Numerical Investigation of Effect of Buoyancy on the Wake Instability of a Heated Cylinder In Contra Flow

Khyati B. Varma\(^1\) and Hui Hu\(^2\) and Z J Wang\(^3\)

\(\text{Iowa State University, Ames, Iowa, 50011}\)

Flow over a cylinder is a classically and practically interesting problem. A number of studies focus on the vortex shedding and wake instability of an unheated cylinder. Inspite of its applications flow over a heated cylinder operating in a mixed convection region has received very little attention. The present study aims at the numerical investigation of unsteady heat transfer for a laminar flow past a circular cylinder with small aspect ratio (\(L/d=7\)) or finite spanwise length with no-slip wall conditions. A circular cylinder is exposed to approaching flow stream in the direction of gravity or opposing the direction of buoyant force (contra flow). The Reynolds number and the temperature of the incoming flow are set as \(\text{Re}=130\) and \(297\) K respectively. The temperature of the heated cylinder is varied between \(297\) K to \(358\) K corresponding to Richardson number varying between \(\text{Ri}=0.0\) and \(\text{Ri}=1.05\). The analysis performed focuses on the effect of buoyancy on the wake instability in terms of vortex shedding frequency, drag forces, wake centerline velocities, averaged Nusselt number and transition phenomenon. In addition to this, preliminary study is conducted to understand the influence of small aspect ratio and end effects on the transition mechanism due to heating. Numerical results are compared to available experimental data and some new results are presented.

\textbf{Nomenclature}

\[\begin{array}{ll}
D & = \text{diameter of the cylinder} \\
L & = \text{length of the cylinder} \\
x, y, z & = \text{cartesian co-ordinates} \\
V_x & = \text{Uniform inlet streamwise velocity} \\
u & = \text{crossflow velocity component} \\
v & = \text{streamwise velocity component} \\
w & = \text{spanwise velocity component} \\
V_c & = \text{mean velocity in streamwise direction} \\
\overline{D} & = \text{time-averaged drag} \\
\overline{p} & = \text{mean pressure} \\
\overline{C_d} & = \text{time-averaged drag co-efficient} \\
f & = \text{frequency of vortex shedding} \\
Gr & = \text{Grashof number} \\
Ri & = \text{Richardson number} \\
Re & = \text{Reynolds number} \\
St & = \text{Strouhal number} \\
T_w & = \text{cylinder wall temperature} \\
T_{\infty} & = \text{temperature of water}
\end{array}\]

\(^1\) Graduate Student, Department of Aerospace Engineering, and AIAA Student Member, email: khyati@iastate.edu.
\(^2\) Assistant Professor, Department of Aerospace Engineering, and AIAA Senior Member.
\(^3\) Associate Professor, Department of Aerospace Engineering, and AIAA Associate Fellow.
I. Introduction

FOR over a century, the wake flow behind a cylinder has captivated the attention of scientists and engineers. Flow over a cylinder results in the vortex shedding phenomenon over a wide range of Reynolds Number ($Re = \rho V d / \mu$). A number of studies focus on the wake stability and vortex shedding characteristics for an unheated cylinder for low Reynolds Number. In the last decade, researchers have worked in the field of 3D wake dynamics behind an unheated cylinder with advances in experimental and numerical studies.

After a thorough review of the literature on wake flow behind unheated and heated cylinder, the present study aims at the numerical investigation of flow over a heated cylinder in a contra (vertically downward) flow arrangement and the effect of buoyancy on the wake behind it. The heat induced buoyancy effects are expressed by Grashof number ($Gr = g \beta (T_w - T_a) d^3 / \nu^2$) and the relative importance of the forced and buoyant effects is indicated by Richardson number ($Ri = Re / Gr^2$). With the increasing $Ri$ the heat transfer process around the heated cylinder changes from forced convection to the mixed convection regime.

Williamson [1] concluded from his experiments that for an unheated cylinder, the flow becomes 3D when the $Re$ is greater than 180. However the 3D behavior can be observed for even a lower $Re$ if the aspect ratio of the cylinder $L/D$ is low with the cylinder of a finite length. The results presented by Williamson [1] vary for the aspect ratio less than 28 and larger aspect ratios. Similar behavior is observed by Mittal [2] and the end conditions determine the mode of vortex shedding for a finite cylinder. His studies indicate that the wake transition regime that is known to exist on the $Re$ range of 190-250 for large aspect-ratio cylinders is either extended and/or delayed for a cylinder of small aspect-ratio with “no-slip” walls. In the current study, the aspect-ratio $AR = L/d = 7$, is used to faithfully model the experimental conditions. For such an aspect ratio and $Re$, the wake transition regime is extended which is presented here. The results for 3D transition are discussed for $Re=130$ and $Ri=0.0$ to 1.05 and results are analyzed to see how heating would affect the transition mechanism. Vortex shedding becomes very complicated, when the buoyant forces take over the viscous phenomenon. This classical and practically interesting phenomenon has its implications in other applications such as heat exchanger tubes, nuclear reactor fuel rods, chimney stack, cooling towers, offshore structures, electronics cooling etc. A number of engineering design parameters connected with fluid flow and heat transfer become important in these studies. Apart from engineering relevance, a study of these mechanisms associated with the laminar flow past cylinders, forms the first step towards the understanding of vastly more complicated phenomenon of turbulence.

Researchers investigated the buoyancy effects for the first time in early seventies to study the influence of the induced heat on the heat transfer coefficient (Hatton [3]). This study gave remarkable results for critical $Ri$ being more than 0.2 due to the influence of thermal effects for various angles between the gravity vector and the streamwise direction. Oosthuizen et al. [4] have studied the influence of mixed convection over a Reynolds number range of 100-300 with their interest in calculating the average heat transfer. Jain et al. [5] have pointed out an increase in the vortex shedding frequency with increasing cylinder temperature. Furthermore, investigations for thermal effects on the wake by Noto et al. [6] and Badr [7] described the flow over a horizontal heated cylinder opposing the direction of gravity. The results by Noto et al. [6] indicated the effect of increasing cylinder temperature on the Strouhal Number. According to them, the Strouhal number decreases to almost zero after a critical Richardson number. The same situation was numerically analysed by Chang and Sa [8] with detailed study on the vortex mechanisms in the near wake of a heated/cooled circular cylinder in a parallel flow (vertically upwards). With the recent development in this area, Kieft and Steehoven [9] have studied the effect of buoyancy
on the heated cylinder in a horizontal cross flow. From their studies they found that the non-parallelism between the crossflow and buoyant forces cause the flow pattern to become asymmetric.

Most recently, Hu and Koochesfahani [10] have done an experimental investigation of the effect of buoyancy on the wake instability of a heated cylinder in a contra flow. So far, there is no published literature on the numerical investigation of the same problem. In the present study, special attention is given to the analysis of buoyancy effects on the wake instability of the heated cylinder in terms of vortex shedding frequency, the wake closure length, drag coefficient and the local and averaged Nusselt number of the heated cylinder. We know that non-intrusive flow visualisation experimental techniques like Laser-Induced Fluorescence (LIF), Particle Image Velocimetry (PIV) etc. turn out to be very expensive. On the other hand CFD simulation of such a problem can give us a preliminary understanding of the flow physics at a much lower cost.

The main objective of the present study is the numerical simulation of unsteady-state heat transfer for a laminar flow past a circular cylinder. The analysis performed focuses both, on the 3D flow transition and the wake instability due to buoyancy effects. A circular cylinder is exposed to approaching flow stream in the direction of gravity or opposing the direction of buoyant flow. The Reynolds number and the temperature of the incoming flow are set as \( \text{Re} = 130 \) and \( 24^\circ C \) respectively. The temperature of the heated cylinder is varied between \( 24^\circ C \) to \( 85^\circ C \) corresponding to Richardson number varying between \( R_i = 0 \) and \( R_i = 1.0 \). When the temperatures of the cylinder and flow stream are the same, an isothermal wake develops behind the cylinder. However, when the cylinder temperature is higher than the flow stream temperature, a wake with buoyant force opposing the flow stream direction occurs. In addition to this, special attention is given to the influence of small aspect ratio (AR) and end effects on the transition mechanism due to heating.

II. Methodology

A. Problem Definition

For a fixed value of Reynolds number \( \text{Re} = 130 \), at varying Richardson number \( R_i = 0.0 \) to \( 1.0 \), both unsteady two-dimensional and three-dimensional flow simulations for water have been implemented trying to maintain similar characteristics of mesh generation and solver set-up to make the results comparable. The flow is assumed to be in the vertically downward direction approaching the isothermal heated copper cylinder as shown in the schematic of the problem in Fig. 1. The six cases for various temperature conditions of the heated cylinder that are studied in the present work are listed in the Table 1.

B. Computational modeling

1. Boundary Conditions

The boundary conditions for the computational domain can be specified as follows. At the inlet, uniform velocity and temperature boundary conditions were imposed. The inlet freestream velocity \( V_\infty = 0.0255 \text{ m/sec} \) as calculated for \( \text{Re} = 130 \) considering the thermophysical properties of water at over cylinder temperature \( T_\infty = 297 \text{ K} \) or \( 24^\circ C \) based on the definition of Re.

\[
\begin{align*}
\text{u}=0, \text{ v}=V_\infty, \text{ T}_\infty = 297 \text{ K}
\end{align*}
\]
A pressure outlet condition is specified at the outlet. With this type of boundary condition, the input pressure is used as the static pressure of the fluid at the outlet plane and all other conditions are extrapolated from the interior of the domain.

For the lateral boundaries, the no-slip condition is applied with adiabatic thermal wall boundary condition.

\[ u = v = w = 0 \text{ and } T = T_{\infty} \]  

The no-slip boundary condition was applied at the isothermal test cylinder where constant wall temperature condition is modeled. Based on \( Ri = 0.0 \) to \( 1.0 \), the temperature \( T_w \) of the copper cylinder ranges from \( 297 K - 358 K \) (\( 24^\circ C - 85^\circ C \)).

\[ u = v = w = 0 \text{ and } T = T_w \]  

2. **Properties of water and copper**

The thermophysical properties of water vary with the temperature. The thermally induced variation of the material properties of water cannot always be safely neglected over this range \( 292 K - 360 K \). For instance, the viscosity assumes almost \( 67\% \) variation from \( 989 \times 10^{-6} N s / m^2 \) to \( 324 \times 10^{-6} \) in the temperature range of \( 292 K - 360 K \). Hence polynomial functional relations are used to account for the temperature dependence. The following equations are used to determine variation with the temperature in density, specific heat, thermal conductivity and dynamic viscosity of water,(Incropera and DeWitt [13]). These formulae are implemented in the numerical computation with the user defined functions (UDFs).

\[ \rho = \frac{a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4 + a_5 T^5}{1 + a_6 T} \]  

\[ C_p = b_0 + b_1 T + b_2 T^2 + b_3 T^3 + b_4 T^4 \]  

\[ k = c_0 + c_1 T + c_2 T^2 + c_3 T^3 + c_4 T^4 \]  

\[ \mu = d_0 + d_1 T + d_2 T^2 + d_3 T^3 + d_4 T^4 \]  

The coefficients of temperature dependence for density, specific heat, thermal conductivity and viscosity are in Table 2. corresponding to equations [4-7] respectively.
C. Numerical Method

In the current study, numerical investigation has been conducted using a time-accurate finite-volume method solving the unsteady Navier-Stokes equations. The commercial finite volume computational fluid dynamics code, FLUENT 6.0 is used for the numerical simulation. Spatial derivatives were discretized using QUICK scheme which is based on averaging the second order upwind and central differences of a variable. The second-order implicit method was used for temporal discretization. The segregated solver was applied to solve the momentum and the energy equations and considering the difference of thermophysical properties between the fluid region and the cylinder, the double-precision solver is used. At each step, the momentum equations are first solved to obtain the velocity field. The pressure field and velocity field were coupled by PISO (Pressure Implicit with Splitting Operator) method. Upon solving the continuity and momentum equations, the energy equation was solved to achieve a quasi-steady periodic condition. The lift force was monitored over the cylinder surface as a function of time. The steady-periodic state was assumed to be achieved when the variation in the oscillating amplitudes of the $C_d$ was less than 1%. The time step size can be calculated based on the vortex shedding frequency for the unheated cylinder in terms of the Strouhal Number. Based on the previous experimental and numerical results, we know that the Strouhal Number for flow past an unheated cylinder for $Re=150$ is approximately 0.18. In order to capture the shedding correctly, we need at least 20 to 25 time steps in one shedding cycle. A thorough time refinement study is performed based on the above calculations and used consistently for all the simulations.

1. Mesh Generation

As discussed earlier, the aspect-ratio of the cylinder used for the present study is low which results in end effects due to no-slip boundary condition. This leads to a 3D transition in the wake for the present $Re=130$ even for the unheated case. Hence we cannot use a two-dimensional grid for our numerical simulation. The two-dimensional simulations were performed mainly to verify the consistency of the choices related to the computational domain dimensions and to its discretization. The computational domain is meshed using CFD-GEOM into a quadratic mesh. To obtain a structured mesh, domain decomposition in sub-volumes is needed and has been implemented as shown in the Fig 2.

Accuracy of CFD solutions strongly depends on the grid system which must be constructed to minimize the grid-induced errors and to resolve the flow physics. For this purpose a final grid system (Fig. 3) is employed after generating solutions on three different grid systems illustrated in the Table 3. The result of the assessment of this grid independence study is represented in Table 3, for the predicted values of the vortex shedding frequency of Strouhal Number. The correct reproduction of the St for the unheated cylinder is achieved indicating grid independence until the variations in the St were less than 2.0% which compares fairly well with the experimental results. Table 3, demonstrates that the fine grid yields the most grid independent result based on the criteria for St. Every solution present in this work is made grid independent in the manner described above.

### Table 2. Coefficients in temperature dependence formulae

<table>
<thead>
<tr>
<th>$n$</th>
<th>$a_n$</th>
<th>$b_n$</th>
<th>$c_n$</th>
<th>$d_n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1819</td>
<td>1.6987</td>
<td>1.8231</td>
<td>46763</td>
</tr>
<tr>
<td>1</td>
<td>$-12.105$</td>
<td>$-0.024932$</td>
<td>$-0.020442$</td>
<td>$-510.93$</td>
</tr>
<tr>
<td>2</td>
<td>$7.04 \times 10^{-2}$</td>
<td>$1.469 \times 10^{-4}$</td>
<td>$1.0365 \times 10^{-4}$</td>
<td>$2.2991$</td>
</tr>
<tr>
<td>3</td>
<td>$-1.9675 \times 10^{-4}$</td>
<td>$-4.3376 \times 10^{-7}$</td>
<td>$-2.0506 \times 10^{-7}$</td>
<td>$-4.602 \times 10^{-3}$</td>
</tr>
<tr>
<td>4</td>
<td>$2.601 \times 10^{-7}$</td>
<td>$6.4132 \times 10^{-10}$</td>
<td>$1.396 \times 10^{-10}$</td>
<td>$3.4604 \times 10^{-6}$</td>
</tr>
<tr>
<td>5</td>
<td>$-1.3494 \times 10^{-10}$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Figure 2. Domain Decomposition](image-url)
As mentioned earlier, two-dimensional numerical results remain far from the physical phenomenon when the cylinder has a finite length or the aspect-ratio is low. This conclusion is clearly evident in Fig 8 and Fig 13. In the since 3D modes arise in the cylinder wake for even for Ri=0.00 and are extended for higher Ri. The assessment of this result is demonstrated in detail in the results for Strouhal number and mean velocity profiles for 2D simulations for both unheated and heated cases. The mesh generation for the 3D model is a clear extension of the 2D one. The length of the cylinder is about 7d to model the experimental settings in Hu et al. [10]. The possibility of extruding the superficial mesh in the spanwise direction allows the use of wall boundary condition for the lateral walls. Also assuming that the flow is symmetric about the center plane (transverse plane), in the spanwise direction (z-direction), a symmetry boundary condition can be used for the center plane. This reduces the total number of cells and in turn the computational time by half.

2. 2D numerical simulations

2D simulations help us to analyze the influence of the domain size and grid resolution. It has also highlighted that to obtain the correct vortex shedding frequency, a fine circumferential discretization is needed even for the unheated case. All the meshes both for 2D and 3D models are structured with quadrilateral and hexahedral cells respectively. The choice is tied to the need of using higher order numerical schemes available in FLUENT, only for such type of grids.

Table 4. 2D simulation settings

<table>
<thead>
<tr>
<th>Settings</th>
<th>Choice</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation</td>
<td>2D</td>
</tr>
<tr>
<td>Solver</td>
<td>Segregated implicit, Unsteady, Laminar</td>
</tr>
<tr>
<td>Temporal Discretisation</td>
<td>Second order</td>
</tr>
<tr>
<td>Pressure</td>
<td>QUICK</td>
</tr>
<tr>
<td>Momentum equation</td>
<td>QUICK</td>
</tr>
<tr>
<td>Pressure-Velocity coupling</td>
<td>PISO</td>
</tr>
<tr>
<td>Energy equation</td>
<td>QUICK</td>
</tr>
<tr>
<td>Inlet boundary condition</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>Outlet boundary condition</td>
<td>Pressure outlet</td>
</tr>
<tr>
<td>Lateral wall boundary condition</td>
<td>Wall</td>
</tr>
<tr>
<td>Total cells amount</td>
<td>32391</td>
</tr>
</tbody>
</table>

It will be discussed later in the results section that the wake behavior is no more 2D in nature if the AR is low. This makes the results obtained from the 2D simulations difficult to interpret and validate.

3. 3D numerical simulations

The 3D numerical settings are as represented in the Table 2.7. As described earlier, the 3D model is an extension of the 2D model with all the solver settings and spatial and temporal discretization consistent with the 2D simulation settings. In the sections to follow all the results for both 2D and 3D simulations are discussed.
Figure 3. 2D grid system
### III. Results and Discussions

#### A. Streamlines and Contours

Streamlines indicate the flow pattern at a given instant of time. The influence of buoyancy on the fluid flow in the wake region is shown by plotting the instantaneous streamlines shown in Fig. 5, for Ri=0.00 and Ri=1.05. Here the separation angle $\theta_{sep}$, measured from the forward stagnation point is higher for Ri=0.0. The wake region is marked by a zone of reverse flow very near the cylinder surface, which can be seen in form of negative streamwise velocity vectors due to the existence of the recirculation zone downstream of the cylinder (Fig. 6). With the increasing temperature of the cylinder (Ri), the separation angle $\theta_{sep}$ decreases since the direction of the flow and buoyancy oppose each other. This results in a wider wake and the recirculation zone for heated cylinder case becomes longer than the unheated cylinder. The near wake patterns, thus obtained can be seen in Fig. 7. As can be observed, the vortex shedding pattern for the unheated cylinder (Ri=0.0) is severely altered by the buoyant force.
Figure 5a. Streamlines for Ri=0.0

Figure 5b. Streamlines for Ri=1.05

Figure 6. z-Vorticity near the cylinder Ri=0.00

Figure 7. z-vorticity near the cylinder Ri=1.05
B. Time-averaged wake centerline velocities

Numerical results for the time-averaged mean centerline velocities along the vertical axis of the cylinder in the wake region on the symmetry plane are shown in Figure 9, for Ri=0.00, Ri=0.72 and Ri=1.05. The experimental observations of Hu et al. [10] are also presented for the purpose of comparison. As discussed earlier, the wake region is marked by a zone of reverse flow very near the cylinder surface be seen in the form of negative velocity values due to the existence of the recirculation zone downstream of the cylinder. But the wake behavior strongly depends on the end conditions. If we have a no-slip wall boundary condition as our end conditions for cylinders with low AR, we are certain to see 3D wake behavior even for Re<180 as mentioned earlier Mittal [2].

This is also clearly represented in Fig. 8, which shows the time-averaged wake centerline velocities for 2D numerical simulation comparing it to the experimental data and 3D numerical results. This indicates that the 2D simulation is unable to capture the flow phenomenon correctly. For the unheated cylinder case Ri=0.00, the velocity deficit is found to recover gradually with the increasing downstream distance around X/D>3 as expected Hu et al. [10]. With the increasing Richardson number, the stronger influence of buoyancy results in a stronger reversing flow near the trailing end of the cylinder shown in . Consequently the strong recirculation zone is pushed further downstream and the recovery rate of the streamwise velocity deficit decreases (Hu et al. [10]). This can be explained more clearly by the time-averaged wake centerline velocities in Fig 9. The trend for the time-averaged centerline velocities from the present calculations gives a good agreement with the experimental observations Hu et al. [10]. According to the definition given by Hu et al. [10], wake closure length lc is the distance from the rear end of the heated cylinder to the point where the time-averaged mean velocity becomes zero and changes its sign. The wake closure length for unheated cylinder from the present result differs by almost 13% from the experimental result. Whereas for the heated cases of Ri=0.72 and Ri=1.05 it matches well within 10% with the experimental results.

C. Pressure Distribution and Time-averaged Drag Coefficient

The pressure distribution over the cylinder is shown in Figure 11, for Ri=0.0, Ri=0.5 and Ri=1.05. It is known that the total drag comprises of drag due to the skin friction and pressure drop. As the temperature increases, the influence of buoyancy increases. Hence the drag due to the skin friction decreases marginally owing to the decreasing viscosity of the surrounding fluid while the drag due to the pressure drop increases substantially. Thus the drag due to skin friction can be neglected for higher Ri cases for the heated cylinder and pressure drag becomes dominant in the total drag calculation. The mean pressure drag is calculated as

\[
\bar{D} \cong \int p dA
\]
Figure 10. Schematic of pressure distribution over cylinder at symmetry plane.

Figure 11. Mean Pressure distribution over the cylinder in contra flow.

Figure 12. Time-averaged Drag Coefficient as function of Ri.

Figure 13. Vortex Shedding Frequency

D. Vortex Shedding Frequency

The phenomenon of vortex shedding from the bluff bodies has been studied since the pioneering work of Von Karman and Strouhal. The Strouhal Number is the measure of the oscillating fluid flow phenomenon in the wake region, and defined using the flow parameters by the following relation:

\[ St = \frac{f \times d}{V_{\infty}} \]  

(10)

where \( f \) is the vortex shedding frequency, \( d \) is the diameter of the cylinder and \( V_{\infty} \) is the free steam velocity. In the present study the Strouhal number for the six varied cases is deduced from the signal traces of the temporal
lift force coefficient Strouhal Number and is compared with some of the existing experimental and numerical investigations for Re=100 and Ri=0.0. This can be seen in Table 6. A good agreement can be noticed with the studies listed in this table. Thus for the unheated cylinder case the 2D calculations produce a good accuracy in terms of vortex shedding frequency. However as we increase the temperature of the cylinder (Ri>0.19), we see variations in the results from the 2D and 3D calculations. This is represented in Table 7 and Figure 15. The results obtained for the heated cylinder case where the Ri>0.00, are compared to the results from Hu et al. [10] and Chang and Sa [8]. In the discussion for the numerical methodology earlier, we have discussed that all the fluid properties like density, viscosity, thermal conductivity and thermal expansion coefficient varying as a function of the temperature.

Based on the same idea, Dumouchel et. al [11] and Wang et al. [12], introduced the concept of effect Reynolds number as a function effective temperature defined as $T_{eff} = T_w + 0.28(T_w - T_x)$ which can be used to predict the vortex shedding frequency. However it is already brought to our attention by Hu et al. [10] that this idea is inapplicable to our present study since Dumouchel et. al [11] and Wang et al. [12] have studied the effect of heating only for very small temperature differences ie. Ri<0.02. For small temperature differences the buoyancy effects can be studied using a Boussinesq approximation in the governing equations. This approximation cannot be implemented for large temperature differences as is the case in the present study.

**Table 6.** Comparison of St at Ri=0.0

<table>
<thead>
<tr>
<th>Re</th>
<th>Researcher</th>
<th>Method</th>
<th>St</th>
</tr>
</thead>
<tbody>
<tr>
<td>130</td>
<td>Present Study</td>
<td>2D-Cal</td>
<td>0.171</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3D-Cal</td>
<td>0.173</td>
</tr>
<tr>
<td>100</td>
<td>Hu (2004)</td>
<td>Exp</td>
<td>0.170</td>
</tr>
<tr>
<td></td>
<td>Noto and Fujimoto(2006)</td>
<td>2D-Cal</td>
<td>0.168</td>
</tr>
<tr>
<td></td>
<td>Zhang(1995)</td>
<td>2D-Cal</td>
<td>0.173</td>
</tr>
<tr>
<td></td>
<td>Persillon(2001)</td>
<td>3D-Cal</td>
<td>0.164</td>
</tr>
<tr>
<td></td>
<td>Sa and Chang(1990)</td>
<td>2D-Cal</td>
<td>0.157</td>
</tr>
<tr>
<td></td>
<td>Williamson (1996)</td>
<td>Exp</td>
<td>0.204</td>
</tr>
</tbody>
</table>

The results shown in Table 7 and Fig. 13, reveal that the 2D simulation for the flow over heated cylinder does not produce accurate results for vortex shedding with Ri>0.3 in comparison to the experimental data of Hu et al. [10] and Chang and Sa [8]. In fact the 2D numerical calculations seem to over predict the vortex shedding frequency for Ri>0.3. Whereas the 3D numerical calculations match very close, almost within 2-10% with the experimental results with the increasing Ri. This forms the first step to understanding the possibility of 3D transition being extended for the Re=130 and Ri>0.02 and we could conclude that 2D numerical simulations are certainly incapable of capturing the flow physics due to the buoyancy effects. For the same reason all the results have been explained only for 3D numerical simulations and not 2D numerical simulations.

The 3D transition and its relation to the vortex shedding for higher Ri will be more clear from the explanation in the following results. The difference in the decreasing trend between the experimental data and the numerical results of Chang and Sa [8] has been explained in detail by Hu et al. [10] For lower Richardson number, Ri=0.31, the vortex shedding pattern in the wake of the heated cylinder is similar to the pattern for the unheated cylinder. However the vortex shedding was found to be delayed and occurs further downstream with the increasing Ri=0.5. Although the numerical simulation by Chang and Sa [8] involved a 2D calculation, it seems to give better agreement with the experimental results than the 2D calculation for
E. Averaged Nusselt Number

The knowledge of the heat transfer coefficients at low Re is invaluable to designers and engineers. Here the averaged Nusselt number is the parameter of interest.

\[
\overline{Nu} = \frac{hD}{k} = \frac{\overline{QD}}{A^2k(T_w - T_\infty)}
\]  

(11)

Here \( \overline{Nu} \) is the averaged Nusselt number and \( \overline{Q} \) is the averaged heat transfer rate. In this case we have a mixed convection resulting from the heat transfer by forced and free convection. The experimental data by Hu et al. [10] suggests that with the increasing Ri, \( \overline{Nu} \) decreases. Fig. 14, shows the comparison of the results obtained from the present numerical study with the experimental results of Hu et al. [10]. The numerical results indicate the same trend as the experimental results. However it seems to over predict the \( \overline{Nu} \) by around 10%. But the results clearly indicate that with the increasing Ri the effect of buoyancy increases. The forced convection and free convection are in opposite direction which results in the decreasing trend.

F. Thermal effect of buoyancy on 3D wake flow

For flows with small AR (less than 28) or finite length of cylinders, we would see a 3D transition of wake behavior different from that for the cylinders with infinite length or large AR. From the studies by Williamson [1], for the unheated cylinders with large AR the transition regime would be between 190-250. However this transition regime is extended and/or delayed for a cylinder with small AR with “no-slip” walls Mittal [2]. This suggests that besides Reynolds number, the end-effects and aspect-ratio (AR) plays a major role in the nature of the flow behind a cylinder. In the present study, the cylinder with a small AR = 7 has been considered. In addition to this, a no-slip boundary condition has been implemented for the walls. These factors significantly affect the wake behavior of the flow although the Re=130 does not lie in the usual transition regime of 190-250. Instead we see a 3D wake behavior for flow at this Re which is also depicted in various results compared above. We have already discussed the influence of heat input and buoyancy on the wake behavior in terms of the vortex shedding frequency, time-averaged centerline velocities, time-averaged drag coefficient and Nusselt number. We have also mentioned that all these results are also influenced significantly by the end conditions which lead to 3D wake behavior.

Fig. 16-22, show the instantaneous pressure, velocity components and vorticity components for the temporally periodic flow for Ri=0.0 when the cylinder is not heated. We are looking at the yz section passing through the axis of the cylinder. We can observe the spatial development of the boundary layer at the wall
due to the no-slip boundary condition and we do not observe vortex shedding here due to the presence of the boundary layer. It is clear that the wake becomes very unstable because of the heating. This would suggest that heating and small aspect ratio would prolong or delay onset of transition regime.

![Figure 16a. Ri=0.00](image1.png)

![Figure 16b. Ri=1.05](image2.png)

Instantaneous pressure contours on yz plane

![Figure 17a. Ri=0.0](image3.png)

![Figure 17b. Ri=1.05](image4.png)

Instantaneous x-velocity component at yz plane
Figure 18a. \( Ri=0.0 \)
Instantaneous y-velocity component at yz plane

Figure 18b. \( Ri=1.05 \)
Instantaneous y-velocity component at yz plane

Figure 19a. \( Ri=0.0 \)
Instantaneous y-velocity component at yz plane

Figure 19b. \( Ri=1.05 \)
Instantaneous y-velocity component at yz plane
Figure 20a. Ri=0.0
Instantaneous x-vorticity component at yz plane

Figure 20b. Ri=1.05

Figure 21a. Ri=0.0
Instantaneous y-vorticity component at yz plane

Figure 21b. Ri=1.05
The transition to the three-dimensionality in the wake can be conveniently illustrated using the relationship between the Strouhal and Reynolds numbers. In the Figure 22, Williamson [1] shows that the first discontinuity of the Strouhal frequency occurs at Re at 180-190 (mode A). The wake in this mode is associated with vortex loops and formation of streamwise vortex pairs due to the formation of primary vortices. The wavelength of the vortex deformations is 3-4 cylinder diameters. With the increasing Re from 230-260, we see another discontinuity (mode B) with the wavelength of streamwise vortex structures as one cylinder diameter approximately. During the transition, the time histories of the lift forces are associated with two peaks: a peak at lower frequency that corresponds to mode A that gradually diminishes and a peak at higher frequency that takes over as the Re increases.
Williamson [1] attributes this to the intermittent swapping between the two modes. A similar trend is observed for Ri=1.05 shown in Figure 24 and Figure 25. The temporal history of lift force shows two different modes. One of the modes diminishes with time and the other one becomes dominant.

In this case, the transition that would start in Re range 190-250 seems to be extended because of heating. However more work is needed to understand the effect of heating and small aspect ratios on transition.

The detailed study of the vortex patterns and wake instability would aid in the understanding of the same. In addition to this, a large number of Ri would give more insight into this phenomenon.

![Figure 24a. Ri=0.0](image1)
![Figure 24b. Ri=1.05](image2)

**Figure 24a.** Ri=0.0
Iso-surface for x-vorticity component (=0.5) for the unsteady solution

**Figure 24b.** Ri=1.05

**IV. CONCLUDING REMARKS**

The influence of buoyancy opposing the flow direction due to mixed convection is numerically simulated for flow past a circular cylinder with small aspect ratio (AR=7) at Re=130. The Richardson number based on the temperature of the cylinder varies between Ri=0.0 to 1.05. Numerical results are generated with time-accurate finite volume method based unsteady compressible Navier-Stokes equations. Most of the previous studies have been conducted with the slip boundary conditions. In the present study the influence of heating in addition to small aspect ratio of the cylinder and no-slip wall boundary conditions on the pressure distribution, time-averaged drag coefficient, time-averaged centerline velocities, Strouhal number, averaged Nusselt number and transition mechanism has been numerically investigated and validated with the available experimental results. Some of the broad conclusions can be outlined as follows:

- Due to the small aspect ratio with no-slip walls, we see a 3D wake for Re=130 unlike the large aspect ratio.
Under the influence of buoyancy the flow separation occurs earlier than when the cylinder is not heated. This results in a wider and longer wake for the heated cylinder and a narrower wake for the unheated cylinder.

The wake closure length increases with the Richardson number and consequently the averaged drag coefficient is higher for the heated cylinder. The drag has a significant contribution from the pressure drop whereas the drag due to skin friction is negligible.

The vortex shedding frequency in terms of Strouhal number almost decreases with the increasing Richardson.

The averaged Nusselt number follows the same decreasing trend as the experimental results with the increasing Richardson number.

Preliminary analysis for influence of buoyancy on the wake behavior at Re=130 for cylinder in contra flow suggests that heating instigates an early transition. It is interesting to observe the significance of small aspect ratio and end effects in addition to the heating, on the transition mechanism. However much more work needs to be done over a larger range of temperature and Reynolds number to determine the onset of transition before any certain conclusions can be drawn. Detailed analysis of the vortex shedding patterns and vortex dislocations can aid to the understanding of various modes in transition regime for a heated cylinder.

References