INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

U·M·I

University Microfilms International
A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
313/761-4700 800/521-0600

Order Number 1349000

**Numerical analysis on Strouhal frequencies in vortex shedding
over square cylinders with surface suction and blowing**

Ling, Lisa Mei, M.S.

Rice University, 1992

U·M·I

300 N. Zeeb Rd.
Ann Arbor, MI 48106

RICE UNIVERSITY

NUMERICAL ANALYSIS ON STROUHAL FREQUENCIES
IN VORTEX SHEDDING OVER SQUARE CYLINDERS
WITH SURFACE SUCTION AND BLOWING

by

LISA M. LING

A THESIS SUBMITTED
IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE
MASTER OF SCIENCE

APPROVED, THESIS COMMITTEE

Ruben D. Cohen, Associate Professor
Mechanical Engineering Department

Bala Ramaswamy, Assistant Professor
Mechanical Engineering Department

J. Ed Akin, Professor
Mechanical Engineering Department

Houston, Texas

May 1992

ABSTRACT

Numerical Analysis on Strouhal Frequencies in Vortex Shedding over Square Cylinders with Surface Suction and Blowing

by

Lisa M. Ling

The effects of normal surface suction and blowing on the Strouhal frequencies in vortex shedding over porous square cylinders was analyzed numerically. The general characteristics determined were (1) an initial increase followed by a decreasing behavior in the Strouhal frequency with increasing suction velocity and (2) a decrease in the Strouhal frequency with increasing blowing velocity. The numerical results were compared to existing analytical predictions, yielding fairly close agreement.

ACKNOWLEDGEMENTS

I would like to give special thanks to Dr. B. Ramaswamy for devoting much of his time working on this research project. I am grateful to him for all the guidance and help he so willingly gave me. I would like to thank Dr. R. Cohen for his guidance and assistance, and for providing insights in analyzing the data obtained in this research. I would also like to thank Dr. J. Akin for serving on my thesis committee. I am greatly appreciative of Mr. T. C. Jue for spending much of his precious time on this research. I would also like to thank the computer facility in the mechanical engineering department for providing me with their service.

I would like to thank my wonderful husband for all his love and support. I would also like to thank my parents and my sister for their constant encouragement.

Finally, I wish to express my appreciation to my supervisors, Dave Long and Dave Geller, at NASA for putting up with my eccentric work schedule during the past few months.

Table of Contents

1	Introduction	1
1.1	Strouhal Frequency and Reynolds Number Relationship Overview	2
1.2	Analytical Predictions of Effects of Surface Suction and Blowing on Strouhal Frequency Overview	6
2	Finite Element Program - Runge-Kutta Scheme	15
2.1	Theory	15
2.1.1	Basic Equations	16
2.1.2	Numerical Algorithm	19
2.2	Problem Definition	23
2.3	Bench Mark Cases for Flows over Solid Rectangular Cylinders	26
2.2.1	Square Cylinder	26
2.2.2	Rectangular Cylinder with Aspect Ratio of Two . . .	34
2.2.3	Heat Transfer for Square Cylinder	34
3	Numerical Analysis on Effects of Surface Suction and Blowing on Strouhal Frequency	46
3.1	Introduction	46
3.2	Results	47
3.2.1	Suction	50
3.2.2	Blowing	57
3.3	Comparison and Discussion of the Analytical and Numerical Solutions	65
4	Conclusions	69
5	Future Work	72
	Appendix A	75
	References	77

List of Figures

Figure 1:	St- Re_{∞} Relationship for Solid, Square Cylinder from Experimental Studies	4
Figure 2:	St- Re_{∞} Relationship for Solid, Rectangular Cylinder with Aspect Ratio of Two from Experimental Study	5
Figure 3:	St- Re_{∞} Relationship for Solid, Circular Cylinder	6
Figure 4:	Flow over a Cylinder	7
Figure 5:	Flow over Cylinder with Surface Suction and Injection . .	9
Figure 6:	Analytical $\log St - \log(Re_0^2/Re_{\infty})$ Relationship	14
Figure 7:	Finite Element Mesh for Square Cylinder	23
Figure 8:	Finite Element Mesh for Rectangular Cylinder with Aspect Ratio of Two	24
Figure 9:	Boundary Conditions Imposed on the Flow Field	25
Figure 10:	a) Velocity Vector and b) Pressure Contour for Partially Developed Flow over Solid, Square Cylinder at $t = 25$ s	27
Figure 11:	a) Velocity Vector and b) Pressure Contour for Fully Developed Flow over Solid, Square Cylinder at $t = 187.5$ s	28
Figure 12:	C_l , C_d , and Local Dimensionless Pressures for Flow over Solid, Square Cylinder	29
Figure 13:	a) Velocity Vector and b) Pressure Contour at $C_{l_{max}}$ for Flow over Solid, Square Cylinder	31
Figure 14:	a) Velocity Vector and b) Pressure Contour at $C_{l_{min}}$ for Flow over Solid, Square Cylinder	32
Figure 15:	a) Velocity Vector and b) Pressure Contour at C_{l_0} for Flow over Solid, Square Cylinder	33
Figure 16:	a) Velocity Vector and b) Pressure Contour for Fully Developed Flow over Solid, Rectangular Cylinder with Aspect Ratio of Two at $t = 187.5$ s	35
Figure 17:	Boundary Conditions Imposed on the Flow Field with Heat Transfer	38
Figure 18:	a) Velocity Vector, b) Isotherms, and c) Pressure Contour for $Gr = 0$ and $Re = 100$	39
Figure 19:	a) Velocity Vector, b) Isotherms, and c) Pressure Contour for $Gr = 4 \times 10^4$ and $Re = 100$	40
Figure 20:	a) Velocity Vector, b) Isotherms, and c) Pressure Contour for $Gr = 16 \times 10^4$ and $Re = 100$	41

Figure 21: a) Velocity Vector, b) Isotherms, and c) Pressure Contour for $Gr = 0$ and $Re = 500$	42
Figure 22: a) Velocity Vector, b) Isotherms, and c) Pressure Contour for $Gr = 2 \times 10^6$ and $Re = 500$	43
Figure 23: Nusselt Number and Skin Friction Coefficient Plots at $Re = 100$ for Various Gr Values	45
Figure 24: Boundary Conditions Imposed on the Flow Field for Cylinder with Normal Surface Suction and Injection . .	47
Figure 25: Strouhal Frequencies Versus Suction ($-v_o$) and Blowing ($+v_o$) Velocities	49
Figure 26: Numerical $\log St - \log(Re_o^2/Re_\infty)$ Relationship	49
Figure 27: a) Velocity Vector and b) Pressure Contour for Partially Developed Flow with 0.1 Suction Velocity at $t = 37.5$ s	51
Figure 28: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.1 Suction Velocity at $t = 187.5$ s	52
Figure 29: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.4 Suction Velocity at $t = 187.5$ s	53
Figure 30: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.45 Suction Velocity at $t = 187.5$ s	54
Figure 31: C_l , C_d , and Local Pressures for Flow with 0.4 Suction Velocity	56
Figure 32: a) Velocity Vector and b) Pressure Contour for Partially Developed Flow with 0.1 Blowing Velocity at $t = 75$ s	58
Figure 33: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.1 Blowing Velocity at $t = 187.5$ s	60
Figure 34: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.15 Blowing Velocity at $t = 247.5$ s	61
Figure 35: a) Velocity Vector and b) Pressure Contour for Fully Developed Flow with 0.2 Suction Velocity at $t = 187.5$ s	62
Figure 36: C_l , C_d , and Local Pressures for Flow with 0.07071 Blowing Velocity at $Re_\infty = 500$	64
Figure A-1: The Outflow Boundary Condition	75

List of Tables

Table 1: Comparison of Numerical and Experimental Results for Flow over Solid, Square Cylinder	30
Table 2: Results of Effects of Normal Surface Suction and Blowing	48
Table 3: Effects of Varying Re_∞ for Suction at $Re_0^2/Re_\infty = 2.5$	57
Table 4: Effects of Varying Re_∞ for Suction at $Re_0^2/Re_\infty = 22.5$. . .	57
Table 5: Effects of Varying Re_∞ for Blowing at $Re_0^2/Re_\infty = 2.5$	65
Table 6: Effects of Varying Re_∞ for Blowing at $Re_0^2/Re_\infty = 5.625$.	65

List of Symbols

Symbol	Description
C	skin friction coefficient
C_d	drag coefficient
C_l	lift coefficient
C_{l0}	zero lift coefficient
$C_{l_{\max}}$	maximum lift coefficient
$C_{l_{\min}}$	minimum lift coefficient
d	diameter of cylinder or height of rectangle
f	frequency
Gr	Grashof number
L	characteristic length
Nu	Nusselt number
p	local pressure
Re	Reynolds number
S	boundary of fluid domain
St	Strouhal frequency
T	temperature
t	time
U	freestream velocity
u	tangential velocity for polar coordinates; x-direction velocity for cartesian coordinates
v	normal velocity for polar coordinates; y-direction velocity for cartesian coordinates
δ	thickness of boundary layer
δ	Kronecker delta
Φ	finite element interpolation function
μ	constant shear viscous coefficient
ν	kinematic viscosity
ρ	fluid density
τ	shedding period
Ω	fluid domain

Subscripts

∞	freestream
0	with suction or injection

Superscripts

*	dimensionless parameter
0	at time equal to zero

1 Introduction

The studies of vortex shedding over various structural shapes in the past have contributed to better understanding of areas such as aerodynamics and aeroelastic characteristics. This in turn will allow improved engineering designs much needed for better energy conservation or higher performance. For example, the increasing scarcity of natural resources have placed a need on reducing the aerodynamic drag of automobiles to lower fuel consumption. A minimal convective heat loss for tall buildings is favored for the conservation of energy. Tall buildings and suspension bridges must be designed such that large-amplitude oscillations will not occur upon encounter of high winds. Better understanding of flows around airfoils at high angle-of-attack is essential in improving the designs of commercial and military aircrafts. Other practical applications involve the designs of heat exchangers, chimneys, flowmeters for measuring flow rates inside closed conduits, and many others. [1,2]

Abundant research had been done in the analysis of Strouhal frequencies (St) in vortex shedding over impermeable bodies. The analytical, numerical, and experimental results can be easily obtained in the publications of such work. However, the Strouhal frequencies in vortex shedding over porous surfaces involving suction or blowing has not been widely researched. The only data found dealing directly with this problem were from the analytical study of flows over porous cylinders by Cohen [3]. In this study, Cohen derived a model to predict the general effect of normal surface

suction and blowing on the St-Reynolds number (Re) relationship. This model is based on an assumption that the viscous diffusion length is directly proportional to the boundary layer thickness.

The interest of this investigation is to numerically analyze the Strouhal frequencies in vortex shedding over porous square cylinders with various suction and blowing velocities. This analysis will provide the general trend of the St as affected by normal surface suction or blowing for a square cylinder. A comparison of the analytical and numerical solutions will then assess the validity of the theoretical predictions and provide insights to the suction and blowing problem.

The finite element program used for this analysis is a fourth-order Runge-Kutta time integration scheme based on the Chorin's splitting scheme [4]. Several bench mark cases of flows over solid rectangular cylinders were examined to verify the numerical program.

1.1 Strouhal Frequency and Reynolds Number Relationship Overview

Experimental studies on the Strouhal frequencies of rectangular cylinders with various width-to-height ratio, or aspect ratio, was conducted separately by Okajima in a wind tunnel and a water tank [5], and by Davis and Moore in a wind tunnel [1]. Numerical analyses of flows over rectangular cylinders were also carried out by

Davis and Moore, Okajima *et al.*, Shimura and Kawahara, and Yoshida and Nomura [1,2,6,7].

A detailed discussion of the St characteristics for flow over a solid, circular cylinder can be obtained from Rothberg [8]. Other numerical studies of flows over circular cylinders were conducted by Briscolini and Santangelo, Tezduyar and Shih, Tanahashi and Okanaga, and Tabata and Fujima [9-12].

The St characteristics for a square cylinder, a rectangular cylinder with an aspect ratio of two, and a circular cylinder is now discussed. Special interest is paid to the St characteristics at $Re_\infty = 250$ for comparison purposes as discussed in Sections 2.2.1 and 2.2.2.

(i) *solid square cylinder* - Figure 1 shows the St versus Re_∞ relationship for a square cylinder obtained from the two, separate experimental studies by Okajima [5] and Davis and Moore [1]. It is seen that there exists noticeable experimental uncertainties, especially at $Re_\infty \approx 400$. The experimental St at $Re_\infty = 250$ ranges from approximately 0.140 to 0.159. The computational St obtained by Davis and Moore at $Re_\infty = 250$ ranged from 0.160 to 0.169 and is dependent on the grid size [1]. The computational St are slightly higher than but agreeable with the experimental results.

The $St-Re_\infty$ curve is continuous and remains fairly constant for a wide range of Re_∞ between 10^2 and 2×10^4 . The St remains at a value of 0.13 at high Re_∞ between 10^4 and 2×10^4 .

Figure 1. $St-Re_{\infty}$ relationship for solid, square cylinder from experimental studies [1,5].

(ii) *solid rectangular cylinder with aspect ratio of two* - The $St-Re_{\infty}$ relationship for a rectangular cylinder with an aspect ratio of two is obtained from Okajima's experimental study [5] as shown in Figure 2. At $Re_{\infty} < 500$, the St increases as Re_{∞} increases, reaching a value of 0.18. At $Re_{\infty} \approx 500$, a discontinuity in the curve occurs where the St decreases sharply and suddenly. As Re_{∞} continues to increase, the St increases gradually but remains small. At $Re_{\infty} > 5 \times 10^3$, the St remains fairly constant at 0.08 to 0.09. The abrupt decrease in the St at $Re_{\infty} \approx 500$ is caused by a widening of the wake due to a sudden detachment of the separated flow from the surfaces of the rectangular cylinders.

The experimental St at $Re_\infty = 250$ ranges from approximately 0.142 to 0.161. The computational St obtained by Okajima *et al.* [2] at $Re_\infty = 250$ was 0.11. This is slightly lower than but agreeable with the experimental results. Okajima *et al.* also found that, at a low Re_∞ such as 250, the Strouhal frequency stays fairly constant for rectangular cylinders with various aspect ratios.

Figure 2. $St-Re_\infty$ relationship for solid, rectangular cylinder with aspect ratio of two from experimental study [5].

(iii) *solid circular cylinder* - Figure 3 shows the $St-Re_\infty$ relationship for flow over a solid, circular cylinder. It is observed that, in the region of Re_∞ between 3×10^5 and 3.5×10^6 , vortex shedding is not present. Else, the St curve shows only slight

variations with respect to changes in the Re_∞ . At Re_∞ between 40 and 150, laminar boundary layer and wake exists. The wake is found to be turbulent while the boundary layer remains laminar at Re_∞ between 150 and 3×10^5 . At Re_∞ greater than 3.5×10^6 , both the wake and the boundary layer are turbulent.

Figure 3. $St-Re_\infty$ relationship for solid, circular cylinder [3]

1.2 Analytical Predictions of Effects of Surface Suction and Blowing on Strouhal Frequency Overview

A simple model was derived by Cohen to predict the general characteristics of vortex shedding over cylinders with surface suction and injection [3]. A model was first derived to predict the characteristics of St for flow over a solid, circular cylinder using

scaling arguments. This model was then extended to account for the presence of blowing and suction. The basis of this model is an assumption that the viscous diffusion length is directly proportional to the boundary layer thickness. A brief summary of the derivation is now given.

For the system shown in Figure 4, the well-known unsteady boundary layer equation derived by simplifying the incompressible Navier-Stokes equation with the assumption of a very thin boundary layer (i.e. $\delta/R \ll 1$) is

$$\frac{\partial u}{\partial t} + \frac{u}{r} \frac{\partial u}{\partial \theta} + v \frac{\partial u}{\partial r} = \frac{U(\theta)}{r} \frac{\partial U(\theta)}{\partial \theta} + \nu \frac{\partial^2 u}{\partial r^2} \quad (1)$$

Figure 4. Flow over a cylinder [3].

Without showing the detailed derivations, the following was obtained for flow over a solid, circular cylinder using order of magnitude scaling of equation (1):

$$\delta(t) \sim \sqrt{\nu t} \quad \text{for } 0 \leq t \leq \tau \quad (2)$$

$$\delta(t) \sim \sqrt{\frac{\nu R}{U}} \quad \text{when } t \sim \tau \quad (3)$$

where τ is the shedding period. As t approaches τ , equations (2) and (3) will approach one another. This yields

$$\tau \sim \frac{R}{U}$$

Written in terms of frequency (f), the cylinder diameter (d), and freestream velocity (U_∞), the above becomes

$$\frac{1}{f} \sim \frac{d}{U_\infty} \quad (4)$$

The Strouhal frequency is defined as fd/U_∞ . Equation (4) will thus lead to the conclusion that St is approximately constant. This is consistent with the $St-Re_\infty$ relationship obtained experimentally and numerically for various body shapes such as circular and square cylinders.

This model is now extended to derive a model for the problem of flow over cylinders with surface suction and blowing. For a cylinder with surface suction or injection as shown in Figure 5, the velocity boundary condition on the cylinder surface is

$$\bar{v}_{\text{surface}} = v_o \hat{i}_r$$

where v_o is positive for injection and negative for suction. This is applied to equation (1) to account for suction and blowing. Using scaling arguments, the boundary layer equation (1) at steady state becomes

$$k \frac{U_\infty^2}{d} + v_o \frac{U_\infty}{\delta} \sim \frac{U_\infty^2}{d} + v \frac{U_\infty}{\delta^2} \quad (5)$$

The dimensionless constant, k , is appended to the convective acceleration term to differentiate it from the pressure gradient term $(U(\theta)/r \times \partial U(\theta)/\partial \theta)$. Using equation (5), the characteristics of the shedding period are now derived separately for the suction and blowing problems.

Figure 5. Flow over cylinder with surface suction and injection [3].

(i) *suction* - For the normal surface suction problem, the boundary layer structure can be divided into two different types. Near the leading edge, the boundary layer is of the Blasius type. Here, the effect of suction is not dominant. This occurs when

$$v_o \frac{U_\infty}{\delta} \ll \frac{U_\infty^2}{d}$$

Applying to equation (5), the following is obtained:

$$\frac{\delta}{d} \sim \sqrt{\frac{v}{U_\infty d}} \quad (6)$$

Further downstream, the flow begins to feel the effect of suction. Therefore,

$$v_o \frac{U_\infty}{\delta} \gg \frac{U_\infty^2}{d} \quad (7)$$

Applying to equation (5) yields

$$\delta \sim \frac{v}{v_o} \quad (8)$$

This shows that the boundary layer thickness approaches a constant value at the downstream portion of the body where strong surface suction exists. Substituting equation (8) into (7), the following is obtained:

$$\delta \sim \frac{v}{v_o} \quad \text{when} \quad \frac{v_o^2 d}{v U_\infty} \gg 1$$

Let $Re_o = v_o d / \nu$, then the above can be written as

$$\frac{\delta}{d} \sim \frac{1}{Re_o} \quad \text{when} \quad \frac{Re_o^2}{Re_\infty} \gg 1 \quad (9)$$

Using equation (2) for the shedding model where $\delta = \sqrt{\nu \tau}$ at time τ , the following is obtained:

$$\frac{\delta}{d} \sim \frac{\sqrt{\nu \tau}}{d} \quad (10)$$

τ can be obtained by setting equations (9) and (10) equal.

$$\tau \sim \frac{d^2}{\nu Re_o^2} \quad \text{when} \quad \frac{Re_o^2}{Re_\infty} \gg 1$$

Written in terms of $St (= d / \tau U_\infty)$, the above becomes

$$St_{\text{suction}} \sim \frac{Re_o^2}{Re_\infty} \quad \text{when} \quad \frac{Re_o^2}{Re_\infty} \gg 1 \quad (11)$$

Equation (11) suggests that an increasing rate of suction will increase the Strouhal frequency.

(ii) *blowing* - The boundary layer for the normal surface blowing problem can also be separated into two types. Once again, the boundary layer in the leading edge region is of the Blasius type as expressed in equation (6). As the boundary layer grows linearly in the downstream direction, the effect of normal blowing becomes dominant such that

$$v_o \frac{U_\infty}{\delta} \gg \frac{\nu U_\infty}{\delta^2} \quad (12)$$

Substituting the above into Equation (5) yields

$$\frac{U_{\infty}^2}{d} \sim \frac{v_0 U_{\infty}}{\delta}$$

Using the above expression, equation (12) becomes

$$\frac{Re_0^2}{Re_{\infty}} \gg 1$$

Therefore, for normal surface blowing, the following is obtained from the two equations above.

$$\frac{\delta}{d} \sim \frac{v_0}{U_{\infty}} \quad \text{when} \quad \frac{Re_0^2}{Re_{\infty}} \gg 1$$

Again, using equation (2) for the shedding model and solving for τ , the above becomes

$$\tau \sim \frac{d^2 v_0^2}{v U_{\infty}^2} \quad \text{when} \quad \frac{Re_0^2}{Re_{\infty}} \gg 1$$

Writing the above in terms of Strouhal frequency, the following is obtained for the normal surface blowing problem:

$$St_{\text{blowing}} \sim \frac{Re_{\infty}}{Re_0^2} \quad \text{when} \quad \frac{Re_0^2}{Re_{\infty}} \gg 1$$

This equation states that increasing normal surface blowing will decrease the Strouhal frequency.

The predicted effects of normal surface suction or blowing are summarized as follows:

$St_{\text{suction}} \sim \frac{Re_o^2}{Re_\infty}$	when $\frac{Re_o^2}{Re_\infty} \gg 1$
$St_{\text{blowing}} \sim \frac{Re_\infty}{Re_o^2}$	when $\frac{Re_o^2}{Re_\infty} \gg 1$
$St_{\text{suction\&blowing}} \sim \text{constant}$	when $\frac{Re_o^2}{Re_\infty} \ll 1$

These predicted results can be shown graphically as in Figure 6. This figure indicates that there exists a critical Re_∞ , defined as

$$Re_{\infty \text{critical}} = Re_o^2$$

such that, if Re_∞ were much less than $Re_{\infty \text{critical}}$, then Re_o must be accounted for. Else if Re_∞ were much greater than $Re_{\infty \text{critical}}$, then the suction or blowing effects become negligible, and the system can be treated as that for flow over an impermeable body. At the region where Re_∞ is approximately $Re_{\infty \text{critical}}$ ($Re_o^2/Re_{\infty \text{critical}} \approx 1$), denoted by the dashed lines in Figure 6, the surface suction and blowing may or may not have noticeable effects on the shedding frequencies. If the surface suction and blowing do have noticeable effects on the shedding frequencies in the region where $Re_\infty \approx Re_{\infty \text{critical}}$, then the resulting change in the St should be less than $\pm 10\%$ or $\pm 20\%$. For systems operating at Re_∞ below the critical value, the predicted results also indicate that, on a log-log plot shown in Figure 6, the slopes of all the $St-Re_o^2/Re_\infty$ curves will asymptotically approach

+1 for suction and -1 for blowing, regardless of the suction and blowing velocities.

Figure 6. Analytical $\log St - \log(Re_o^2/Re_\infty)$ relationship at different rates of normal suction and injection [3]

It was also mentioned by Cohen that there may exist an upper bound for the normal surface suction problem. As the suction velocity increases, the onset of flow separation may be delayed, and vortex shedding may become totally suppressed in the suction dominated flow.

2 Finite Element Program - Runge-Kutta Scheme

The finite element program used for this study was developed by Ramaswamy and Jue [13]. In computing the solutions for flows past cylinders, a fixed perturbation was not imposed, and it is the numerical noise which leads the symmetric flow into a vortex shedding state. The numerical noise is composed of discretization and roundoff errors. [14]

To verify the numerical program, several bench mark cases for flows over solid, rectangular cylinders were examined, and the solutions were then compared with existing numerical and experimental data summarized in Section 1.1.

2.1 Theory

The finite element program used in this analysis is a semi-implicit finite element method developed for viscous incompressible flows. It is based on a set of finite element equations of the primitive variable formulation and a time-splitting procedure. The primitive variable formulation involves the separation of velocity and pressure. An intermediate step is introduced to calculate a velocity field which satisfies only the momentum equation without the pressure term. To attain velocity with zero divergence, the intermediate velocity and pressure are subsequently corrected by the pressure gradient and by the divergence of the intermediate

velocity, respectively. The temperature distribution can also be obtained using the energy equation. This finite element program was designed specifically for the efficient calculations of highly transient incompressible flows.

2.1.1 Basic Equations

For a flow describable by the two-dimensional plane form of the Navier-Stokes equations, the fluid motion is assumed to be laminar, unsteady, isothermal, and incompressible. The mathematical expressions of the fluid motion are described herein using indexical notation in the spatial rectangular coordinate system x_i ($i = 1, 2$), and the summation convention is used for repeated indices. Let Ω be a fluid domain bounded by a piecewise smooth boundary S . The Navier-Stokes and continuity equations for an incompressible viscous Newtonian fluid in Eulerian representation are

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j u_{i,j} - \sigma_{ij,j} - \rho f_i = 0, \text{ in } \Omega \quad (13)$$

$$u_{i,i} = 0, \text{ in } \Omega \quad (14)$$

where

$$\sigma_{ij} = -p \delta_{ij} + \mu (u_{i,j} + u_{j,i})$$

u_i ($i = 1, 2$) are the x_i -components of the fluid velocity. ρf_i are the body forces. ρ is the constant density. δ_{ij} is the Kronecker delta. μ is the constant shear viscous coefficient.

All the parameters involved are nondimensionalized using the characteristic length, L , a characteristic velocity, u , and the fluid density, ρ .

$$u_i^* = \frac{u_i}{u}$$

$$x_i^* = \frac{x_i}{L}$$

$$p^* = \frac{Lp}{\rho v u}$$

$$f_i^* = \left(\frac{vu}{L^2} \right)^{-1} f_i$$

$$v^* = \frac{\mu}{\rho u L} = \frac{1}{Re}$$

$$t^* = \frac{v}{L^2} t$$

The equation of motion and the continuity equation are then nondimensionalized using the above expressions. Dropping the asterisks for simplicity, equations (13) and (14) become

$$\frac{\partial u_i}{\partial t} + u_j u_{i,j} - \frac{1}{Re} (u_{i,j} + u_{j,i})_{,j} + p_{,i} = f_i, \text{ in } \Omega$$

$$u_{i,i} = 0, \text{ in } \Omega$$

To complete the formulation of the basic equations, a set of boundary and initial conditions are required. The continuum boundary conditions are specified for velocity and stress. At a

boundary node, the proper boundary value is enforced by the constraint condition

$$u_i = \hat{u}_i, \text{ on } S_1$$

On boundary S_2 , the surface force is specified in terms of the stress components.

$$\sigma_{ij} n_j = \hat{h}_i, \text{ on } S_2$$

where n_j is the direction cosine outward-normal on the boundary with respect to the x_j -axis. The superposed h denotes the function given on the boundary. The subsets S_1 and S_2 of S satisfy the following conditions:

$$\overline{S_1 \cup S_2} = S$$

$$S_1 \cap S_2 = \phi$$

where the superposed bar represents set closure and ϕ denotes the empty set.

The initial conditions are specified for velocity.

$$u_i(x_i, 0) = u_i^0(x_i)$$

With the initial velocity $u_i^0(x_i)$ and satisfying the incompressibility condition, the following is obtained:

$$u_{i,i}(0) = 0$$

2.1.2 Numerical Algorithm

The method applied to solve the Navier-Stokes equation is the fourth-order Runge-Kutta time integration scheme based on the Chorin's [4] splitting technique. According to this scheme, the equation of motion in tensor form are

$$\frac{\partial u_i}{\partial t} + \frac{\partial p}{\partial x_i} = F_i$$

where

$$F_i = -u_j \frac{\partial u_i}{\partial x_j} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_j^2}$$

At each of the four stages, the determination of the velocity is split into two steps. The first step is to determine the predicted velocity by time integration of the momentum equations without the pressure term. The second step is to develop the pressure and the corrected velocity fields that satisfy the continuity equation. Let u_i^n and p^n denote the velocity and pressure solution of the discretized equations at time level t^n , where $t^n = t^{n-1} + \Delta t$ and $t^0 = 0$. From u_i^n and the boundary conditions, the fields u_i^{n+1} , p^{n+1} at time t^{n+1} are calculated as follows:

stage 1:

$$\bar{u}_i^{-1} = u_i^n + \frac{\Delta t}{2} F_i(u_i^n)$$

$$u_i^1 = \bar{u}_i^{-1} - \frac{\Delta t}{2} p_{,i}$$

$$u_{i,i}^1 = 0$$

stage 2:

$$\bar{u}_i^{-2} = u_i^n + \frac{\Delta t}{2} F_i(u_i^1)$$

$$u_i^2 = \bar{u}_i^{-2} - \frac{\Delta t}{2} p_{,i}$$

$$u_{i,i}^2 = 0$$

stage 3:

$$\bar{u}_i^{-3} = u_i^n + \frac{\Delta t}{2} F_i(u_i^2)$$

$$u_i^3 = \bar{u}_i^{-3} - \frac{\Delta t}{2} p_{,i}$$

$$u_{i,i}^3 = 0$$

stage 4:

$$\bar{u}_i^{-n+1} = u_i^n + \Delta t \left(F_i \frac{u_i^n}{6} + F_i \frac{u_i^1}{3} + F_i \frac{u_i^2}{3} + F_i \frac{u_i^3}{6} \right)$$

$$u_i^{n+1} = \bar{u}_i^{-n+1} - \Delta t (p_{,i}^{n+1})$$

$$u_{i,i}^{n+1} = 0$$

In the above, the explanation of the algorithm was given in a continuum space. In the finite element method, the variable u_i and p are discretized by the following finite element approximations:

$$u_i = \Phi_\alpha u_{\alpha i}$$

$$p = \Phi_\alpha P_\alpha$$

where Φ_α is the finite element interpolation function, $u_{\alpha i}$ denotes the nodal value of velocity at the α -th node of the finite element in the i th direction, and P_α indicates the nodal value of pressure at the α -th node.

In the present study, the four node isoparametric element is used. Using the conventional Galerkin-Bubnov finite element method, the four stage procedure explained previously is discretized in space and reduced to the following set of equations:

stage 1:

$$M_{\alpha\beta}^D \bar{u}_{\beta i}^1 = M_{\alpha\beta} u_{\beta i}^n - \frac{\Delta t}{2} (K_{\alpha\beta rj} u_{\beta j}^n u_{ri}^n + s_{\alpha i\beta j} u_{\beta j}^n - \omega_{\alpha i}^n)$$

$$M_{\alpha\beta}^D u_{\beta i}^1 = M_{\alpha\beta} \bar{u}_{\beta i}^1 - \frac{\Delta t}{2} H_{\alpha i\beta} P_\beta^n$$

$$H_{\alpha\beta}^T u_{\beta i}^1 = 0$$

stage 2:

$$M_{\alpha\beta}^D \bar{u}_{\beta i}^2 = M_{\alpha\beta} u_{\beta i}^n - \frac{\Delta t}{2} (K_{\alpha\beta rj} u_{\beta j}^1 u_{ri}^1 + s_{\alpha i\beta j} u_{\beta j}^1 - \omega_{\alpha i}^1)$$

$$M_{\alpha\beta}^D u_{\beta i}^2 = M_{\alpha\beta} \bar{u}_{\beta i}^2 - \frac{\Delta t}{2} H_{\alpha i\beta} P_\beta^n$$

$$H_{\alpha\beta}^T u_{\beta i}^2 = 0$$

stage 3:

$$\begin{aligned} M_{\alpha\beta}^D \bar{u}_{\beta i}^3 &= M_{\alpha\beta} u_{\beta i}^n - \frac{\Delta t}{2} (K_{\alpha\beta rj} u_{\beta j}^2 u_{ri}^2 + s_{\alpha i\beta j} u_{\beta j}^2 - \omega_{\alpha i}^2) \\ M_{\alpha\beta}^D u_{\beta i}^3 &= M_{\alpha\beta} \bar{u}_{\beta i}^3 - \frac{\Delta t}{2} H_{\alpha i\beta} p_{\beta}^n \\ H_{\alpha\beta}^T u_{\beta i}^3 &= 0 \end{aligned}$$

stage 4:

$$\begin{aligned} M_{\alpha\beta}^D \bar{u}_{\beta i}^{n+1} &= M_{\alpha\beta} u_{\beta i}^n - \Delta t \left[\frac{1}{6} (K_{\alpha\beta rj} u_{\beta j}^n u_{ri}^n + s_{\alpha i\beta j} u_{\beta j}^n + \omega_{\alpha i}^n) \right. \\ &\quad + \frac{1}{3} (K_{\alpha\beta rj} u_{\beta j}^1 u_{ri}^1 + s_{\alpha i\beta j} u_{\beta j}^1 + \omega_{\alpha i}^1) \\ &\quad + \frac{1}{3} (K_{\alpha\beta rj} u_{\beta j}^2 u_{ri}^2 + s_{\alpha i\beta j} u_{\beta j}^2 + \omega_{\alpha i}^2) \\ &\quad \left. + \frac{1}{3} (K_{\alpha\beta rj} u_{\beta j}^3 u_{ri}^3 + s_{\alpha i\beta j} u_{\beta j}^3 + \omega_{\alpha i}^3) \right] \\ M_{\alpha\beta}^D u_{\beta i}^{n+1} &= M_{\alpha\beta} \bar{u}_{\beta i}^n - \Delta t H_{\alpha i\beta} p_{\beta}^{n+1} \\ H_{\alpha\beta}^T u_{\beta i}^{n+1} &= 0 \end{aligned}$$

where $M_{\alpha\beta}$, $K_{\alpha\beta rj}$, $s_{\alpha i\beta j}$ and $H_{\alpha i\beta}$ are the mass, convection, diffusion, and pressure gradient matrices, respectively. $\omega_{\alpha i}$ represents the surface integral terms. $M_{\alpha\beta}^D$ is the lumped mass matrix obtained simply by summing across each row of the consistent mass matrix $M_{\alpha\beta}$ and placing the results in the diagonal of $M_{\alpha\beta}^D$.

The advantage of using this scheme is that the timestep used can be much larger than that of the first-order projection scheme. For flows which may involve high Reynolds numbers, it is beneficial to adapt this approach since a large timestep is allowed.

2.2 Problem Definition

Figure 7 displays the finite element mesh for flow over a rectangular cylinder with an aspect ratio of one (a square cylinder). This mesh system yields 3010 nodes and 2880 elements. The finite element mesh for a rectangular cylinder with an aspect ratio of two is shown in Figure 8. This mesh system generates 3115 nodes and 2976 elements.

Figure 7. Finite element mesh for square cylinder

Figure 8. Finite element mesh for rectangular cylinder with aspect ratio of two

The boundary conditions imposed on the open boundaries of the flow field and on the cylinder surface are described in Figure 9. Note that the zero-velocity boundary conditions on the cylinder surface is for flow over a solid cylinder. For flow over a porous cylinder involving normal surface suction or blowing, the boundary conditions on the cylinder surface will not necessarily be zero. Referring to Figure 9, the length of the cylinder for the rectangular cylinder with an aspect ratio of two would be $2H$ instead of H , and the length of the wake region will be $8.5H$ instead of $9.5H$.

Figure 9. Boundary conditions imposed on the flow field

Note that the velocity vector and pressure contour plots given in the following sections do not display the whole flow field domain defined by the mesh system. The partial domain plotted are from $1H$ to $14.6H$ in the x -direction (instead of 0 to $15H$ for the full domain) and $3.7H$ to $11.3H$ in the y -direction (instead of 0 to $15H$ for the full domain).

Variations of the lift coefficient (C_l), drag coefficient (C_d), and the non-dimensional surface pressure at the center of each side of the cylinder (see Figure 12) are calculated. The C_l is the difference between the pressures on sides S_2 and S_4 , and C_d is correlated with the variation of the pressure on the back side S_3 .

The non-dimensional freestream velocity and Reynolds number chosen for the cases investigated in this study are 1 and 250 , respectively, unless otherwise stated. The timestep is 0.0075 sec.

2.3 Bench Mark Cases for Flows over Solid Rectangular Cylinders

Several bench mark cases were examined, and the solutions were compared to existing numerical and experimental data given in Section 1.1 for verification of the numerical program used for this study. The bench mark cases involve flow over solid, rectangular cylinders with aspect ratios of one and two. The rectangular cylinder with an aspect ratio of one is a square cylinder. Heat transfer analysis for flow over a square cylinder in a horizontal channel is also included as a bench mark case.

2.2.1 Square Cylinder

The first bench mark case examined was vortex shedding over a solid, square cylinder. Figure 10 shows the velocity vector and pressure contour plots for the flow field at 25 s. The flow had not fully developed yet, and the flow appears symmetric, with twin vortices forming behind the cylinder. As time progresses, the flow becomes asymmetric, and periodic vortex shedding over the cylinder is attained. Figure 11 shows the velocity vector and pressure contour plots at 187.5 s for a fully developed flow where the period of vortex shedding is constant.

The lift coefficient, drag coefficient, and the non-dimensional pressure values at the center of each cylinder side are plotted in Figure 12 for t ranging from 162 s to 178 s. Note that the oscillation

a)

b)

Figure 10. a) Velocity vector and b) pressure contour for partially developed flow over solid, square cylinder at $t = 25$ s

a)

b)

Figure 11. a) Velocity vector and b) pressure contour for fully developed flow over solid, square cylinder at $t = 187.5$ s

Figure 12. C_l , C_d , and local pressures for flow over solid, square cylinder

frequency of C_d is twice that of C_l . At $C_{l_{\max}}$ or $C_{l_{\min}}$, Figures 13 and 14 show that the flow separates at the leading edge, and the vortex is detached from the cylinder. At C_{l_0} , Figure 15 shows that the growing vortex is ready to depart from the cylinder.

The resulting shedding period is 5.655 s, and the corresponding Strouhal frequency is 0.177. This is agreeable with the experimental results obtained by Okajima and Davis & Moore ($St = 0.140$ to 0.159), as shown in Figure 1, and the numerical St obtained by Davis and Moore ($St = 0.160$ to 0.169). This is summarized in Table 1, which also includes other comparisons of numerical results obtained in this analysis with the computational results of Davis and Moore [1].

Table 1. Comparison of numerical and experimental results for flow over solid, square cylinder

Source	St	Period	C_l	C_d
Semi-implicit	0.177	5.655	-0.51 to 0.51	1.57 to 1.72
Others	0.140 # to 0.169	5.917 * to 7.143	-0.50 * to 0.50	1.73 * to 1.81

- experimental results from Figure 1 and computational results from Davis and Moore [1]

* - computational results from Davis and Moore

a)

b)

Figure 13. a) Velocity vector and b) pressure contour
at $C_{l_{\max}}$ for flow over solid, square cylinder

a)

b)

Figure 14. a) Velocity vector and b) pressure contour at C_{lmin} for flow over solid, square cylinder

a)

b)

Figure 15. a) Velocity vector and b) pressure contour
at C_{l0} for flow over solid, square cylinder

2.2.2 Rectangular Cylinder with Aspect Ratio of Two

The next bench mark case investigated was flow over a solid, rectangular cylinder with an aspect ratio of two. Figure 16 shows the velocity vector and pressure contour plots for 187.5 s. The resulting shedding period is 5.670 s, and the corresponding Strouhal frequency is 0.176. These are almost equivalent to those obtained for the square cylinder and is supportive of the conclusion made by Okajima *et al.* that the St remains almost constant at low R_∞ regardless of the aspect ratio of the rectangular cylinder. The St obtained is consistent with Okajima's experimental St ($= 0.142$ to 0.161), as shown in Figure 2, and Okajima *et al.*'s computational St ($= 0.11$). The C_l , C_d , and local dimensionless pressure plots are similar to those of the square cylinder (Figure 12), as are the velocity vector and pressure contour plots at $C_{l_{max}}$, $C_{l_{min}}$, and C_{l_0} (Figures 13-15).

2.2.3 Heat Transfer for Square Cylinder

The third bench mark case involved mixed convection heat transfer in a horizontal channel with a built-in square cylinder. Investigative work on the mixed convection heat transfer problem have been carried out by researchers such as Biswas *et al.* [15].

With a relatively low flow velocity and a large temperature difference between the body surface and the fluid, the flow and heat transfer characteristics become greatly affected by the thermal

a)

b)

Figure 16. a) Velocity vector and b) pressure contour for fully developed flow over solid, rectangular cylinder with aspect ratio of two at $t = 187.5$ s

buoyancy forces. The thermal buoyancy forces will induce velocity normal to the forced flow direction and, thus, distorting the wake of the cylinder. The strength of the thermal buoyancy force is expressed in terms of the Grashof number (Gr). In this section, numerical data for the heat transfer (in terms of the local Nusselt number, Nu_x) and flow losses (in terms of the local skin friction coefficient, C_x) with respect to the Reynolds number and Grashof number are generated and compared to the numerical results of Biswas *et al.*

The governing equations for the mixed convection heat transfer problem are

$$\frac{\partial u_i}{\partial t} + u_j u_{i,j} = -P_{,j} \delta_{ij} + \frac{1}{Re} (u_{i,j} + u_{j,i})_{,j} + \frac{GrT}{Re^2} f_i$$

$$\frac{\partial T}{\partial t} + u_j T_{,j} = \frac{1}{RePr} T_{,jj}$$

$$u_{i,i} = 0$$

where T is the temperature, and Pr is the Prandtl number.

For the mathematical formulation, the dimensionless scales used include

$$x_i = \frac{x_i^*}{H}$$

$$u_i = \frac{u_i^*}{U_{av}}$$

$$p = \frac{p^*}{\rho U_{av}^2}$$

$$T = \frac{(T^* - T_f)}{(T_w - T_f)}$$

$$t = \frac{t^*}{H/U_{av}}$$

where H is the height of the channel, U_{av} is the average velocity at the inlet, T_f is the reference temperature at the inlet, and T_w is the temperature of the walls and the cylinder.

The dimensionless parameters used include

$$Re = \frac{U_{av} H}{\nu}$$

$$Gr = \frac{g\beta(T_w - T_f)H^3}{\nu^2}$$

$$Pr = \frac{\nu}{\alpha}$$

$$Nu_x = \frac{T_w - T_f}{T_w - T_b} \left. \frac{\partial T}{\partial y} \right|_x$$

$$C_x = 2 \left. \frac{\partial u}{\partial y} \right|_x$$

where g is the normal gravity, β is the coefficient of thermal expansion, α is the thermal diffusivity, and T_b is the bulk temperature.

The boundary conditions are shown in Figure 17. The flow field domain is covered by rectangular mesh of isoparametric elements (99 x 32).

Figure 17. Boundary conditions imposed on the flow field with heat transfer

Figure 18 displays the velocity vectors, isotherms, and pressure contour plots for $Gr = 0$ and $Re = 100$. Here, thermal buoyancy forces are absent, and symmetric velocity profiles exist throughout the flow field which is at steady state. However, once thermal buoyancy forces are introduced with $Gr = 4 \times 10^4$ and $Gr = 16 \times 10^4$, the flow field becomes asymmetric as shown in Figures 19 and 20. The mass flow becomes more concentrated near the lower wall as the thermal buoyancy force increases. Periodic shedding also occurs in the wake region. These results compare well with those of Biswas *et al.*

The Reynolds number is now increased to 500. At $Gr = 0$, typical vortex shedding is present as shown in Figure 21. However, with the strong influence of thermal buoyancy at $Gr = 2 \times 10^6$, a reverse vortex is formed near the upper wall, and the vortex street disappears (see Figure 22).

a)

b)

c)

Figure 18. a) Velocity vector, b) isotherms, and c) pressure contour for $Gr = 0$ and $Re_{\infty} = 100$

a)

b)

c)

Figure 19. a) Velocity vector, b) isotherms, and c) pressure contour for $Gr = 4 \times 10^4$ and $Re_{\infty} = 100$

a)

b)

c)

Figure 20. a) Velocity vector, b) isotherms, and c) pressure contour for $Gr = 16 \times 10^4$ and $Re_\infty = 100$

a)

b)

c)

Figure 21. a) Velocity vector, b) isotherms, and c) pressure contour for $Gr = 0$ and $Re_{\infty} = 500$

a)

b)

c)

Figure 22. a) Velocity vector, b) isotherms, and c) pressure contour
for $Gr = 2 \times 10^6$ and $Re_{\infty} = 500$

The local Nusselt number and skin friction coefficient at the upper and lower walls are plotted in Figure 23 for various Grashof numbers at $Re = 100$. From the channel inlet to a distance of $x \approx 1.375$ before confronting the cylinder, Nu decreases continuously. This is followed by an increasing Nu , reaching the maximum at nearly the end of the cylinder. Nu will once again decrease, approaching its asymptotic value of a fully developed channel flow. As Gr increases, Nu will decrease at the upper wall and increase at the lower wall. The opposite behavior is observed for C . These results are all in excellent agreement with those of Biswas *et al.*.

Figure 23. Nusselt number and skin friction coefficient plots at $Re_{\infty} = 100$ for various Gr values

3 Numerical Analysis of Effects of Surface Suction and Blowing on Strouhal Frequency

3.1 Introduction

The primary objective of this investigation is to numerically determine the effects of surface suction and blowing on the Strouhal frequency. It is also of interest to determine the slopes of the $\log St - \log(Re_0^2/Re_\infty)$ curves and to determine if and how the St is affected by varying Re_∞ while keeping Re_0^2/Re_∞ constant. The fourth objective is to determine the asymptotic value of v_0 at which vortex shedding becomes totally suppressed. Lastly, the critical value of v_0 below which the change in St becomes negligible should also be determined. Some of these results are then compared to the theoretical predictions derived by Cohen.

In the analysis of the effects of surface suction and blowing on the Strouhal frequency, only the square cylinder was utilized. The mesh system is the same as that for the solid, square cylinder shown in Figure 7. The boundary conditions for the flow field is shown in Figure 24. The only difference between the solid square cylinder and the porous square cylinder problems is the boundary conditions on the cylinder surface. As shown in Figure 9, the boundary conditions on the solid square cylinder surface are zero for u and v , which is not the case for the porous square cylinder.

(Note: v_o is positive for blowing and negative for suction)

Figure 24. Boundary conditions imposed on the flow field for cylinder with normal surface suction and injection

3.2 Results

The results of the numerical analysis on the effects of normal surface suction and blowing on the Strouhal frequency are summarized in Table 2. The Strouhal frequencies are plotted versus the suction and blowing velocities in Figure 25, and the numerically obtained $\log St - \log(Re_o^2/Re_\infty)$ relationship is plotted in Figure 26. Discussions of these and other results are now given separately for the suction and blowing problems.

Table 2. Results of effects of normal surface suction and blowing

Type of Cylinder Surface	Surface Velocity	St	Period	$\frac{Re^*^2}{Re_\infty}$	% $\Delta St^{(1)}$	C_l Amplitude	C_d Range
with Suction	-0.45	0	∞	50.625	-100	0	1.00
	-0.40	0.089	11.190	40	-49.4	1.58	1.28-1.72
	-0.30	0.128	7.830	22.5	-27.8	1.61	1.67-1.98
	-0.20	0.157	6.368	10	-11.2	1.39	1.77-2.15
	-0.15	0.180	5.543	5.625	+2.0	1.22	1.72-2.15
	-0.10	0.205	4.883	2.5	+15.8	1.04	1.64-2.09
	-0.05	0.206	4.845	0.625	+16.7	0.73	1.43-1.70
	-0.025	0.190	5.273	0.156	+7.3	0.53	1.43-1.58
	-0.01	0.182	5.505	0.025	+2.8	0.50	1.51-1.65
Solid	0	0.177	5.655	0	-	0.51	1.57-1.72
with Blowing	0.01	0.173	5.798	0.025	-2.4	0.53	1.63-1.79
	0.025	0.166	6.030	0.156	-6.2	0.56	1.71-1.88
	0.05	0.151	6.608	0.625	-14.4	0.68	1.80-1.98
	0.10	0.142	7.043	2.5	-19.7	0.58	1.95-2.04
	0.15	0.132	7.605	5.625	-25.6	0.30	1.98-2.01
	0.20	0	∞	10	-100	0	1.86

Note: (1) % ΔSt is the percent deviation of St from the St for an impermeable cylinder.

Figure 25. Strouhal frequencies versus suction ($-v_o$) and blowing ($+v_o$) velocities

Figure 26. Numerical $\log St - \log(Re_o^2/Re_\infty)$ relationship

3.2.1 Suction

As shown in Table 2, the range of suction velocity between 0.01 and 0.45 was investigated. As the suction velocity increases, the St also increased as expected. However, the behavior of the St took an unexpected turn and began to decrease once v_o became larger than a limiting value as shown in Figure 25. Therefore, a certain suction velocity slightly greater than 0.15 exists at which the St will be the same as that for an impermeable cylinder. Further discussion on this phenomenon is given in Section 3.3.

The partially developed flow for a moderate suction velocity of 0.1 at $t = 37.5$ s is shown in Figure 27. Vortex shedding is obtained early in the flow. However, it is seen that the wake region is not fully developed when compared with that of the fully developed flow for the same suction velocity at $t = 187.5$ s (Figure 28). Figure 29 shows the fully developed flow for a high suction velocity of 0.4 at $t = 187.5$ s. At this suction velocity the wake region is not as disturbed by the vortex shedding as the flow with suction velocity of 0.1 is (Figure 28). The suction velocity of 0.4 is very close to the asymptotic velocity at which vortex shedding becomes totally suppressed, which is shown in Table 2 to be between 0.4 and 0.45. Therefore, the flow at a suction velocity of 0.4 is almost dominated by the suction, and its wake region becomes less affected by the shedding of vortices. Figure 30 shows the suction dominated flow with a suction velocity of 0.45 at $t = 187.5$ s where vortex shedding disappears. The effect of the suction on the flow velocity in the wake

a)

b)

Figure 27. a) Velocity vector and b) pressure contour for partially developed flow with 0.1 suction velocity at $t = 37.5$ s

a)

b)

Figure 28. a) Velocity vector and b) pressure contour for fully developed flow with 0.1 suction velocity at $t = 187.5$ s

a)

b)

Figure 29. a) Velocity vector and b) pressure contour for fully developed flow with 0.4 suction velocity at $t = 187.5$ s

a)

b)

Figure 30. a) Velocity vector and b) pressure contour for fully developed flow with 0.45 suction velocity at $t = 187.5$ s

far downstream is negligible; here, the velocity is very close to V_∞ .

Note that the pressure contour is symmetric.

For suction velocities below 0.2, the variations of the C_l , C_d , and the surface pressures at the center of each side of the cylinder are similar to those for the impermeable cylinder shown in Figure 12. At suction velocities of 0.2 and above (but below the asymptotic value), the C_l , C_d , and surface pressure variations begin to display different fluctuations. An example of the different C_l , C_d , and surface pressure variations is given in Figure 31 for the suction velocity of 0.4. The $C_{l_{\max}}$, $C_{l_{\min}}$, and C_{l_0} velocity vector and pressure contour plots for all of the suction velocities analyzed (other than the upper bound case) are similar to those for the impermeable cylinder shown in Figures 13-15.

The effects of varying Re_∞ on St while keeping Re_o^2/Re_∞ constant was determined. At a constant ratio of $Re_o^2/Re_\infty = 2.5$, the Re_∞ was increased from 250 to 500, and the suction velocity was lowered from 0.1 to 0.07071. The results are shown in Table 3. Note that the increase in St is about 3%. At a constant ratio of $Re_o^2/Re_\infty = 22.5$, the Re_∞ was increased from 250 to 500, and the suction velocity was lowered from 0.3 to 0.21213. The results are shown in Table 4. Note that the increase in St is 28.4%. Therefore, these initial results indicate that St is dependent on Re_∞ at a constant ratio of Re_o^2/Re_∞ .

Figure 31. C_l , C_d , and local pressures for flow with 0.4 suction velocity

Table 3. Effects of varying Re_∞ for suction at $Re_o^2/Re_\infty = 2.5$

Suction	St	period	C_l Amplitude	C_d Range
$v_o = -0.1$ $Re_\infty = 250$	0.205	4.883	1.04	1.64 to 2.09
$v_o = -0.07071$ $Re_\infty = 500$	0.212	4.718	1.08	1.69 to 2.24

Table 4. Effects of varying Re_∞ for suction at $Re_o^2/Re_\infty = 22.5$

Suction	St	period	C_l Amplitude	C_d Range
$v_o = -0.3$ $Re_\infty = 250$	0.128	7.830	1.61	1.67 to 1.98
$v_o = -0.21213$ $Re_\infty = 500$	0.164	6.098	1.68	2.11 to 2.68

3.2.2 Blowing

The range of blowing velocity between 0.01 and 0.2 was investigated as shown in Table 2. As expected, the St decreased as the blowing velocity increases.

The partially developed flow for a blowing velocity of 0.1 at $t = 75$ s is shown in Figure 32. At this stage, it is clearly seen in the

a)

b)

Figure 32. a) Velocity vector and b) pressure contour for partially developed flow with 0.1 blowing velocity at $t = 75$ s

velocity vector plot that the vortex shedded was not immediately broken down as it is carried downstream; two separate and complete vortices are observed in the domain plotted. As the flow becomes fully developed at $t = 187.5$ s, only one complete vortex is seen in the velocity vector plot shown in Figure 33. Figure 34 shows the fully developed flow for a high blowing velocity of 0.15 at $t = 247.5$ s. It is seen that the region close to the cylinder is dominated by the blowing process. Vortices are therefore shed not directly off of the cylinder surface but at a relatively long distance downstream. Note that the blowing velocity of 0.15 is very close to the asymptotic velocity at which vortex shedding becomes totally suppressed, which is determined to be between 0.15 and 0.2 from Table 2. At this blowing velocity of 0.15, a much longer time was required for the flow to become fully developed at which the Strouhal frequency remains constant. The flow shown in Figure 34 is at the stage where the flow had just become fully developed. This may explain the existence of two small, complete and separate vortices seen on the velocity vector plot. If the flow were allowed to proceed to a later timestep, the older vortex may cease to exist as it becomes broken down at the birth of the new vortex. When the blowing velocity is further increased to 0.2, the flow field becomes totally dominated by the blowing, and vortex shedding disappears as seen in Figure 35 at $t = 187.5$ s. The effect of blowing is strong in the wake even at a distance far downstream. It can be seen that some actions, perhaps circulations, may be occuring in the wake far downstream. This is a possible cause for the pressure contour to be slightly asymmetric in the region far downstream. However, the existence of these weak

a)

b)

Figure 33. a) Velocity vector and b) pressure contour for fully developed flow with 0.1 blowing velocity at $t = 187.5$ s

a)

b)

Figure 34. a) Velocity vector and b) pressure contour for fully developed flow with 0.15 blowing velocity at $t = 247.5$ s

a)

b)

Figure 35. a) Velocity vector and b) pressure contour for fully developed flow with 0.2 blowing velocity at $t = 187.5$ s

circulations, or vortices, will not suffice to constitute vortex shedding since a large region around the cylinder will remain dominated by the strong blowing process, and the lift coefficient remains zero.

For the blowing velocities below the asymptotic value examined, the variations of the C_l , C_d , and the surface pressures at the center of each side of the cylinder are similar to those for the impermeable cylinder shown in Figure 12. The only exception is the case of $Re_\infty = 500$ and $v_o = 0.07071$, to be discussed in the following paragraph. The velocity vector and pressure contour plots at $C_{l_{max}}$, $C_{l_{min}}$, and C_{l_0} for all of the blowing velocities analyzed (other than the upper bound case) are also similar to those for the impermeable cylinder shown in Figures 13-15.

The effects of varying Re_∞ on St at a constant Re_o^2/Re_∞ was determined. At a constant ratio of $Re_o^2/Re_\infty = 2.5$, the Re_∞ was increased from 250 to 500, and the blowing velocity was lowered from 0.1 to 0.07071. The results are shown in Table 4. Note that the decrease in St is less than 8%. At a constant ratio of $Re_o^2/Re_\infty = 5.625$, The Re_∞ was increased from 250 to 500, and the blowing velocity was lowered from 0.15 to 0.10607. Note that the change in the St is 1%. Therefore, these initial results indicate that the effects of varying Re_∞ on the St for a particular Re_o^2/Re_∞ are only slight. The variations of the C_l , C_d , and the surface pressures at the center of each side of the cylinder are shown in Figure 36 for a blowing velocity of 0.07071 at $Re_\infty = 500$.

Figure 36. C_l , C_d , and local pressures for flow with 0.07071 blowing velocity at $Re_\infty = 500$

Table 5. Effects of varying Re_∞ for blowing at $Re_o^2/Re_\infty = 2.5$

Blowing	St	period	C_l Range	C_d Range
$v_o = 0.1$ $Re_\infty = 250$	0.142	7.043	-0.58 to 0.58	1.95 to 2.04
$v_o = 0.07071$ $Re_\infty = 500$	0.131	7.620	-1.63 to 1.50	2.09 to 2.40

Table 6. Effects of varying Re_∞ for blowing at $Re_o^2/Re_\infty = 5.625$

Blowing	St	period	C_l Range	C_d Range
$v_o = 0.15$ $Re_\infty = 250$	0.132	7.605	-0.3 to 0.3	1.98 to 2.01
$v_o = 0.10607$ $Re_\infty = 500$	0.130	7.680	-1.16 to 1.16	2.10 to 2.31

3.3 Comparison and Discussion of the Analytical and Numerical Solutions

The numerical results obtained in this investigation supported Cohen's theoretical predictions for a certain range of suction and blowing velocities. As predicted, the Strouhal frequency decreased as the blowing velocity decreases, and the St increases as the suction velocity increases. However, a limiting velocity exists for the suction

problem such that the St will begin to decrease, instead of increasing, once the suction velocity is larger than the limiting value. The non-dimensional limiting velocity was determined numerically to be between -0.025 and -0.1 . This phenomenon does not appear in Cohen's analytical prediction. As suggested by Cohen, an upper bound, or asymptote, exists for the suction problem where vortex shedding becomes completely suppressed in a suction dominated flow. This non-dimensional upper bound v_o was determined to be between -0.4 and -0.45 . For a cylinder with a suction velocity greater than this upper bound, the boundary layer will be captured by the suction process and will disappear as it is pulled into the cylinder. Since vortex shedding originates from the boundary layer, vortex shedding will also disappear ($St = 0$) as the boundary layer disappears. For the St to be a continuous function of v_o , then St must decrease for a certain range of v_o before reaching zero at the upper bound value of v_o . This mathematical reasoning explains the initial increase followed by the decreasing behavior of the St as v_o increases. Near the limiting value of v_o , a possible physical explanation for the increasing St to reverse and begin decreasing is that the vortex shedding may begin to overlap as v_o increases and δ decreases. The theoretical prediction was for the vortex shedding to increase as v_o increases. However, as the vortices are formed and shed at an increasing rate, they may begin to overlap, thus decreasing the actual shedding frequency instead of increasing it.

According to Cohen's prediction as seen in Figure 6, vortex shedding will disappear as the blowing velocity reached its asymptotic value. This is because, similar to the suction problem, the

boundary layer will eventually get "blown away" as v_o increases to its asymptotic value. As can be seen in Table 2, this non-dimensional upper bound occurs at v_o between 0.15 and 0.2, which is less than 20% of the freestream velocity. This shows that a much smaller $|v_o|$ is required for the vortex shedding to disappear in the blowing problem than for suction. Therefore, if normal surface suction or blowing were to be used for practical design applications such as in suppressing vortex shedding, then normal surface blowing would be favored over normal surface suction.

The change in St increases more rapidly as v_o increases, and the slopes of the $\log St - \log(Re_o^2/Re_\infty)$ curves are not equal to ± 1 as predicted by Cohen. The slopes vary for different v_o , ranging from -0.05 to -0.09 for the blowing problem. For the suction problem, the slope varies from 0.02 to 0.06 for v_o below the limiting value and from -0.16 to -0.62 for v_o greater than the limiting value.

As seen in Table 2, the change in the St is only less than $\pm 20\%$ at $Re_o^2/Re_{\infty \text{critical}} \approx 1$ which is in accordance with Cohen's prediction. If $\Delta St < 10\%$ were the criteria for the ΔSt to be considered as negligible, then the critical value of v_o for the suction problem would fall between -0.025 and -0.05. For the blowing problem, the critical v_o would fall between 0.025 and 0.05. Porous cylinders operating at v_o below the critical value can then be treated as impermeable.

In Section 1.1, the $St - Re_\infty$ relationships were given for impermeable circular and square cylinders. It can be seen from Figures 1 and 3 that the St characteristics for the impermeable circular and square cylinders are not significantly different. By extrapolating these results to include normal surface suction and

blowing, it is assumed that the results for the porous circular and square cylinders will not differ significantly at a low Re_∞ . This is the reason why the analytically predicted results were compared with the numerically determined results, even though different body shapes were used. However, it is important to keep in mind that future experimental results may indicate otherwise, where the results may actually be significantly dependent on the body shape.

Overall, by comparing the analytical and the numerical $\log St - \log(Re_0^2/Re_\infty)$ relationships plotted in Figures 6 and 26, fairly good agreement exists for the analytical and numerical results. The only significant difference is the unexpected decreasing portion of the curve determined numerically for suction.

4 Conclusions

The Runge-Kutta finite element program used in this analysis was verified by examining several bench mark cases for flows over solid, rectangular cylinders. The solutions were found to be in excellent agreement with existing numerical and experimental data.

In determining the effects of blowing and suction on the $St-Re_\infty$ relationship, this numerical investigation was able to support a large part of Cohen's analytical predictions (see Section 3.3). The results obtained for the normal surface suction and blowing problems are now summarized separately.

For the suction problem, the following conclusions were formed:

- 1) St increases with increasing suction for v_0 below the limiting value determined to be between -0.025 and -0.1 . At v_0 beyond the limiting value, St will decrease with increasing suction.
- 2) A value of v_0 exists between -0.15 and -0.2 where the St value is equal to that for the impermeable cylinder.
- 3) The asymptotic value of v_0 at which vortex shedding is totally suppressed ($St = 0$) is determined to be between -0.4 and -0.45 .
- 4) For v_0 below the limiting value, the critical value of the v_0 at which the change in St (with respect to the St for

the impermeable cylinder) is less than 10% falls between -0.025 and -0.05. Letting $\Delta St < 10\%$ be the criteria for negligible change in St , then cylinders with surface suction velocities less than this critical value can be treated as impermeable.

- 5) For the same value of Re_0^2/Re_∞ , the St will increase slightly as Re_∞ increases. It was not determined if and how this behavior would change depending on whether the v_0 is below or beyond the limiting value.
- 6) The slope of the $\log St - \log(Re_0^2/Re_\infty)$ curve ranges from 0.02 to 0.06 for v_0 below the limiting value and from -0.16 to -0.62 at v_0 greater than the limiting value. The change in St increases more rapidly as v_0 increases.

For the blowing problem, the following conclusions were made:

- 1) St decreases with increasing blowing.
- 2) The asymptotic value of v_0 at which vortex shedding is totally suppressed is determined to be between 0.15 and 0.2.
- 3) The critical value of v_0 at which the change in St (with respect to the St for the impermeable cylinder) is less than 10% falls between 0.025 and 0.05. Letting $\Delta St < 10\%$ be the criteria for negligible change in St , then

cylinders with surface blowing velocities less than this critical value can be treated as impermeable.

- 4) For the same value of Re_0^2/Re_∞ , the St will decrease slightly as Re_∞ increases.
- 5) The slope of the $\log St - \log(Re_0^2/Re_\infty)$ curve ranges from -0.05 to -0.09. The change in St increases more rapidly as v_0 increases.

5 Future Work

Currently, the theoretical analysis by Cohen and the numerical analysis presented in this thesis are the only studies found dealing directly with the problem of flow over porous cylinders with normal surface suction and blowing. Therefore, much analytical, numerical, and experimental work remain to be done. Included in the following paragraphs are suggestions of some future work from the current perspective. As investigation of the suction and blowing problem progresses, many new questions will surely arise, as will new areas for further research.

The trend of the behavior of the Strouhal frequency for the suction and blowing problem was determined numerically. Among the conclusions, Cohen's boundary layer growth and collapse model of vortex shedding appears to successfully predict the deviation of the Strouhal number (from the impermeable case) near the critical Re_o^2/Re_∞ , i.e. suction tends to increase the Strouhal frequency, whereas blowing decreases it.

The model, however, fails grossly at larger values of Re_o^2/Re_∞ . In the case of suction, this could be attributed to the suppression of vortex shedding, which is a well-known phenomenon. For normal blowing, however, a high rate of injection causes the formation of a "solid body", analagous to the Rankine half body (i.e. line source superposed on uniform flow) in potential flow theory. This is evident upon examining Figure 35a. As a consequence, viscous

effects arising from fluid-solid wall interactions are considerably reduced, and hence, vortex shedding is eliminated altogether.

In the computational area, further investigations need to be carried out to determine if and how this trend is affected by varying the resolution of the mesh and the body shape (i.e. rectangular cylinders with various aspect ratios). Secondly, it was reported by Anderson *et al.* [14] that, for a numerical scheme which depends on numerical errors to lead the flow away from a symmetric state, favorable comparisons of computed and experimental solutions (in parameters such as the Strouhal frequency, drag, and so forth) does not sufficiently validate the numerical scheme. Since there are no experimental data available to verify the numerical results for the suction and blowing problems obtained in this study, the validity of these numerical results is difficult to determine. Anderson *et al.* further stated that, to numerically calculate the development of vortex shedding accurately, a fixed perturbation should be introduced. For this particular problem of vortex shedding over a cylinder, the symmetric initial condition can be perturbed by introducing a vortex near the cylinder. It was shown that without the imposition of a fixed perturbation, the vortex street actually disappears as the mesh resolution increases. Therefore, before experimental data becomes available for the suction and blowing problem, the validity of the current results can be checked by introducing perturbations into the numerical calculations and determining whether the resulting solutions remain similar to the current results. Lastly, further investigation is needed to completely

understand the effects of varying the Re_∞ on the St for a particular Re_o^2/Re_∞ ratio.

Currently, there are no experimental data found which deals directly with the problem of normal surface suction and blowing. Therefore, experiments need to be conducted to determine the trend of the St behavior as affected by surface suction and blowing. The asymptotes of the suction and blowing velocities at which the vortex shedding becomes suppressed also need to be determined. A physical explanation is needed for the initial increase of the St at suction velocities below the critical value followed by the decreasing behavior of the St for suction velocities greater than the critical value. This behavior of the Strouhal frequency could be related to the width of the wake [5] and can be determined experimentally by wind tunnel testing or in a water tank. Also, the range of suction and blowing velocities which has negligible impact on the Strouhal frequency such that the cylinder can be treated as impermeable should be determined. Finally, the impact of varying the Re_∞ and the shape of the object on the St behavior should also be experimentally determined.

Appendix A

The coordinate system for the outflow boundary, Γ_3 , is shown in Figure A-1, where the two right-most columns of grid points in the mesh system are considered.

Figure A-1. The outflow boundary condition

The normal and tangential projections of the Navier-Stokes equations on Γ_3 are

$$\frac{\partial p^{n+1}}{\partial \eta} = \frac{1}{\text{Re}} \nabla^2 \mathbf{u}_\eta^n - (\mathbf{u}^n \cdot \nabla) \mathbf{u}_\eta^n - (\mathbf{u}_\eta^{n+1} - \mathbf{u}_\eta^n) / \Delta t^{n+1} \quad (\text{A1})$$

$$\frac{\partial p^{n+1}}{\partial \tau} = \frac{1}{\text{Re}} \nabla^2 \mathbf{u}_\tau^n - (\mathbf{u}^n \cdot \nabla) \mathbf{u}_\tau^n - (\mathbf{u}_\tau^{n+1} - \mathbf{u}_\tau^n) / \Delta t^{n+1} \quad (\text{A2})$$

$$\frac{\partial \mathbf{u}_\eta^{n+1}}{\partial \eta} + \frac{\partial \mathbf{u}_\tau^{n+1}}{\partial \tau} = 0 \quad (\text{A3})$$

Taking the normal and tangential derivatives of equations (A1) and (A2) and using (A3), the boundary pressure Poisson equation is obtained.

$$\left(\frac{\partial^2}{\partial \eta^2} + \frac{\partial^2}{\partial \tau^2} \right) p^{n+1} = 2 \frac{\partial \mathbf{u}_\eta^n}{\partial \eta} \frac{\partial \mathbf{u}_\tau^n}{\partial \tau} - 2 \frac{\partial \mathbf{u}_\eta^n}{\partial \tau} \frac{\partial \mathbf{u}_\tau^n}{\partial \eta} \quad (\text{A4})$$

Dirichlet boundary condition is required to solve for equation (A4) using the Galerkin method at each time step. However, the nodes on which Dirichlet boundary condition is specified will not vary with time, which would not be the proper boundary condition. Therefore, the adjacent nodes to the open boundary are considered in an approximate sense. The integration of equation (A4) can be considered as those over the nodes adjacent to Γ_3 and can be written as

$$[\mathbf{L}] \begin{pmatrix} \mathbf{p}^{n+1} \\ \mathbf{p}^n \end{pmatrix} = \begin{pmatrix} \mathbf{f}^n \\ 0 \end{pmatrix}$$

where $\mathbf{p}^{n+1}$ on Γ_3 are unknown, $\mathbf{p}^n$ on the adjacent nodes are known, $\mathbf{L}$ is the Laplace matrix, and $\mathbf{f}^n$ is the velocity gradient vector on the right hand side of equation (A4). Thus, only the pressure at time zero is necessary to solve the boundary pressure Poisson equation (A4).

References

1. Davis, R., and Moore, E., "A Numerical Study of Vortex Shedding from Rectangles", *Journal of Fluid Mechanics*, Vol. 116, 1982, p. 475-506.
2. Okajima, A., Nagahisa, T., and Rokugoh, A., "A Numerical Analysis of Flow around Rectangular Cylinders", *JSME International Journal, Series II*, Vol. 33, No. 4, 1990, p. 116-125.
3. Cohen, R., "Predicting the Effects of Surface Suction and Blowing on the Strouhal Frequencies in Vortex Shedding", *JSME International Journal, Series II*, Vol. 34, No. 1, 1991, p. 30-38.
4. Chorin, A., "Numerical Solution of the Navier-Stokes Equations", *Mathematics of Computations*, Vol. 22, 1968, p. 745-762.
5. Okajima, A., "Strouhal Numbers of Rectangular Cylinders", *Journal of Fluid Mechanics*, Vol. 123, 1982, p. 379-398.
6. Shimura, M., and Kawahara, M., "Two Dimensional Finite Element Flow Analysis Using the Velocity Correction Method", *Structural Engineering/Earthquake Engineering*, Vol. 5., No. 2, October 1988, p. 255-263.
7. Yoshida, Y., and Nomura, T., "A Transient Solution Method for the Finite Element Incompressible Navier-Stokes Equations", *International Journal of Numerical Methods in Fluids*, Vol. 5, 1985, p. 873-890.
8. Rothberg, R., "A Numerical Study of Vortex-Shedding Suppression in Laminar Flow about a Cylinder near a Plane Boundary", Ph.D. Thesis, Dept. of Mechanical Engineering & Materials Science, Rice University, Houston, 1989.
9. Briscolini, M., and Santangelo, P., "Development of the Mask Method for Incompressible Unsteady Flows", *Journal of Computational Physics*, Vol. 84, 1989, p. 57-75.

10. Tezduyar, T. E., and Shih, R., "Numerical Experiments on Downstream Boundary of Flow Past Cylinder", *Journal of Engineering Mechanics*, Vol. 117, No. 4, 1991, p. 854-871.
11. Tanahashi, T., and Okanaga, H., "GSMAC Finite Element Method for Unsteady Incompressible Navier-Stokes Equations at High Reynolds Numbers", *International Journal of Numerical Methods in Fluids*, Vol. 11, 1990, p. 479-499.
12. Tabata, M., and Fujima, S., "An Upwind Finite Element Scheme for High-Reynolds-Number Flows", *International Journal for Numerical Methods in Fluids*, Vol. 12, 1991, p. 305-322.
13. Ramaswamy, B., and Jue, T. C., "Some Recent Trends and Developments in Finite Element Analysis for Incompressible Thermal Flows", *International Journal for Numerical Methods in Engineering* (to appear).
14. Anderson, C., Greengard, C., Greengard, L., Rokhlin, V., "On the Accurate Calculation of Vortex Shedding", *Physics of Fluids A*, Vol. 2, No. 6, June 1990, p. 883-885.
15. Biswas, G., Laschefski, H., Mitra, N., Fiebig, M., "Numerical Investigation of Mixed Convection Heat Transfer in a Horizontal Channel with a Built-in Square Cylinder", *Numerical Heat Transfer, Part A*, Vol. 18, 1990, p. 173-188.