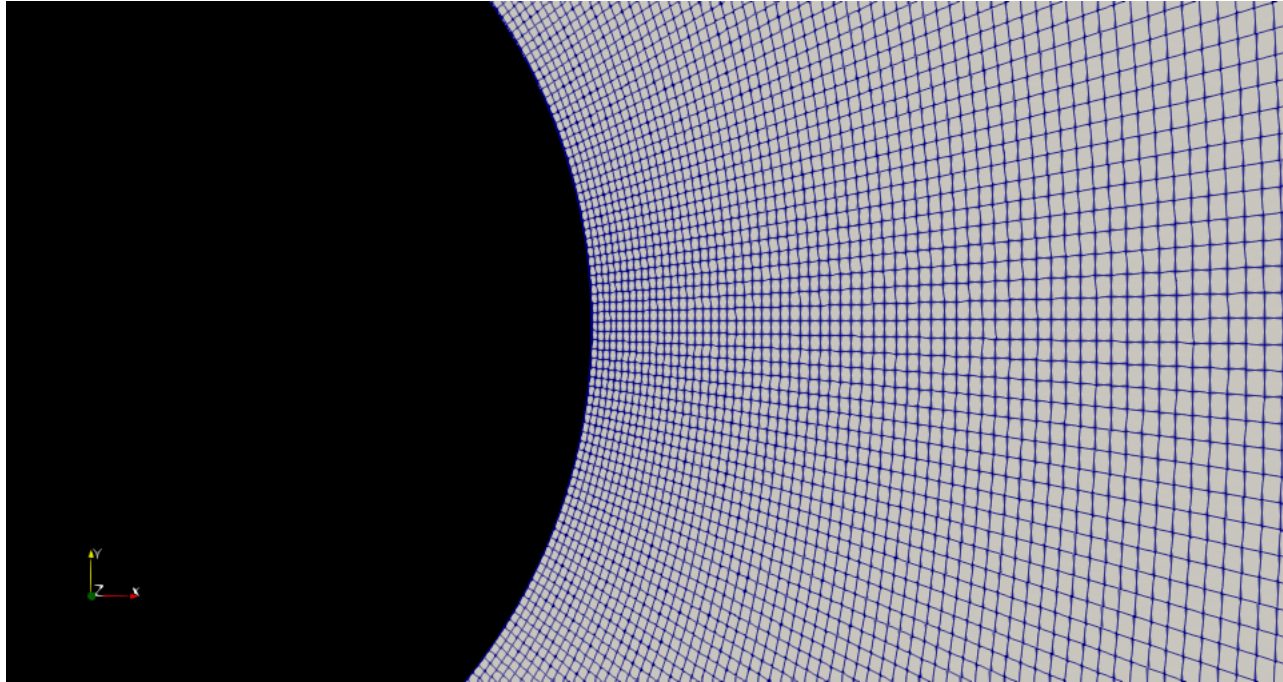


Figure 3: close-up display of mesh around cylindrical body

The domain consists of 82000 hexahedral cells. The maximum aspect ratio of the domain's cells is 3.65 and the maximum skewness of cells present in the domain is 0.88 as shown in fig 3.

## 4.2 Initial and Boundary Conditions

Since this case deals with temperature gradients using energy equation this case will have to deal with  $T$  and  $prgh$  i.e temperature and pressure-density variations and hence must initialise these parameters along with the pressure and velocity parameters which are typical for any fluid flow analysis. The initial conditions applied to the boundaries of the domain are detailed in table 2 and for the case of Reynolds number 200 we will be using a turbulence model and the necessary additional initial conditions of the turbulence model for the case is mentioned in table 3 .

Boundary	Initial Conditions			
	P	U	T	prgh
Inlet	calculated, internalField	fixedValue, uniform (0.0032 0 0)	zeroGradient	fixedFluxPressure, internalField
Outlet	fixedValue, Uniform 0	zeroGradient	zeroGradient	fixedValue, internalField
Wall1	calculated, internalField	slip	zeroGradient	fixedFluxPressure, internalField
Wall2	calculated, internalField	slip	zeroGradient	fixedFluxPressure, internalField
Cylinder	calculated, internalField	noSlip	fixedValue, uniform 320	fixedFluxPressure, internalField
frontAndBack	empty	empty	empty	empty

Table 2: Table consists of initial boundary conditions applied to the solver

Boundary	Initial Conditions			
	alpha-t	epsilon	k	nut
Inlet	zeroGradient	fixedValue, internalField	fixedValue, internalField	fixedValue, internalField
Outlet	zeroGradient	zeroGradient	zeroGradient	zeroGradient
Wall1	zeroGradient	zeroGradient	zeroGradient	zeroGradient
Wall2	zeroGradient	zeroGradient	zeroGradient	zeroGradient
Cylinder	compressible::alphiatWallFunction	epsilonWallFunction	kqRWallFunction	nutkWallFunction, internalField
frontAndBack	empty	empty	empty	empty

Table 3: Table consists of additional initial boundary conditions applicable only for Re.no 200

### 4.3 Solver

buoyantSimpleFoam is the solver selected to solve the case of laminar flow and was selected on the basis of its ability to simulate flow with temperature variations in the fluid, also a other necessary criteria for the selection of solver is the steady-state nature of equations used in buoyantSimpleFoam solver. Also buoyantPimpleFoam is the solver used in solving the case having Reynolds number 200, due to the flow developing unsteady flow in the wake region of the domain. The CFL(Courant-Fredrichs-lewy) number for the unsteady flow has been maintained to 0.5 which results in a  $\Delta t$  of 0.02

As mentioned in the previous section, buoyantSimpleFoam and buoyantPimpleFoam requires additional parameters to be initialised as they are required to solve the energy equation for modelling temperature variations in the fluid. Since we are operating the fluid in low Reynolds number region, the simulationType has been set to represent laminar flow in the entire fluid domain.

## 5 Results and Discussions

The results are categorised based on the individual Reynolds number under which the variations of velocity and temperature contours are represented and their behaviours are studied along with providing examples of any practical applications where an engineer might use this information to help appreciate the study being conducted.

### 5.1 Reynolds number 20

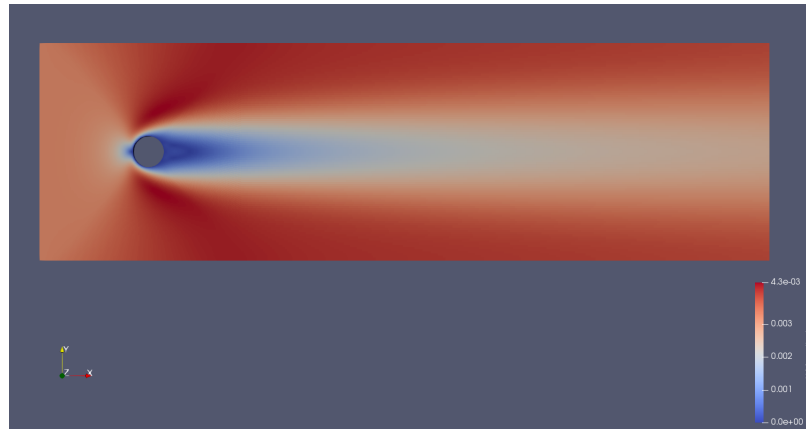


Figure 4: velocity contour of model at Re.no 20

The velocity contour as shown in fig 4 represents the variation in the magnitude of velocity in the fluid domain in the presence of a heated 2-D cylindrical object. From the figure we can observe that a stagnation region is formed at the leading edge of the cylinder with velocity of fluid increasing as it flows along the curvature of the cylinder walls as expected from theoretical understanding of the nature of flow around curved surfaces. As the flow travels further towards the trailing edge of the cylinder, there is a formation of wake region behind the cylinder and this wake region extends further downstream to create the pattern of velocity magnitude.



Figure 5: temperature contour of model at Re.no 20

The temperature contour as shown in fig 5 represents the variation of fluid domain in the presence of a heated 2-D cylinder and also shows us the behaviour of heat in a condition of forced convection where a fluid (air in this case) is flowing over the heated object. From the temperature contour, we can observe the heat dissipation occurring in the downstream of the flow and can study the influence of the downstream temperature due to the presence of the heated object upstream, this contour can provide a better preliminary understanding to situations of cooling towers and their influence on nearby structures downstream of local airflow directions and appreciate increase in cooling rate due to forced convection.

## 5.2 Reynolds number 100



Figure 6: velocity contour of model at Re.no 100

From fig 6 we can observe that the region of stagnation at the leading edge of the 2-D cylinder has reduced and this behaviour is expected to continue as the reynolds number increases since as the reynolds number increases fewer upstream fluid particles are to receive any kind of information about the presence of the cylindrical object in the fluid's path. There is an increase in the velocity of the fluid as it curves around the cylinder as reasoned in the previous section for flow in the Re no 20 domain. Whereas the region of wake behind the cylindrical body has reduced in width which again can be attributed to the increase in momentum of the fluid at higher reynolds number compared to previous case.





Figure 7: temperature contour of model at Re.no 100

Fig 7 shows the variation of temperature along the fluid domain with a flow of Reynolds number 100. As expected there is a significant reduction in the influence of the heated cylindrical body's heat dissipation in the downstream region of the fluid domain. The practical implication of this observation is that as the velocity of the fluid is increased, the temperature of the downstream region reduces as the fluid's temperature can now mix with the temperature of the cylinder downstream more effectively due to increased momentum of the fluid particle.

### 5.3 Reynolds number 200



Figure 8: velocity contour of model at Re.no 200

As we consider a higher Reynolds number case of 200, we can observe some interesting developments in the nature of fluids. From fig 8 we can observe that the transient nature of flow is prevalent in the downstream region of the domain. As the literature (W.A Khan et al., 2005) suggests that the flow over a 2-D cylindrical body will start to develop a transient unsteady flow behaviour at somewhere around Reynolds number of 150. We can see that the nature of fluid velocity downstream of the body is identical to the von Karman street pattern and is an onset of vortex

shedding downstream of the body. Further information on Von Karman street can be obtained from (Houghton.E.L et al., 2003)



Figure 9: temperature contour of model at Re.no 200

The temprature contour shown in fig 9 shows the unsteady transient nautre of fluid in the domain along with representation of the influence of te fluid's unsteady behaviour on the nature of temperature's propogation downstream of the source of heat(cylindrical body) and the way it intermixes with the ambient air at room temperature. This type of turbulent mixing helps reduce the effect of increase in temperature of the domain by effectively mixing the fluid which is heated by the cylindrical body with the ambient un-disturbed fluid. This behaviour can have applications in the industries where better mixing between two materials having different properties are required to mix well and form an homogenous byproduct.

## References

- Houghton.E.L, P.W.Carpenter, steven.H.Collicott, and daniel.T.valentine (2003). *Aerodynamics for Engineering Students*. Butterworth-Heinemann.
- Mohammed Salqlain (2023). Numerical analysis of flow past a blunt heated cylinder using open-foam. <https://cfd.fossee.in/case-study-project/case-study-run/238>. FOSSEE Case Study.
- W.A Khan, J.R.Culham, and M.M.Yovanovich (2005). Fluid flow around and heat transfer from an infinite circular cylinder. *Journal of Heat transfer*, 127(7):785–790.