NUMERICAL SIMULATION OF FLOW AND FORCED CONVECTION HEAT TRANSFER IN CROSSFLOW OF INCOMPRESSIBLE FLUID OVER TWO ROTATING CIRCULAR CYLINDERS

Nikolay Pavlovich Moshkin* and Jakgrit Sompong

Received: Aug 20, 2009; Revised: Oct 13, 2009; Accepted: Oct 20, 2009

Abstract

In this paper, the problem of laminar two dimensional heat transfer from two rotating circular cylinders in cross flow of incompressible fluid under isothermal boundary condition is investigated. The study is based on the numerical solution of the full conservation equations of mass, momentum and energy for Reynolds numbers (based on cylinder diameter and velocity of uniform stream) up to 40 while Prandtl number ranges between 0.7 and 50. For the range of parameters considered, the study revealed that the rate of heat transfer decreases with the increase of speed of cylinders rotation for the gap between cylinders more than one diameter. The increase of Prandtl number resulted in an appreciable increase in the average Nusselt number. The streamlines and isotherms are plotted for a number of cases to show the details of the velocity and thermal fields.

Keywords: Heat transfer, flow over two rotating cylinders, incompressible fluid, finite difference method

Introduction

The flows of fluid and forced convection across a heated bluff body have been the subject of considerable research interest because of their relevance to many engineering applications. The flow past a cylinder is considered to be an ideal bluff body by which to study the important phenomena of heat and mass transfer. For instance, the knowledge of the hydrodynamic forces experienced by submerged cylindrical objects such as off-shore pipelines is essential for the design of such structures. Furthermore, because of changing process and climatic conditions, one also needs to determine the rate of heat transfer from such structures.

Heat transfer and fluid flow around a single rotating cylinder has been studied by several researchers, see for example recent works of Badr et al. (1989); Kang et al. (1999); Mahfouz and Bard (1999a, 1999b); Stojković et al. (2002); Mittal and Kumar (2003); Gshwendtner (2004).

The heat transfer and flow around two stationary or rotating circular cylinders can be
Numerical Simulation of Flow and Forced Convection Heat Transfer

considered as an elementary flow which is helpful in understanding the flow patterns, heat transfer mechanism, and hydrodynamic characteristics around multiple bluff bodies in engineering practice.

Numerical computations for the flow over multiple cylinders have intensified during the last decade of the previous century. Chang and Song (1990) used a vorticity-stream function method to compute the flow past a pair of cylinders in side-by-side and tandem arrangements at $Re = 100$. Flow visualization and force coefficients were shown to be in good agreement with experiments. Mittal et al. (1997) used a finite element method to simulate three configurations, side-by-side, tandem and staggered arrangements of the cylinder pair at $Re = 100$ and 1000. Again, the results compared well with experiments. Recently, Kang (2003) investigated numerically the characteristics of flow around two side-by-side circular cylinders in the range of low Reynolds number defines as $Re = U_\infty D/\nu$ (where $U_\infty$ and $\nu$ are the free-stream velocity and kinematic viscosity, respectively) over the range of $40 \leq Re \leq 160$, respectively, and the normalize gap spacing $g^* < 5$; he identified six kinds of wake patterns ($g^* = g/D$, where $g$ and $D$ are the distance between two cylinders surfaces and the cylinder diameter, respectively).

The problem of flow pass two rotating cylinders in side-by-side arrangement has not been investigated widely. Papers of Sungnul and Moshkin (2006, 2008) are devoted to study the self-motion of two rotating circular cylinders. Only two recent studies (Yoon et al., 2007, 2009) have been found that investigate the flow around two rotating circular cylinders in side-by-side arrangement in the range of $|\alpha| \leq 2$ for various gap spacing at $Re = 100$ ($\alpha$ is the rotational speed at the cylinder surface normalized by the free-stream velocity). Only one paper of Joucavel et al. (2008) has been found, which studies the thermal behavior of an assembly of rotating cylinders aligned in a cross-flow.

As described above, the effect of rotation for a single cylinder and of the gap spacing between two cylinders at rest in side-by-side arrangement on the corresponding flow and heat transfer has been studied by numerous researchers. Only a few researchers have studied the problem of flow pass two rotating cylinders in a side-by-side arrangement. However, the heat transfer and fluid flow past a pair of rotating circular cylinders in side-by-side arrangement have not been addressed at all. This paper presents a numerical investigation of the characteristics of the two-dimensional heat transfer and the laminar flow around two rotating circular cylinders in side-by-side arrangements. In order to consider the combined effects of rotation and spacing between the two cylinders on the flow and heat transfer, numerical simulations are performed at a various range of absolute rotational speeds ($1 |\alpha| \leq 2.5$), for different gap spacing and Reynolds number in the range $Re \leq 40$. Quantitative information about the flow and the heat transfer variables such as the local and average Nusselt number, pressure and friction coefficients on the cylinder surfaces is highlighted. The patterns of the flow and temperature fields are analyzed for a wide range of parameters.

The mathematical formulation of the heat/mass transfer problem of a flow past two rotating circular cylinders is described in the next section. The problem is recast in terms of a cylindrical bipolar coordinate system. The following sections present the details of the numerical algorithm based on the projection method to approximate the solution of the momentum equation, and the fractional step stabilizing correction method to approximate the solution of the energy equation. The validation of the numerical algorithm was done by comparing our computational results for large gap between the cylinders with available numerical and experimental data for flow and heat transfer over a single cylinder. The results of the various numerical experiments are reported and discussed in the final part of our paper.

Mathematical Formulation of the Problem

Consider the flow of a viscous incompressible fluid along the $y$-direction normal to the line
between the centers of two rotating circular cylinders with a constant velocity and temperature at infinity, \( U_\infty, T_\infty \). The cylinders rotate about their axes at angular velocities \( \omega_L \) and \( \omega_R \), assuming that a positive value corresponds to counter-clockwise rotation. A sketch of the flow geometry, coordinate system and notations are shown in Figure 1.

![Figure 1. Schematic representation of the flow and heat transfer over a pair of rotating circular cylinders in side-by-side arrangement](image)

The present study is restricted to long cylinders and low Reynolds number, \( Re < 40 \). The flow across the cylinders is steady and two-dimensional. All flow variables are functions of the coordinates, \( x \) and \( y \) alone. The thermodynamical properties of the fluid (density, specific heat capacity \( c_p \), thermal conductivity \( k \)) are assumed to be independent of temperature. Under these conditions, the momentum and energy equations are not coupled.

For the study of heat transfer and fluid flow around two cylinders, the cylindrical bipolar coordinate system is the most suited. The ubiquitous definition of the cylindrical bipolar coordinates \((\xi, \eta, z)\) is

\[
\begin{align*}
    x &= a \sinh \eta \left( \cosh \eta - \cos \xi \right), \\
    y &= a \sin \xi \left( \cosh \eta - \cos \xi \right), \\
    z &= z,
\end{align*}
\]

where \( \xi \in [0, 2\pi], \eta \in (-\infty, \infty) \) and \( z \in (-\infty, \infty) \), \( a > 0 \) is the characteristic length in the cylindrical bipolar coordinates (so-called “focal distance”).

The following identities show that curves of constant \( \xi \) and \( \eta \) are circles in the \( xy \)-space

\[
\begin{align*}
    x^2 + (y - a \cot \xi)^2 &= a^2 \csc^2 \xi, \\
    (x - a \coth \eta)^2 + y^2 &= a^2 \csch^2 \eta.
\end{align*}
\]

Figure 1 shows the two cylinders described by \( \eta = \eta_R \) (with \( \eta_R > 0 \)) and \( \eta = \eta_L \) (with \( \eta_L < 0 \)), respectively. The cylinders’ radii \( r_L \) and \( r_R \) and the distances of their centers from the origin \( d_L \) and \( d_R \) are given by

\[
\begin{align*}
    r_i &= a \csc |\eta_i|, \\
    d_i &= a \coth |\eta_i|, \quad i = L, R
\end{align*}
\]

The center-to-center distance between the cylinders is equal to \( d = d_L + d_R \). If \( r_L + r_R \) and \( d \) are given, one can find \( a \), \( \eta_L \) and \( \eta_R \) from relations (1)-(3) as follows

\[
\eta_{L,R} = \ln \left[ \frac{d^2 + r_L^2 + r_R^2}{2dr_Lr_R} \left( \frac{d^2 + r_L^2 - r_R^2}{2dr_Lr_R} \right)^{-1} \right]
\]

\[
a = \sqrt{\frac{d^4 - 2d^2(r_L^2 + r_R^2) + (r_L^2 - r_R^2)^2}{4d^2}}
\]

The fluid flow is governed by the conservation laws of momentum, mass and energy. The nondimensional form of the governing equations in the cylindrical bipolar coordinate system are

\[
\begin{align*}
    \frac{\partial v_x}{\partial t} + \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= - \frac{1}{k} \frac{\partial \rho}{\partial \xi}, \\
    \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= \frac{1}{k} \frac{\partial \rho}{\partial \eta},
\end{align*}
\]

\[
\begin{align*}
    \frac{\partial v_x}{\partial t} + \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= - \frac{1}{k} \frac{\partial \rho}{\partial \xi}, \\
    \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= \frac{1}{k} \frac{\partial \rho}{\partial \eta},
\end{align*}
\]

\[
\begin{align*}
    \frac{\partial v_x}{\partial t} + \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= - \frac{1}{k} \frac{\partial \rho}{\partial \xi}, \\
    \frac{1}{h} \left( \frac{\partial v_x}{\partial \xi} + \frac{\partial v_y}{\partial \eta} \right) - \frac{1}{a} \left( v_y v_z \sinh \eta - v_x v_z \sin \xi \right) &= \frac{1}{k} \frac{\partial \rho}{\partial \eta},
\end{align*}
\]
Numerical Simulation of Flow and Forced Convection Heat Transfer

\[
\frac{1}{\kappa} \left( \frac{\partial (\kappa \nu_{\xi})}{\partial \xi} + \frac{\partial (\kappa \nu_{\eta})}{\partial \eta} \right) = 0, \tag{7}
\]

\[
\frac{\partial \nu_{\xi}}{\partial t} + \frac{1}{\kappa} \left( \nu_{\xi} \frac{\partial \nu_{\xi}}{\partial \xi} + \nu_{\eta} \frac{\partial \nu_{\xi}}{\partial \eta} \right) = -\frac{2}{\kappa} \frac{1}{Re} \left( \frac{\partial^2 \nu_{\xi}}{\partial \eta^2} + \frac{\partial^2 \nu_{\xi}}{\partial \xi^2} \right) \tag{8}
\]

where \( \nu_{\xi} \) and \( \nu_{\eta} \) are the velocity components in \( \xi \) and \( \eta \) directions, respectively, \( p \) is the pressure, \( t \) is the time, \( T \) is the fluid temperature, and \( h = a (\cosh \eta - \cos \xi) \). In the above equations all quantities are rendered dimensionless, the velocities by means of the free stream velocity \( U_\infty \), all lengths by means of the radius \( r_R \) of right cylinder, time by \( r_R / U_\infty \), pressure by \( \frac{p}{\rho U_\infty^2} \), and temperature by \( \frac{T - T_R}{T_R - T} \). The two nondimensional parameters which appear in the above equations are Reynolds number, \( Re = \frac{2 \rho U_\infty r_R}{\mu} \), and Prandtl number, \( Pr = \frac{c_p \mu}{k} \) where \( \mu \) is the viscosity of the fluid, \( \rho \) is the fluid density. On the cylinder surfaces the constant temperature condition and the no-slip condition for the velocity vector are imposed. The constant streamwise velocity and the uniform temperature are specified at infinity.

**Discretization of the Governing Equations**

**Description of Grid**

The computational domain is divided into a mesh by points \( \xi_j = (j - 0.5) \Delta \xi \) and \( \eta_i = \eta_i + (i - 0.5) \Delta \eta \) where \( \Delta \xi = 2 \pi N, \Delta \eta = (\eta_R - \eta_L) / M \) are spatial mesh sizes in both \( \xi \) and \( \eta \) directions, respectively. A staggered placement of variables is used with velocity components \( u = \nu_{\eta} \) located on the vertical sides of each cell and \( v = \nu_{\xi} \) on the horizontal sides of each cell. We used the fractional indexes to denote grid values of variables at time \( t_n = n\tau \) where \( \tau \) is the step size in time.

**Discretization of Navier-Stokes Equations**

Simulating incompressible flows presents a difficulty of satisfying the property of mass conservation. The velocity field must satisfy the incompressibility constraint, which reflects the inability of pressure to do compression work. For developing numerical approximations to this problem, it is natural to exploit the techniques of the fractional step projection method of Chorin (1968). The main idea of the fractional step projection method is the splitting of the viscosity effect from the incompressibility, which are dealt with in two separate subsequent steps.

The time derivatives are represented by forward differences. In case of a steady solution, time is considered as artificial (iterative) time. If the integer \( n \) represents the time level, then the intermediate velocity field can be calculated from

\[
\tilde{v}_{\xi+1/2} = \nu_{\xi+1/2} - \tau (CONV)_{\xi+1/2} + \frac{2}{Re} (DIFFV)_{\xi+1/2}, \tag{9}
\]

\[
\tilde{u}_{\eta+1/2} = u_{\eta+1/2} - \tau (CONU)_{\eta+1/2} + \frac{2}{Re} (DIFFU)_{\eta+1/2}. \tag{10}
\]

Here we used the following notation for convective and diffusive terms

\[
CONV = \frac{1}{h} \left( \frac{\partial \nu_{\xi}}{\partial \xi} + \frac{\partial \nu_{\eta}}{\partial \eta} \right) a \{ \nu_{\eta} \sin \xi - \nu_{\xi} \sin \eta \},
\]

\[
CONU = \frac{1}{h} \left( \frac{\partial \nu_{\xi}}{\partial \eta} + \frac{\partial \nu_{\eta}}{\partial \xi} \right) a \{ \nu_{\xi} \sin \eta - \nu_{\eta} \sin \xi \},
\]

\[
DIFFV = \frac{1}{h} \left[ \frac{\partial^2 \nu_{\xi}}{\partial \eta^2} + \frac{\partial^2 \nu_{\xi}}{\partial \xi^2} \right] - \frac{2}{a} \left( \sin \xi \frac{\partial \nu_{\eta}}{\partial \xi} - \sin \eta \frac{\partial \nu_{\xi}}{\partial \eta} \right),
\]

\[
DIFFU = \frac{1}{h} \left[ \frac{\partial^2 \nu_{\eta}}{\partial \xi^2} + \frac{\partial^2 \nu_{\eta}}{\partial \eta^2} \right] - \frac{2}{a} \left( \sin \xi \frac{\partial \nu_{\eta}}{\partial \xi} - \sin \eta \frac{\partial \nu_{\eta}}{\partial \eta} \right).
\]

The second order central difference scheme has been used to discretize both the convective and diffusive terms. The velocity components \( \tilde{v}_{\xi+1/2} \) and \( \tilde{u}_{\eta+1/2} \) are computed for
all faces of the cells except one, where the velocity components are given by the boundary condition. Figure 2 shows the location of grid points where velocity components are known from the no-slip boundary condition or from the boundary condition at infinity. The boundary condition at infinity is shifted on the boundary of finite domains $\Omega_1$ and $\Omega_2$.

The explicit advanced tilde velocity may not necessarily lead to a flow field with zero mass divergence in each cell. This is because at this stage the pressure field not used. Pressure $p^{s+1}$ and velocity components $u^{s+1}$ and $v^{s+1}$ have to be computed simultaneously in such a way that no net mass flow takes place in or out of a cell. In such case, we make use of an iterative correction procedure in order to obtain a divergence free velocity field. First, the velocity components are updated in the following form

\begin{align}
\tilde{u}_{i+1/2,j}^{s+1} &= \tilde{u}_{i+1/2,j} - \frac{\tau}{h_{i+1/2,j}} \left( \frac{p_{i+1,j}^{s+1} - p_{i,j}^{s+1}}{\Delta \xi} \right), \\
\tilde{v}_{i,j+1/2}^{s+1} &= \tilde{v}_{i,j+1/2} - \frac{\tau}{h_{i,j+1/2}} \left( \frac{p_{i+1,j}^{s+1} - p_{i,j}^{s+1}}{\Delta \eta} \right),
\end{align}

where the index $s$ is used to denote iteration number, $s = 1, 2, 3, \ldots$. In the case $s = 1$, we assume that $p^{s+1,0} = p^s$. The point iterative pressure equation becomes

\begin{equation}
\tilde{p}_{i,j}^{s+1} = \tilde{p}_{i,j}^{s+1,0} - \frac{\beta}{\tau} \text{div}(u, v)^{s+1},
\end{equation}

where $\text{div}(u, v)^{s+1}$ is the unsatisfied divergence at the $(i, j)^{th}$ cell due to incorrect velocities $\tilde{u}_{i+1/2,j}^{s+1}$ and $\tilde{v}_{i,j+1/2}^{s+1}$.

The pressure (velocity) equations (11)-(13) are to be iterated until the continuity equation is satisfied to the prescribed accuracy and then the computation proceed to next time step (artificial time)

\begin{equation}
\text{div}(u, v)^{s+1} \leq 10^{-6},
\end{equation}

where the discretized form of continuity equation is

\begin{align}
\text{div}(u, v)^{s+1} &= \frac{1}{h_{i,j}^2} \left[ \frac{(hv)_{i+1/2,j}^{s+1} - (hv)_{i-1/2,j}^{s+1}}{\Delta \xi} \right] \\
&\quad - \frac{1}{h_{i,j}^2} \left[ \frac{(hu)_{i,j+1/2}^{s+1} - (hu)_{i,j-1/2}^{s+1}}{\Delta \eta} \right].
\end{align}

The optimal value of the relaxation parameter $\beta$ was found by trial and error.

The pressure advanced equation (13) can be interpreted as Jacobi iterative method to
solve the finite-difference analog of Poisson equation for pressure. Let iterative process converges, i.e., \( v^{n+1} = v^{n+1, s} \), \( u^{n+1} = u^{n+1, s} \), and \( p^{n+1} = p^{n+1, s} \). Substituting (11) and (12) into (15) and requiring that \( \text{div}(u, v)^{n+1, s}_{i,j} = 0 \), we get the following finite-difference approximation of Poisson equation for pressure,

\[
\text{div}(\nabla p^{n+1})_{i,j} = \frac{1}{\tau} \text{div}(\bar{u}, \bar{v})_{i,j}.
\]

(16)

Finite difference equation (16) is an algebraic system with respect to the unknown vector \([v^{n+1}_{i,j}, p^{n+1}_{i,j}, v^{n+1}_{i,j}, p^{n+1}_{i,j}, v^{n+1}_{i,j}, p^{n+1}_{i,j}, \ldots] \). System of equations (16) can be solved by the method of false transient

\[
\left( \frac{\partial p^{n+1}}{\partial \beta} \right)_{i,j} = \text{div}(\nabla p^{n+1}) - \frac{1}{\tau} \text{div}(\bar{u}, \bar{v}),
\]

(17)

where \( \beta \) is an iterative parameter. Equation (13) is a particular case of (17) where the forward finite difference is used to approximate the left-hand-side in (17),

\[
\left( \frac{\partial p^{n+1, s}}{\partial \beta} \right)_{i,j} = \frac{p^{n+1, s}_{i,j} - p^{n+1, s}_{i,j}}{\beta}.
\]

(18)

On the surface of the cylinders, the no-slip condition is applied, which is equivalent to setting the tangential velocity at the boundary to \( \alpha_i, \ i = L, R \) and the normal velocity to zero.

Where \( \alpha_i \) is defined as \( \alpha_i = \omega_i D / \sqrt{2u_x} \), \( \omega_i \) is the angular velocity of the cylinders. Implementation of no-slip and no-penetration boundary conditions is straightforward. Because of the staggered arrangement of the variables, we used the second order one-side finite differences to approximate the derivatives \( \frac{\partial v}{\partial \eta} \) and \( \frac{\partial^2 v}{\partial \eta^2} \).

In a numerical simulation, it is impossible to satisfy constant streamwise velocity as \( x^2 + y^2 \rightarrow \infty \). Usual practice involves the placement of the conditions at a faraway (“artificial”) boundary, which is located at a large distance from the body. In our computations, the far boundary coincides with lines \( \xi = \text{const} \) and \( \eta = \text{const} \) (recall that images of the infinity in computational domain are two points \((\xi = 0, \eta = 0)\) and \((\xi = 2\pi, \eta = 0)\)). To be more exact, we choose far boundary as the boundary of the following domain

\[
\Omega_1 = \{(\xi, \eta) | (\eta = \varepsilon \xi, 0 \leq \xi \leq \xi_1) \}
\]

\[
\cup (\xi = \xi_1 \text{ and } -\varepsilon \xi \leq \eta \leq \varepsilon \eta_1)
\]

\[
\Omega_2 = \{(\xi, \eta) | (\eta = \varepsilon \xi, 2\pi - \varepsilon \xi \leq \xi \leq 2\pi) \}
\]

\[
\cup (\xi = 2\pi - \varepsilon \xi \text{ and } -\varepsilon \xi \leq \eta \leq \varepsilon \eta_1)
\]

where \( \varepsilon_\eta = \pm(\kappa_\eta + 0.5)\Delta \eta \), \( \varepsilon_\xi = \pm(\kappa_\xi + 0.5)\Delta \xi \), and \( \kappa_\eta, \kappa_\xi \) are integer numbers. In Figure 2 a sketch of these domains \( \Omega_1 \) and \( \Omega_2 \) is shown by shadow regions.

At the nodes of the mesh which are located on the boundary of the regions \( \Omega_1 \) and \( \Omega_2 \) we assumed that the tangent component of velocity vector and pressure \( p = p \) are known.

\[
v = -\left( u_{\xi} \right)_L \frac{h}{a} \sinh \eta \sin \xi + \left( u_{\eta} \right)_L \frac{h}{a} (\cosh \eta \cos \xi - 1),
\]

\[
u = -\left( u_{\xi} \right)_R \frac{h}{a} (\cosh \eta \cos \xi - 1) - \left( u_{\eta} \right)_R \frac{h}{a} \sinh \eta \sin \xi.
\]

Here we have utilized the idea that prescribing tangent component of velocity and pressure gives a well-posed problem for the Navier-Stokes equations (Antontsev et al. (1990), Moshkin and Yambangwai (2009)). The normal to the boundary component of the velocity vector is computed from the requirement of continuity equation for the cells contained in this boundary. For example, for the case shown in Figure 2 the boundary of region \( \Omega_1 \) (or \( \Omega_2 \)) passes through points ‘N’, ‘S’, and ‘C’. The tangential component of velocity \( v_{\eta}^{n+1} \) and \( v_{\xi}^{n+1} \) are given by boundary condition as well as the pressure at point ‘C’ is equal to pressure at infinity. The component of velocity vector \( u_{\xi}^{n+1} \) is computed from the requirement of zero divergence for the cell centered at the point ‘C’.
Discretization of the Energy Equation

The momentum and energy equations are not coupled. The energy equation is solved separately from the Navier-Stokes equations. When the steady solution of Navier-Stokes equations is computed, the iterative method of stabilizing correction is used (Yanenko, 1971) to find the steady distribution of temperature. The structure of the scheme of stabilizing correction is the following:

- the first fractional step produces absolute consistency with the energy equation,
- all succeeding fractional steps are corrections and serve to improve the stability.

For the 2D case, the scheme possesses strong stability and satisfies the property of complete consistency. The requirement of complete consistency guarantees convergence of the unsteady solution to the steady solution for arbitrary time and space step size (Yanenko 1971).

To describe the scheme of stabilizing correction, consider a two dimensional convection-diffusion equation in the form

\[ \frac{\partial \psi}{\partial t} = -\nabla \cdot (\vec{v} \psi), \]

where \( \nabla = \frac{\partial}{\partial x} + \frac{\partial}{\partial y} \) is the gradient operator, and \( \psi \) is the Laplace operator. To find the steady solution of (19) we use the method of false transient. Consider the related unsteady problem with “fiction” time

\[ \frac{\partial T}{\partial t} = \Lambda_1 T + \Lambda_2 T, \]

where \( \Lambda_1, \Lambda_2 \) represent the derivatives in only one space direction. For the solution of (20) Douglas and Rachford (1956) proposed the following scheme

\[ \frac{T^{k+1/2} - T^k}{\tau} = \Lambda_1 T^{k+1/2} + \Lambda_2 T^k, \]  

\[ \frac{T^{k+1} - T^{k+1/2}}{\tau} = \Lambda_1 (T^{k+1} - T^k). \]

Eliminating \( T^{k+1/2} \) we can rewrite system (21) and (22) in the following uniform scheme

\[ \frac{T^{k+1} - T^k}{\tau} = \Lambda_1 T^{k+1} + \Lambda_2 T^{k+1} \]

It follows from this that scheme (21) and (22), and the equivalent scheme (23) approximate equation (20) with the same order of accuracy as the implicit scheme

\[ \frac{T^{k+1} - T^k}{\tau} = \Lambda_1 T^{k+1} + \Lambda_2 T^{k+1}. \]

Temperature references to nodes of main computational grid related with center of cells of discrete domain \( \Omega_k = \{(\xi_j, \eta_j) | \xi_j = \xi_0 + 0.5 \Delta \xi_j, \eta_j = \eta_0 + (i-0.5) \Delta \eta, j = 0, 1, ..., n, i = 0, 1, ..., m \} \).

On the boundary of the shadowed domains \( \Omega_1 \) and \( \Omega_2 \) the constant temperature of uniform stream \( T = T_u \) is prescribed. The constant temperature at the cylinder surfaces is approximated by the following

\[ \frac{T_{0,j} + T_{0,j}}{2} = T_{\xi}, \frac{T_{n,j} + T_{n,j}}{2} = T_{\eta}, \]

where \( T_{0,j} \) and \( T_{n,j} \) are “ghost” points, which introduced for convenience in writing the computational code. For each fractional step a system of linear algebraic equation with tridiagonal matrix is solved.

Computation of Heat Transfer and Flow Characteristics

Two groups of characteristic quantities are of interest in the present study, one for characterizing the forces at the cylinder surface
Numerical Simulation of Flow and Forced Convection Heat Transfer

and the other for the heat transfer. If $F_{xi}$ and $F_{yi}$, $i = L, R$ are the lift and drag forces on the cylinders; the lift and drag force coefficients are defined by

$$C_{li} = \frac{F_{li}}{\rho U_D^2 D}, C_{di} = \frac{F_{di}}{\rho U_D^2 D}, \quad i = L, R, \quad (25)$$

and each consists of components due to the friction forces and the pressure. Hence

$$C_L = C_{li} + C_{di}, \quad C_D = C_{di} + C_{pi}, \quad (26)$$

where

$$C_{di} = -\frac{1}{\rho U_D} \int \rho \tilde{n} \tilde{T}_i \, dS,$$

$$C_{li} = -\frac{1}{\rho U_D} \int \mu (\tilde{n} \times \tilde{v}) \tilde{T}_i \, dS,$$

$$C_{pi} = -\frac{1}{\rho U_D} \int \rho \tilde{n} \tilde{T}_i \, dS,$$

$$C_{pi} = -\frac{1}{\rho U_D} \int \mu (\tilde{n} \times \tilde{v}) \tilde{T}_i \, dS.$$

Here $\tilde{T}_i$ are the unit vectors in $x$ and $y$ axes directions, $\Sigma$ is the cylinder perimeter, and $\tilde{n}$ is the outward unit normal to surface vector. To evaluate the integrals we used the middle point rule, for example, in order to compute $C_{pi}$ we used formula:

$$C_{pi} = \sum_{i=1}^{\Sigma} \rho \left( \frac{h}{a} (\cosh \eta \cos \xi - 1) \right) n_i + \left( \frac{h}{a} \sinh \eta \sin \xi \right) \Lambda \xi, \quad (27)$$

where $i = L, R$, that is, we computed the coefficient on the left and right cylinder surfaces, respectively. Pressure on the cylinder surfaces is evaluated by extrapolation from interior points.

The vorticity in the cylindrical bipolar coordinate system is given by the following formula

$$\omega = \frac{1}{h} \left[ \frac{\partial v}{\partial \eta} - \frac{\partial u}{\partial \xi} \left( \cosh \eta - u \sin \xi \right) \right].$$

On the cylinder surfaces the tangent derivative is equal zero $\frac{\partial u}{\partial \xi} = 0$. The normal derivative $\frac{\partial v}{\partial \eta}$ is approximated by one-side second order finite-differences.

The important parameter of interest in heat transfer problems is the heat transfer rate per unit area from the cylinder wall to the ambient fluid. The local Nusselt number in the cylindrical bipolar coordinate system based on the diameter of the cylinder is

$$Nu = -\left( \frac{\partial T}{\partial \eta} \right)_T = -\nabla T \cdot \tilde{n} = -\frac{1}{h} \frac{\partial T}{\partial \eta}$$

$$= \frac{\cosh \eta - \cosh \xi}{a} \frac{\partial T}{\partial \eta}, \quad (28)$$

The average Nusselt number is calculated by averaging the local Nusselt number over the surface of the cylinder

$$\overline{Nu} = -\frac{1}{2\pi} \int_0^{2\pi} \frac{\partial T}{\partial \eta} \, d\xi, \quad \xi \in [0, 2\pi]. \quad (29)$$

The integral is approximated by the middle point rule.

Validations

The first task in any numerical work is to validate the codes ability to accurately reproduce published experimental and numerical results. Unfortunately, there are no data in the literature to verify the accuracy of the considered problem (flow over two rotating cylinders). Since for large gap spacing between the cylinders, the mutual influence of the cylinders on each other is negligible we can assume that the flow and heat transfer will be similar to flow and heat transfer over a single cylinder. The comparisons with flow and heat
transfer around a single cylinder can be viewed as a partial validation of the algorithm presented. Thus, the comparisons were carried out of the characteristics of flow and heat transfer as obtained from our numerical results (gap spacing $g = 14$, $g = (d - r_L - r_R)/D_R$) with the published data for a single cylinder. All computations were performed in a large domain in order to reduce the influence of the outer boundary. A sequence of uniform grids is used. The wake behind a single cylinder is steady in the flow regime $Re \leq 46 \pm 1$. We assume that in the case of two rotating circular cylinders, the steady regime exists at least for the same range of the Reynolds numbers. Due to symmetry with respect to the $y$ -axis, all our results are presented for the left cylinder only.

The important characteristics of flow are the drag coefficients, $C_D = C_{Df} + C_{Dp}$, the local and average Nusselt numbers. They are presented in Table 1 for $Re = 20$, $g = 14$, and $Pr = 0.7$. Table 1 shows convergence of our results on a sequence of grids and good agreement with results from other publications.

Representative results showing the variation of the pressure coefficient $C_{Dp}$ and vorticity $\omega$ on the surface of the left cylinder are plotted in Figure 3 for the two values of the Reynolds number $Re = 20$ and $40$ and for the gap spacing $g = 14$.

Comparison of our numerical simulations is performed with the numerical results of Dennis and Chang (1970) for $Re = 20$ and $40$, with the experimental results of Thom for $Re = 36$, $45$ and Apelt for $Re = 40$ (Batchelor, 2000). Our data come from simulations on the grid with $81 \times 81$ nodes in the $\xi - \eta$ plane. The closest distance from the cylinder surfaces to the far boundary for this grid is $53$ cylinder diameters. The angle variable $\theta$ is zero at the front stagnation point and increases in the clockwise direction on the left cylinder. The data

Table 1. Validations of the numerical algorithm. Effect of grid refinement upon $C_D$, $C_{Dp}$, $C_{Di}$ and average Nusselt number $\overline{Nu}$ for $Re = 20$ and $g = 14$

<table>
<thead>
<tr>
<th>Source</th>
<th>$C_D$</th>
<th>$C_{Dp}$</th>
<th>$C_{Di}$</th>
<th>$\overline{Nu}$ ($Pr = 0.7$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present (21 x 21)</td>
<td>2.149</td>
<td>1.274</td>
<td>0.875</td>
<td>2.669</td>
</tr>
<tr>
<td>Present (41 x 41)</td>
<td>2.112</td>
<td>1.274</td>
<td>0.838</td>
<td>2.481</td>
</tr>
<tr>
<td>Present (81 x 81)</td>
<td>2.064</td>
<td>1.242</td>
<td>0.822</td>
<td>2.478</td>
</tr>
<tr>
<td>Soares (2005) et al.</td>
<td>2.035</td>
<td>1.193</td>
<td>0.842</td>
<td>2.430</td>
</tr>
<tr>
<td>Stålberg et al. (2006)</td>
<td>2.052</td>
<td>1.229</td>
<td>0.823</td>
<td>-</td>
</tr>
<tr>
<td>Bharti et al. (2007)</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2.465</td>
</tr>
</tbody>
</table>

Figure 3. Validation of numerical algorithm (a)- pressure coefficient over cylinder surface, (b)-the vorticity distribution over cylinder surface, for the left cylinder at $Re = 20, 40$, and $g = 14$
from our simulations (∗ - sign for Re = 20 and o - sign for Re = 40) match the results from other publications.

To the authors’ knowledge, even for the flow past a single circular cylinder, there are only a few published sets of data pertaining to the drag and lift coefficients at Re ≤ 40 and nonzero rotation. Table 2 shows comparisons of our numerical results for Re = 20, g = 14 with those numerically obtained by Badr et al. (1989); Ingham and Tang (1990); Sungnul and Moshkin (2006) and Chung (2006). The left cylinder rotates with constant angular velocity in clockwise direction and the right cylinder rotates with the same speed in counterclockwise direction. The validation results inspire confidence in the reliability and accuracy of the numerical method used.

**Results**

Numerical computations were carried out for Re = 5 - 40, for a range of values of Prandtl number from 0.7 to 20, for range of rotation α = 0.1 + 2.5, and different gap spacing. In all cases we represent results obtained on a grid with 81 x 81 nodes in the computational domain.

The influence of the gap spacing on the heat transfer rate in terms of the local Nusselt number Nu(θ) is shown by Figures 4(a) and 4(b) for Re = 20, α = 0 and Pr = 0.7 and 20. These figures show that a significant influence of the distance between cylinders on the local Nusselt number Nu(θ) is observed for g ≈ 5.

**Table 2. Hydrodynamic parameters of flow over a rotating circular cylinder at Re = 20 with g = 14**

<table>
<thead>
<tr>
<th>Contribution</th>
<th>CD</th>
<th>CL</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>α = 0.1</td>
<td>α = 1.0</td>
</tr>
<tr>
<td>Present (21 x 21)</td>
<td>2.146</td>
<td>2.035</td>
</tr>
<tr>
<td>Present (41 x 41)</td>
<td>2.108</td>
<td>1.897</td>
</tr>
<tr>
<td>Present (81 x 81)</td>
<td>2.052</td>
<td>1.847</td>
</tr>
<tr>
<td>Sungnul &amp; Moshkin (2006)</td>
<td>2.120</td>
<td>1.887</td>
</tr>
<tr>
<td>Badr et al. (1989)</td>
<td>1.990</td>
<td>2.000</td>
</tr>
<tr>
<td>Ingham et al. (1990)</td>
<td>1.995</td>
<td>1.925</td>
</tr>
<tr>
<td>Chung (2006)</td>
<td>2.043</td>
<td>1.888</td>
</tr>
</tbody>
</table>

**Figure 4. Local Nusselt number for different gap spacing at Re = 20, α = 0, and**

(a)- Pr = 0.7, 
(b)- Pr = 20
Another aspect that seems to be interesting is the decreasing $Nu(\theta)$ for $0^\circ \leq \theta \leq 360^\circ$ with decreasing $g$ in the case of $Pr = 0.7$ (rate of convection and conduction almost equal). In the case of $Pr = 20$ (convection is the dominant mechanism of heat transfer) $Nu(\theta)$ decreases with $g$ decreasing for $0^\circ \leq \theta \leq 180^\circ$ and $270^\circ \leq \theta < 360^\circ$, and for $180^\circ \leq \theta < 270^\circ$ the values of $Nu(\theta)$ increases with $g$ decreasing. It is not surprising that the average Nusselt number for each individual cylinder increases with increasing gap spacing and tends to the average Nusselt number for a single cylinder. Figure 5 shows the variation of the average Nusselt number with gap spacing between non-rotating cylinders at different Prandtl numbers and fixed Reynolds number, $Re = 20$.

As expected, the average Nusselt number for the cylinders increases with the Prandtl number.

**Large Gap Spacing**

The effect of steady rotation of the cylinders on heat transfer is first studied for large gap spacing, $g = 14$. Figure 6 shows the variation of the local Nusselt number $Nu(\theta)$ at $Re = 20$, $Pr = 0.7$ and $Pr = 20$, and for different values of $\alpha$ ($\alpha \in [0, 2]$). An increase in the rotation of the cylinders leads to a displacement of the points of maximum and minimum Nusselt number in the direction of rotation. At the same time, the maximum value of the local Nusselt number decreases with increase of rotational speed. The minimum value of the local Nusselt number slightly increases in step with increasing rotation. For $Pr = 20$ and high rotational speed $\alpha = 2$, the rate of heat transfer around the cylinder surface becomes almost uniform as shown in Figure 6(b). This behavior is quite expected, since due to the no-slip condition the fluid layer adjacent to the cylinder enwraps the cylinder and rotates with almost the same angular velocity. Increasing rotational speed to $\alpha = 2$ creates a thick rotating layer (buffering layers) around the cylinder. The heat transfer through that layer is mostly due to thermal conduction.

Figure 7 shows streamline patterns and temperature contours plot for large gap spacing $g = 14$, Reynolds number $Re = 20$.
$Pr = 0.7, 20$, and $\alpha = 0.1, 1.0, 2.0$. The rotation effect substantially changes the flow pattern in the vicinity of the cylinder. Without rotation, the flow field exhibits a symmetric pair of standing vortices behind the cylinder defining a closed recirculation region. With the increase of the rotation rate, the flow becomes asymmetric: the vortex detaches from the cylinder and the stagnation point rotates in the direction opposite to the direction of cylinder rotation departing from the surface of the cylinders (see first row in Figure 7). The minimum of the local Nusselt number is observed near this point. In downstream direction, the streamlines are shifted clockwise and the same occurs for the isotherms (see second and third rows in Figure 7) resulting in the asymmetrical distribution of the local Nusselt number as shown in Figure 6. The oncoming fluid stream is accelerated by the rotating cylinder on the ‘west’ side of an egg-shaped region of closed streamlines. As a result, the convective heat transfer between the ambient fluid and the fluid within the egg-shaped region is increased at the point of maximum velocity. This observation explains the deviation of the maximum of the local Nusselt number in the direction of cylinder’s rotation. An increase in the Prandtl number increases the compactness of the isotherms toward the downstream direction (compare second and third rows of Figure 7). Owing to rotation a wake-shapes region of

![Streamline patterns and temperature contours](image)

**Figure 7.** Streamline patterns (first row), temperature contours over two circular cylinders at $Re = 20$, $g = 14$, $\alpha = 0.1, 1.0$ and $2.0$ and for $Pr = 0.7$, (second row) and $Pr = 20$ (third row)
is otherms is shifted in the clockwise direction for the left-hand side cylinder (see third row in Figure 7) and in the counterclockwise direction for the right-hand side cylinder. This behavior can be explained by the increasing role of convection in the mechanism of heat transfer with increasing Pr number. We remind here that we represent results only for left-hand side cylinder, which rotates in the clockwise direction.

Figure 8(a) shows the dependence of the average Nusselt number on the rotational speed for Re=20, g=14 and Pr=0.7, 1.0, 5.0, 10.0, and 20.0. The average Nusselt number decreases with increasing ω. In the case Pr=1 the average Nusselt number drops down by 10% and in the case of Pr=20 drops down by almost 50% when ω increases from 0 up to 2. This behavior can be explained by the fluid layers adjacent to the rotating cylinders due to no-slip requirement.

**Small Gap Spacing**

For small gap spacing, g=1, the effect of rotational speed is shown in Figures 8-12, for Re=10 and 20, Pr=0.7 and 20.0, and ω in the range 0 - 2.5. Representative plots of the streamlines and isotherms for Re=10 and 20, g=1, Pr=0.7, 10 and 20, and 0.5 < ω < 20 are shown in Figures 9-11. Due to the no-slip requirement, there are regions of closed streamlines near the cylinders for all values of ω. The size of these regions increases with increasing rotational speed. At some ω=ω* the two regions merge together. The main stream flows around the fluid region, which surrounds both cylinders and consists of two regions of closed streamlines. These regions acts as a buffer (blanket) isolating both cylinders from the main stream and causing a decrease in the overall heat transfer rate from the cylinder surfaces. The values of ω* depends on Reynolds number. As can be seen from Figure 9 for Re=10 and ω = 0.5 main stream can not pass through gap between cylinders, however for Re=20 and ω = 0.5 main stream is strong enough to go through the gap between two cylinders. A further increase in angular velocity ω > ω* is a reason of further increasing of regions size in both x and y directions. The stagnation points are now located on the y-axis, both upstream and downstream, as illustrated in Figure 9.

The details of the steady thermal fields are presented in Figures 10 and 11 (for the cases of Re=10 and 20, g=1, Pr=1, 10, 20, and ω=0.5,1.0,1.5, and 2.0) in the form of constant temperature contours. Isotherm patterns depend on the Prandtl number. For Re=20, Pr=20 and ω=0.5 there are two almost separate wake-type isotherm patterns downstream of each cylinders. One of the interesting features of the temperature field is topological similarity of isotherm patterns in cases of ω=1.5 and ω=2.0. For the cases of large Prandtl numbers Pr=40 and 20, the convection is dominant mechanism in heat transfer. The lines of constant

![Figure 8](image-url)

**Figure 8.** Average Nusselt number at Re = 20, (a)-g = 14 and (b)- g = 1, for different Prandtl numbers
temperature in the regions of closed streamlines follow to the streamlines contours. Heat exchange from the cylinders to the main stream occurred through the boundary of the fluid region, which enwraps both cylinders. Heat transfer inside this region is mostly due to conduction. By this reason, the average Nusselt number almost does not change for $\alpha > 1.5$ as can be seen from Figure 8(b). Comparison of isotherms in Figures 10 and 11 shows that an increase in Prandtl number makes wake-shape region of temperature field more narrow downstream behind the cylinder. It is interesting to point out that the rate of rotation does not significantly affect the size of the wake-shaped region.

Another interesting observation is the location of the saddle critical point in the temperature field. This critical point is located on the line connecting the cylinder centers in the cases of $Re = 10$, $\alpha = 0.5$, $Pr = 0.7$, and $Pr = 1$, $g = 1$, and $\alpha = 0.5, 1.0, 1.5, \text{ and } 2.0$.

Figure 9. Streamlines contours over two circular cylinders at $Re = 10$ left column and $Re = 20$ right column $Pr = 1$, $g = 1$, and $\alpha = 0.5, 1.0, 1.5, \text{ and } 2.0$
Re = 20, α = 1.0, Pr = 0.7. For larger values of α, α > 1.5 the saddle point is shifted down (in the negative direction of y-axis) by the fluid layers which rotate in step with cylinders due to no-slip condition.

Figure 12 shows effect of rotation on heat transfer in terms of the local Nusselt number distribution. For Pr = 20 and α = 2.5 the local Nusselt number is almost constant on the cylinder surfaces. As a result, the average Nusselt number drops as α increase. Dependence of the average Nü number on rotational speed is demonstrated in Figure 8(b). Figure 8(b) shows that for 0 < α ≤ 1.5, Nü decreases sufficiently fast, whereas for α ≥ 1.5 the rate of decrease of Nü is very low. This allows us to assume that heat transfer is almost insensitive to the speed of rotation if α ≥ 1.5.

Conclusions

A detailed numerical study of the two-dimensional heat-transfer problem and laminar flow around two rotating circular cylinders in side-
Numerical Simulation of Flow and Forced Convection Heat Transfer

by-side arrangement has been carried out. An efficient finite difference algorithm has been developed for the 2D Navier-Stokes equation in the cylindrical bi-polar coordinate system. Comparing the results for the case of large gap spacing with the available in the literature experimental and numerical data for the case of a single cylinder shows a good agreement.

For the problem of heat-transfer and flow around two rotating circular cylinders in side-by-side arrangement, the average Nusselt number decreases with increasing rotational speed.

For large Prandtl number \((Pr = 20)\) and high rotational speed \((\alpha \approx 2.0)\), the local Nusselt number distribution on the cylinder surfaces becomes almost uniform. This is due to the no-slip condition, the increasing rotational speed creates thick rotating layers of fluid (buffer layers) around the cylinders.

In case of small gap spacing between the cylinders, there are two flow regimes. In the first, the main stream passes through the gap between cylinders and in the second case, there is a fluid region which surrounds both cylinders and which consists of two sub-regions of closed

Figure 11. Temperature contours of fluid flow over two circular cylinders at \(Re = 20, \ g = 1,\ Pr = 0.7, 10, 20\) and \(\alpha = 0.5, 1.0, 1.5\) and 2.0
streamlines. It has been observed that the temperature contour pattern is similar to the streamline contour pattern. Owing to the rotation, the wake-shaped region of the isotherms is shifted in direction of the cylinder rotation. An increase in the Prandtl number increases the compactness of the isotherms toward the downstream.

References


Numerical Simulation of Flow and Forced Convection Heat Transfer

Physics of fluids, 15(9):2,486-2,498.