Analysis of convective heat transfer over a square cylinder with rounded corners under steady flow regime at low Reynolds number

S Bose and S D Patle

Department of Mechanical engineering, National Institute of technology, Raipur, Chhattisgarh, 492010, India

Abstract. This work deals with the numerical simulation of square cylinder with different rounded corners and numerical analysis of convective heat transfer is studied, here emphasis is being laid on both type of flow regime i.e. unsteady flow regime and steady flow regime i.e. between 40 ≤ Re.≤ 160 and Pr is kept as 0.7 and thus the finding suggests that the heat transfer increases significantly with rounded corners in square cylinder.

Keywords – Nusselt number (Nu), Prandtl number (Pr), rounded corners, Reynolds number (Re)

1. Introduction

Heat transfer through bluff bodies encapsulates a broad area of research; this particular area is attracting the focus and attention of researchers for quite a long time. The heat transfer through bluff bodies over a defined geometry is generally considered as a problem of both theoretical and practical importance, it incorporates many practical applications such as heat transfer taking place through a heat ex-changer, it has got wide applications in food packaging industries i.e. ensuring uniform quality of food products where rate of heat transfer through food particles has to be maintained efficiently. The most commonly used geometries used for heat transfer are generally square and circular surface, study of flow over a circular cylinder is one of the classical problem in fluid mechanics which is associated with a rich variety of flow phenomenon, intrinsic interest and study of which provides the overall picture of the bluff body dynamics. These two bluff bodies are preferably used but the two objects differ in many aspects such as they have different wake regions and their temperature distribution varies [1]. Heat transfer through bluff bodies has many industrial applications such as design of heat ex-changer tubes, cooling towers, important aspects related to the design of these equipment involves enhancement of vortex shedding to ensure better mixing and increment in heat transfer rate or reducing the drag forces by weakening the wake region of the flow associated with it [2]. In a broader respect the flow across a square cylinder remains broadly speaking for an unconfined square cylinder, the flow remains attached to the cylinder up to about Re ≥1 and then it transits to the so-called (two dimensional)steady symmetric flow which is characterized by the formation of a wake in the rear of the obstacle. This flow pattern persists up to about Re = 40 and with the further increase in the value of the Reynolds number, the symmetry of the flow about the mid-plane is destroyed. Though the flow is still two dimensional but it is no longer steady [1]. This work is generally focused on the heat transfer taking place in intermediate geometries and to
create highly substantial wake regions which will lead to significant rise in heat transfer by reducing lift and drag forces to a major extent so as to ensure high stability in the test structure when the flow takes place. Convective heat transfer is a topic of heat transfer which consists of extensive use of non-dimensional numbers. This non-dimensional numbers are of major use in order to define certain parameters which are associated with the flow characteristics as well as heat transfer. By the previous knowledge of heat transfer we can very well say that the heat transfer taking place due to forced convection mainly depends on two non-dimensional number which are Reynolds number (Re) and Prandtl number (Pr). Reynolds number mainly used to determine the type of flow taking, in this work we are mainly concerned with laminar flow whereas the Prandtl number deals with the relationship between the thermal and hydrodynamic boundary layer, thus also depicts about the magnitude of viscosity of fluid so used i.e. more viscous the fluid, higher is Prandtl number. Another important non-dimensional number which mainly deals with the amount of heat transfer taking place due to forced convection is Nusselt number (Nu). Thus, it is quite obvious that with increase in Nusselt number, convective heat transfer also increases. In context of the present work Nusselt number is going to be determined by performing simulation of a test section involving fluid flow over a cylindrical surface having intermediate geometry over a 2D computational domain in due course the temperature profile of the whole computational domain will be studied in order to study the amount of heat transfer taking place in every part of the test section aided with the variation of local Nusselt number taking place in the whole computational domain also depicting the variation in heat transfer taking place in the test section. The local Nusselt number variation with the distance will also be used to calculate the average Nusselt number which is going to be used for further as per requirements. Similarly, the nature of drag and lift coefficient also studied with course of the iterations carried on, for the stability of the test section this two variables are to be minimized to a major extent. Thus, a 2D simulation is being carried out numerically by using governing equations i.e. continuity, momentum, energy in analytic software which is FLUENT for this particular project. Before putting the model for analysis, it has to be designed with proper dimensions and precision and more importantly it has to be meshed properly so that the errors occurring gets nullified or can be minimized as far as possible in order to attain maximum accuracy. In this particular work all, the tasks related to modelling are being performed by using a modelling software GAMBIT.

2. Objective
Enhancement of heat transfer plays a major role in today’s industrial world. Heat transfer enhancement is very useful for increasing efficiency of several devices in many cases such as heat exchanger, condenser, evaporator etc. where heat transfer plays crucial role. Heat transfer also becomes handy in food industries where uniform quality of food product is required for achieving that a particular temperature difference is required to be maintained. The main aim of this project is to enhance heat transfer i.e. to maximize heat transfer by reducing drag and lift forces which are responsible for creating instability in the structure. The technique used in this project is rounding the corners of square cylinder, since previous literature only focuses on regular geometries such as square and circular cylinder. Here in this particular work numerical simulations are being carried out to investigate the heat transfer characteristics of a square cylinder having fillet radius \( R_f \), so based on different fillet radii various geometries are being used for further numerical analysis. Emphasis is being laid upon several dimensionless numbers such as Prandtl number (Pr) and Reynolds number (Re). But the main attention is being paid on Nusselt number (Nu), because here the mode of heat transfer is forced convection and from the knowledge of heat transfer we can say that convection becomes more effective with increase in Nusselt number (Nu). Previous literature of rounded geometry shows that the drag and lift parameters decreases significantly with the use of rounded corners. So in this particular work efforts are being made to study its effect on heat transfer characteristics with the use of different geometries having different fillet radius and the Nusselt number (Nu) is being kept under consideration, also the effect of vortex shedding is being studied because different wake regions will be created under different
geometries, substantial wake regions facilitates heat transfer by ensuring proper mixing among the surface of cylinders and fluid used.

3. Problem description
The problem consists of a computational domain having following specifications, \( L_U = 9D \), Upstream length of the computational domain \( L_D = 17D \), Downstream length of the computational domain, \( H = 20D \), Length of the cylinder \( D = \) Diameter of the square cylinder having fillet radius \( U_\infty = \) Free velocity of the fluid, m/sec, \( T_\infty = \) Free stream temperature of the fluid, \( K \), \( T_w = \) Temperature of the cylinder, \( K \),

![Figure 1. Computational domain](image)

4. Governing equations

4.1. Continuity equation for 2D compressible flow:

\[ \frac{\partial \rho}{\partial t} + \rho \nabla \vec{V} = \frac{\partial \rho}{\partial t} + \nabla (\rho \vec{V}) \]  

Where for 2D compressible flow,

For incompressible flow \( \rho = \) constant and (1) simplifies to (2)

\[ \nabla \cdot \vec{V} = 0 \]  

Any vector field that satisfies the equivalent condition given by (2) is called the divergence free or the solenoidal field.

Where, \( \vec{V} = u \hat{i} + v \hat{j} \) is the velocity vector and \( u \) and \( v \) are the Cartesian components of velocity vector in \( x \) and \( y \) directions.

4.2 Momentum equations

4.2.1 X-Momentum equations

\[ \rho \frac{Du}{Dt} = \left( - \frac{\partial P}{\partial x} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} \right) + \rho f_x \]  

4.2.2 Y-Momentum equations

\[ \rho \frac{Dv}{Dt} = \left( - \frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} \right) + \rho f_y \]  

Now so equation (3) can be written in the following conservative form:
\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \vec{V}) = \left( -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} \right) + \rho f_x \ldots \ldots \ldots (5)
\]

Where \( \tau \) signifies shear stress, \( f_x \) and \( f_y \) are acceleration in x and y direction respectively.

### 4.2.3 Energy equations

\[
\frac{d\rho e}{dt} = \frac{\partial \rho e}{\partial t} - e \left( \frac{\partial p}{\partial t} + \nabla \cdot \vec{V} \right) + \nabla \cdot (\rho e \vec{V}) \ldots \ldots \ldots \ldots (6)
\]

Where \( \nabla \cdot (\rho e \vec{V}) = \rho \nabla \cdot \vec{V} + \rho e \vec{V} \cdot \nabla e \).

Where \( e \) is the sum of the internal energy per unit mass.

### 5. Boundary conditions

The aforementioned equations along with the above noted boundary conditions yields the velocity, pressure and temperature fields and these, in turn are processed further to deduce the global characteristics like drag coefficients and Nusselt number. In this work SIMPLE scheme is being used aided with second order upwind scheme for further iterations, time step size is assigned as 0.1- and 1000-time steps are being processed accompanying 20 iterations per time step. In addition, several boundary conditions are being imposed such as the temperature of the square cylinder is taken as \( T_w = 300.05 \) °K, so heat transfer takes place and generally in this particular case the condition of constant wall temperature is being used and heat flux is considered to be zero in this for further study. Now with the use of this boundary conditions many graphs are being plotted by the values obtained from the iterations and thus the validation with the published literature is being done in order to validate the procedure. Now with the help of this very procedure, several other results are being calculated in order to expand the domain of the work.

#### 5.1. Inlet boundary

The front half of the surrounding envelope of fluid is designated as the inlet and on this surface, the uniform velocity in x-direction with constant fluid temperature is specified i.e. \( u = 1, v = 0, \theta = 0 \)

#### 5.1.1 Surface of cylinders

Since these are solid boundaries which are maintained at a constant temperature \( T_w \), the usual no-slip boundary condition is used on these surfaces, i.e.,

#### 5.2 Outlet boundary:

\( u = 0, \quad v = 0, \quad \theta = 1 \)

The rear half of the surrounding fluid envelope is designated as the outlet. Since the value of \( D \) is expected to be sufficiently large, the flow is expected to be fully developed in the axial direction, albeit the gradients might still exist in the lateral direction. This is the so-called default outflow boundary condition of ANSYS FLUENT. It is also equivalent to the usual Neumann type boundary condition given as follows, \( \frac{\partial u}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0, \quad \frac{\partial \theta}{\partial x} = 0 \). The numerical solution of the governing differential equations subject to the aforementioned boundary conditions maps the flow domain in terms of the pressure, velocity and temperature fields. The temperature field, in turn, can be used to deduce the value of the local Nusselt number on the surface of the cylinder.

### 6. Mesh convergence study

The grid structure used in the present work is shown in Fig.2. The grid is divided into five separate zones in both directions, and uniform as well as non-uniform grid distributions are employed. The grid
distribution was made uniform with a constant cell size, \( D = 0.25B \), outside a region around the cylinder that extended four units upstream, downstream and sideways. A grid of much smaller size \( d \), is clustered around the cylinder over a distance of 1.5 units to adequately capture wake–wall interactions in both directions. The hyperbolic tangent function has been used for stretching the cell sizes between these limits of \( d \) and \( D \). In this present numerical study, three different mesh sizes (Grid 1-117506, Grid 2-91866 and Grid 3-88955, Grid 4-84810) are adopted for \( \lambda' = 0.2 \) and \( \Theta = 5 \), in order to check the mesh independency. A detailed grid independency study has been performed and results are obtained for the Lift coefficient (\( C_L \)) & drag coefficient (\( C_D \)) but there are no considerable changes between Grid2 and Grid3. The results are shown in Table 1. Thus, a grid size 40000 is found to meet the requirements of the both grid independency and computation time limit. Eswaran and Prakash [22] and of Sharma and Eswaran [23] initially developed for complex 3D geometries on a non-staggered grid has been used here in its simplified form for 2D flows. In brief, the semi-explicit method is used to solve the unsteady Navier–Stokes equations in which the momentum equations are discretized in an explicit manner. In order to obtain reliable and accurate results, it is important to choose carefully the length and width of the computational domain and grid size. A thorough grid independence study for the problem under consideration has been done by Sharma and Eswaran [9] at \( Re = 160 \) and \( Pr = 0.7 \) and Dhiman et al. [11] at \( Re = 45 \) and \( Pr = 50 \). Based on these studies, in the present study, for the highest Reynolds number of 160 and Prandtl number of 20, five non-uniform grids (223x179, 246x200, 295x241, 325x271 and 365x303) were used to demonstrate the grid independence of the results, as shown in Fig. 3. The percentage change in the values of the pressure and total drag coefficients and the average Nusselt number for the coarsest grid (MxN = 223x179) and the finest grid (MxN = 365x303) are 3.3%, 3%, 2% (constant temperature case) and 2.5% (constant heat flux case), respectively.

![Figure 2. Non-uniform computational study 325x271 grid points by Sahu a.k et al [1]](image)

| NDES    | CELLS   | VALUE OF C_D | VALUE OF C_L | % CHANGE  | % CHANGE  |
|---------|---------|--------------|--------------|-----------|-----------|
| 117506  | 116776  | 1.6109       | 0.3250       |           |           |
| 91866   | 91210   | 1.5328       | 0.2867       | 4.848     | 3.83      |
| 88955   | 88321   | 1.5298       | 0.2750       | 0.19      | 1.17      |
| 84810   | 84186   | 1.5108       | 0.3000       | 1.9       | 8.33      |

Table 1 showing the results of grid dependency study carried on the computational domain. From the above table showing grid in dependency it is very clear that the grid having 88955 nodes is showing the least fluctuation in the values of \( C_D \) and \( C_L \) so it would be in the best of interest to choose this...
very grid in order to minimize the computation time without making any compromise with the accuracy of the results. So, the new computational domain will also be very close to the above mentioned non uniform grid suggested by Sahu A.k et al [1].

![Figure 3. Non uniform grid containing 88955 nodes.](image)

7. Results and Discussions

Numerical simulations are being carried out rigorously in order to simulate the heat transfer characteristics of a square cylinder having rounded ducts by considering different geometries at different fillet radii, enhancing heat transfer and thereby reducing drag forces thus the results illustrate the effect of corner radius on square cylinder. In this particular section all the plots and contours will be discussed related to the project and those will be demonstrated and finally reaching to the concluding and thereby discussing the results in details.

**Table 2. Data for steady flow regime over a square cylinder having fillet radius at Re =40**

| S.NO | NEW DIAMETER (R) | Fillet radius (RF) | COEFFICIENT OF DRAG (Cd) | COEFFICIENT OF LIFT (Cl) | NUSSELT NO (Nu) | GEOMETRY | Re |
|------|------------------|--------------------|--------------------------|--------------------------|----------------|----------|----|
| 1    | R = 0.7 D        | -                  | 2.9932399                | 0                        | 3.1637144      | SQUARE   | 40 |
| 2    | R = 0.5D        | 0.5                | 1.7426306                | 0                        | 3.3790436      | CIRCLE   | 40 |
| 3    | R = 0.68D       | 0.0554             | 2.9843021                | 0                        | 3.2597222      |          | 40 |
| 4    | R = 0.66D       | 0.0978             | 2.9471523                | 0                        | 3.3146365      |          | 40 |
| 5    | R = 0.64D       | 0.1422             | 2.8863332                | 0                        | 3.3587868      |          | 40 |
| 6    | R = 0.62D       | 0.1886             | 2.8329057                | 0                        | 3.4091001      |          | 40 |
| 7    | R= 0.6 D        | 0.2380             | 0                        |                          |                |          | 40 |

Results above in Table 2 illustrates the details of the numerical simulation which is being carried out rigorously and here it is apparent that the Nusselt number is increasing significantly with increasing the fillet radii which is as mentioned earlier one of the popular techniques to reduce the hydrodynamic forces such as drag and lift. Thus, increase in Nusselt number hereby concludes the heat transfer enhancement since the Nusselt number is on the higher side with subsequent reduction of fluid forces.
Figure 4. Variation of Nusselt number in a steady flow regime

The above Figure 4 also confirms the increase in Nusselt number with decrease in new radius R up to circular cylinder. Thus, with increase in Nusselt number the convective heat transfer is on a high, shows the effect of rounded corner geometry.

8. Conclusion

From the above results it is pretty clear that as the magnitude of fillet radius is increased to a certain extent the Nusselt number increases which is particularly the aim of the work with a subsequent decrease in drag and lift forces and the recirculation period also decrease with increase in fillet radius thereby ensuring proper mixing and enhancement of heat transfer.

References

[1] C.A. Lareo, P.J. Fryer J. Food Eng. 36(1998)417–443.
[2] K.P.Sandeep,C.A.Zuritz J. FoodEng. 25 (1) (1995)31–44.
[3] K.P. Sandeep, C.A. Zuritz, V.M. Puri, Int. J. Food Sci. Technol. 35 (2000)511–522.
[4] M.M.Zdravkovich:Fundamentals,vol.1,OxfordUniversity Press, New York, 1997.
[5] M.M.Zdravkovich,Applications,vol.2,OxfordUniversity Press, New York, 2003.
[6] L. Baranyi, J.Comput. Appl. Mech. 4 (1) (2003)
[7] R.P. Bharti, R.P. Chhabra, V. Eswaran Heat Mass Transfer 41 (2005)824–833.
[8] G.Juncu,Int. J. Heat Mass Transfer 50 (2007) 3799–3808.
[9] A. Sharma, V. Eswaran, Numer. Heat Transfer A 45 (3) (2004)247–269.
[10] A.K.Dhiman,R.P.Chhabra,V.Eswaran,Int.J.HeatMassTransfer48 (2005)4598–4614.
[11] A.K.Dhiman,,R.P.Chhabra,A.Sharma,V.Eswaran ,Numer.HeatTransferA 49 (2006)717–731.
[12] K.M.Kelkar,S.V.Patankar, Int. J. Numer. Methods Fluids 14 (1992)327–341.
[13] S. Turki, H. Abbassi, S.B. NasrallahInt. J. Therm.Sci. 42 (2003)1105–1113.
[14] M.Rahnama,H.Hadi-MoghadamHeatTransferEng.26(10)(2005)21–29.
[15] S. Bhattacharyya, S. MahapatraHeat Mass Transfer 41 (2005)824–833.
[16] T.H. Ji, S.Y. Kim, J.M. Hyun, Int. J. Heat Mass Transfer51 (2008)1130–1138.
[17] B. Paliwal, A. Sharma, R.P. Chhabra, V. Eswaran Chem. Eng. Sci.58 (2003) 5315–5329.
[18] A.K.Gupta,,A.Sharma,,R.P.Chhabra,V.Eswaran, Ind. Eng. Chem. Res. 42 (2003) 5674–5686.
[19] S.Whitaker AIChEJ.18 (1972)361–371.
[20] S. Sanitjai, R.J. Goldstein Int. J. Heat Mass Transfer 47(2004)4795–4805.
[21] J. C. Hu, Y. Zhou, C. Dalton, 2006 Experiments in Fluids 40: 106–118
[22] A.C.MandalandG.M.G.Faruk2010 Th einstitution o fengineers, Bangladesh, vol. me 41, no.1,
[23] Justin S. Leontini, Mark C. Thompson 2013 *Journal of Fluids and Structures* 39 371–390

[24] Tetsuro Tamura, Tetsuya Miyagi 1999, “The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes”, *Journal of Wind Engineering and Industrial Aerodynamics* 83 135–145

[25] Luigi Carassale, Andrea Freda, Michela Marrè-Brunenghi 2013 *J. Wind Eng. Ind. Aerodyn.* 123 274–280.

[26] Luigi Carassale, Andrea Freda, Michela Marrè-Brunenghi 2014 *Journal of Fluids and Structures* 44 195–204.