FUNDAMENTALS OF VORTEX METHOD TO FLOW AROUND A BLUFF BODY

T. K. Sheel
Université Catholique de Louvain
Batiment Euler, 4 Avenue G. Lemaitre
1348 Lovain-La-Neuve, Belgium
Fax: +32 10 47 21 80
E-mail: tarun.sheel@uclouvain.be

Abstract: *The present study aims at an efficient application of the vortex method to flow around bluff body. Most bluff body systems react with fluids one way or the other. This can be designed to reduce heat transfer or drag, only if the surrounding fluid is calculated correctly. The pressure and velocity field have been calculated for the convection of vortex elements around the circular cylinder. The corresponding drag and lift coefficients have been investigated carefully. Further investigations are considered.*

Keywords: *Vortex Method, Bluff Body, Drag Force, Lift Force, Instability, Martensen Analysis.*

1. Introduction

Vortex shedding behind a circular cylinder has been the subject of a number of studies [1]. Given a long circular cylinder with its axis perpendicular to fluid flow, the well-known Karman-type (asymmetric) vortex shedding may occur behind the cylinder, the control or suppression of which is of great interest as it is closely related to various fluid-mechanical properties of practical importance, such as flow-induced forces, vibrations and noises, and the efficiencies of heat and mass transfer. There are several situations where this type of vortex shedding may cease, and one of them is when a plane boundary or ground is located near the cylinder; the focus of the present study is on this flow configuration.

The characteristics of flow around a circular cylinder placed near and parallel to a ground are governed not only by the Reynolds number ***Re*** but also by the gap ratio, i.e., the ratio of the gap between the cylinder and the ground, *h*, to the cylinder diameter *d* [2]. However the mechanisms of the flow and force variations caused by different *h/d*, or 'ground effect', are in general rather complicated since they can be significantly affected by the state of the boundary layer formed on the ground [2;3].

A vortex method is a computation technique for simulation fluid flows. To simulate the fluid flow, vortex methods attempt to simulate only the evolution of the vorticity field which is the curl of the velocity field. The reason why some people are only interested in the vorticity of a flow is that there are many interesting flows where the vorticity is confined to a very small region of space even though the whole flow occupies are larger or even unbounded area. Flow around a circular cylinder, however, is still a very challenging subject in itself in today's computational fluid dynamics (CFD) even if the cylinder is outside the ground effect.

Vortex methods were developed as a grid-free methodology that would not be limited by the fundamental smoothing effects associated with grid-based methods. To be practical, however, vortex methods require means for rapidly computing velocities from the vortex elements.

As is the case in every numerical simulation, the first step was taken by applying the calculation to a flow around a body. In the first case the convection of a single vortex element around a circular wall was calculated. It was found that certain corrections had to be made to account for the curvature of the body and to conserve the total circulation of the field. In the next case these corrections were applied to calculate the vortex shedding from a circular cylinder. The representation of vortex shedding was successful but the consideration of the viscous diffusion and dissipation was still insufficient. This is another matter of controversy in the vortex method. The two main ways to account for the viscous effects are the deterministic method and the random walk method. They are both under careful consideration.

2. Vortex Element Method (VEM)

Vortex element method have been growing in popularity in last three decades. As their name indicates, they are based on the discretization of vorticity-a quantity that has a compact support in many physical problems-thereby making this approach interesting [4].

The three-dimensional incompressible flow of a viscous fluid has been studied here. The evolution equation for vorticity is

$$\frac{D\boldsymbol{\omega}_i}{Dt} = (\boldsymbol{\omega}_i \cdot \nabla)\mathbf{u} + \nu\nabla^2\boldsymbol{\omega}_i \quad (1)$$

where $\boldsymbol{\omega}_i$ is vorticity defined as $\boldsymbol{\omega} = \nabla \times \mathbf{u}$, $\mathbf{u}$ is the velocity of vortex element, $(\boldsymbol{\omega} \cdot \nabla)\mathbf{u}$ is called stretching term and represents the rate of change of vorticity by deformation of vortex lines and the term $\boldsymbol{\nu}\nabla^2\boldsymbol{\omega}$ represents the change of vorticity by viscous diffusion. The velocity field on three-dimensional problem is,

$$\mathbf{u}(\mathbf{x}) = -\frac{1}{4\pi}\int \frac{(\mathbf{x}-\mathbf{x}')\times\boldsymbol{\omega}(\mathbf{x}')}{|\mathbf{x}-\mathbf{x}'|^3} dV(\mathbf{x}') \quad (2)$$

where $\mathbf{x}$, and $\mathbf{x}'$ are positions of vortex elements and dV is the volume of element.

Using the Winckelmans model [5] as a cutoff function, Biot-Savart law is as follows

$$\mathbf{u}_i = -\frac{1}{4\pi}\sum_{j=1}^{N} \frac{\mathbf{r}_{ij}^2 + (5/2)\sigma_j^2}{\left(\mathbf{r}_{ij}^2 + \sigma_j^2\right)^{5/2}} \mathbf{r}_{ij} \times \boldsymbol{\gamma}_j \quad (3)$$

where $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$, σ_j and $\boldsymbol{\gamma}_j$ are distance of the position vector, core radius and strength of element. The subscript *i* stands for the target elements, while *j* stands for the source elements.

When the stretching term of Eq. (1) can be discretized as follows:

$$\frac{d\boldsymbol{\omega}_i}{dt} = (\boldsymbol{\omega}_i \cdot \nabla)\mathbf{u} \quad (4)$$

if I put vortex strength $\gamma_i = \boldsymbol{\omega}_i d^3\mathbf{x}_i$ in equation (4), then it becomes

$$\frac{d\gamma_i}{dt} = (\gamma_i \cdot \nabla)u_i \quad (5)$$

Hence, the vortex strength of an individual element is expressed by Eq. (3) in a discretized formulation as

$$\frac{d\boldsymbol{\gamma}_i}{dt} = \frac{1}{4\pi}\sum_j \left\{ -\frac{|\mathbf{r}_{ij}|^2 + (5/2)\sigma_j^2}{\left(|\mathbf{r}_{ij}|^2 + \sigma_j^2\right)^{5/2}} \boldsymbol{\gamma}_i \times \boldsymbol{\gamma}_j + 3\frac{|\mathbf{r}_{ij}|^2 + (7/2)\sigma_j^2}{\left(|\mathbf{r}_{ij}|^2 + \sigma_j^2\right)^{7/2}} (\boldsymbol{\gamma}_i \cdot \mathbf{r}_{ij})(\mathbf{r}_{ij} \times \boldsymbol{\gamma}_j) \right\} \quad (6)$$

where all notations denotes the same meaning as of Eq. (3). Further details mathematical formulations see [6;7].

2.1 Viscous Diffusion

Vortex methods originate in inviscid methods and recent studies mostly focus on their extension to viscous flows. Though, this has not been a straightforward task and the diversity of methods has become quite large. The random vortex method (RVM)[8] uses a stochastic interpretation of the diffusion equation. It has served an important role in the early development of viscous diffusion schemes, but its slow convergence rate prompted the development of alternative methods. The core spreading method (CSM) by Kuwahara[9] and Leonard[10] uses a deterministic approach, which changes the standard deviation of the Gaussian distribution to match the fundamental solution of the diffusion equation. A straightforward implementation of this method lacks convergence due to the fact that the ever-expanding Gaussian distribution moves with the velocity at its center. Local spatial refinement [11] can circumvent this problem, though this will introduce a large amount of error without careful consideration [12;13;14].

The particle strength exchange (PSE) by [15] redistributes the strength among vortex elements by solving the integral equation of the Laplacian operator. The location of elements are used as quadrature points, thus requires them to be nearly uniform for an accurate calculation [16]. The vortex redistribution method (VRM) by [17] also redistributes the strength of vortex elements but by solving an underdetermined system of equations to equate the truncated Taylor series of the new distribution with that of the exactly diffused vorticity. Although, restrictions of particle nonuniformity are not as severe as the PSE, it is obvious that a sufficient number must exist in the neighborhood. The insertion and merging of particles is still an open area of research, as is the case with CSMs.

In most cases a vortex element has three properties, circulation, core radius, and velocity. The CSM changes the core radius, PSE and RVM change the circulation to account for diffusion. The diffusion velocity method by [18] modifies the velocity instead, where the diffusion velocity becomes the product of $-\nu/\omega$ and the gradient of vorticity. For regions of zero vorticity the $-\nu/\omega$ becomes singular, so an algorithm which does not increase the vorticity magnitude outside of the computational vorticity support [19] is essential to this scheme. There exist many other ways to calculate the viscous diffusion of vorticity using a semi-Lagrangian discretization, such as the vortex in cell (VIC), free Lagrangian, triangulated, moving particle semi-implicit method(MPS) [20], and moving least squares (MLS). The present study focuses on pure Lagrangian schemes (with remeshing in some cases), thus semi-Lagrangian methods are out of scope.

In particular, we will focus only the scheme of core spreading method as follows.

The CSM is way to discretize the viscous diffusion equation

$$\frac{D\boldsymbol{\omega}_i}{Dt} = \nu\nabla^2\boldsymbol{\omega}_i \qquad (7)$$

where the Green's function solution is

$$\omega_i = \frac{\gamma_j}{(4\pi\nu t)^{d/2}}\exp\left(-\frac{|x_j - x_i|^2}{4\nu t}\right) \qquad (8)$$

ω is the vorticity, ν is the kinematic viscosity, γ is the circulation, $\boldsymbol{x}$ is the position vector, and d is the dimensionality of the problem. The subscript i stands for the target elements, while j stands for the source elements. The CSM uses a cutoff function ζ to discretize the diffusion equation. In this case the vorticity at an arbitrary point can be expressed as

$$\omega_i = \sum_j \gamma_j \zeta\left(|x_j - x_i|\right) \qquad (9)$$

A common choice for the cutoff function is the Gaussian distribution

$$\zeta = \frac{1}{(2\pi\sigma^2)^{d/2}}\exp\left(-\frac{|x_j - x_i|^2}{2\sigma^2}\right) \qquad (10)$$

If we substitute Eq. (10) into Eq. (9), we can see that changing the variance of the Gaussian distribution according to

$$\sigma^2 = 2\nu t \qquad (11)$$

will result in the heat kernel Eq. (8). σ is often referred to as the core radius of the vortex blob, and represents the physical length scale of the vortex elements.

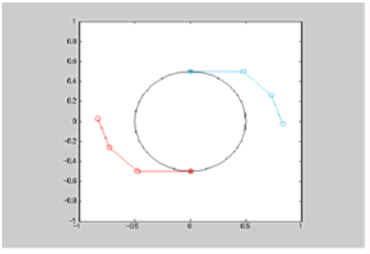
The radial basis function interpolation [12] is used every ten time steps to ensure the convergence of the core spreading method [21]. The convection is solved by updating the position of vortex elements according to their velocity

$$\frac{dx_i}{dt} = u_i \qquad (12)$$

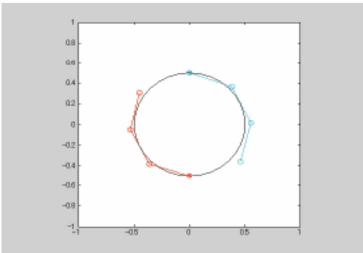
3. Numerical Result

3.1 Convection Term

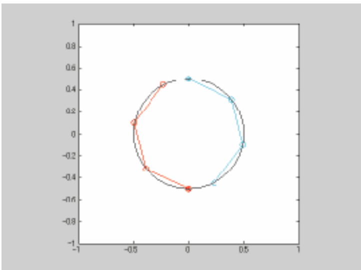
The convection term was evaluated using a simple program of two vortex elements circling around one another. The velocities induced on the body were calculated from the vorticity of each element by using the Biot-Savart law (Eq. 3). The displacement was calculated from the induced velocity by using the predictor-corrector method. While the common method for time advancing in the vortex method is the Adams-Bashforth method, which has a constant accuracy of the 2nd order. By using the predictor-corrector method, the order of accuracy can be controlled by using more correction steps. Although, the number of correction steps depends on how much time one can afford to spend on the convection term. Figure 1 shows the convection of two elements before and after the correction step.



(a) Before correction



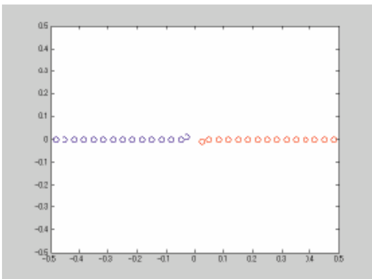
(b) One step correction



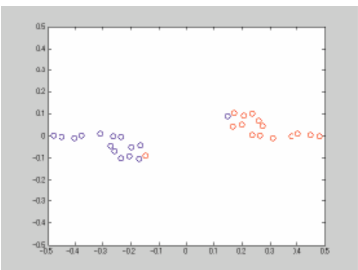
(c) Five steps correction

Fig. 1: Convection term

3.2 Kelvin-Helmholtz Instability



(a) Initial Displacement



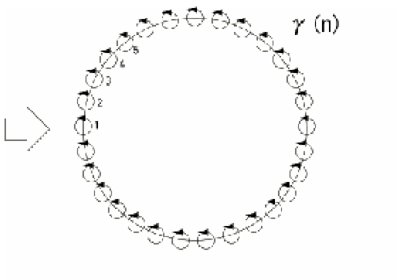
(b) After 40 time steps

Fig. 2: Instability

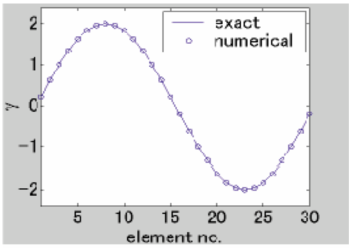
This simple program was then applied to an actual calculation of the Kelvin-Helmholtz instability, which can be observed in a field where there are two parallel flows of different velocities. In the first case the two elements in the middle were slightly displaced in order to simulate an initial disturbance shown in figure 2(a). The results were as expected causing the vortex sheet to roll-up eventually as seen in figure 2(b). In the second case, initial disturbance was not applied but the same type of roll-up was observed. It turned out that this instability was caused by the accumulating round-off error, which resulted in an artificial displacement of the vortex elements.

3.3 Martensen Analysis

The next step was taken by applying this calculation to a flow around a body. The no-slip and no-through-flow conditions at the surface of the body were satisfied by using a vortex panel and source panel treatment. Application of elements to the wall may seem artificial but it is important to point out that this is the result of a discretized boundary integral equation and is a direct numerical solution of the governing equations. The vorticity of the wall elements is calculated from a matrix analysis called the Martensen analysis. In the first case, the convection of a single vortex element around a circular wall was calculated. It was found that certain corrections had to be made to account for the curvature of the body and to conserve the total circulation of the field as shown in Fig. 3. The curvature correction accounts for the self-induced velocity on a wall element due to curvature, while the circulation correction assures Kelvin's circulation theorem. It was confirmed that, without these corrections the convection error would become unacceptable near the wall. Figure 3 represents the analytical (exact) and calculated (numerical) results for this analysis. It can be observed that the both results are coincided each other after applied the correction method (source panel).



(a) Before correction



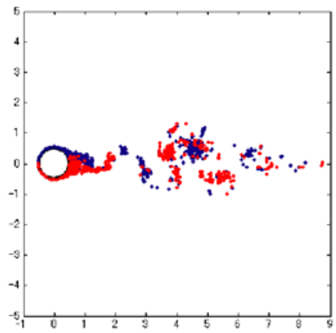
(b) After correction

Fig. 3: Velocity of wall elements

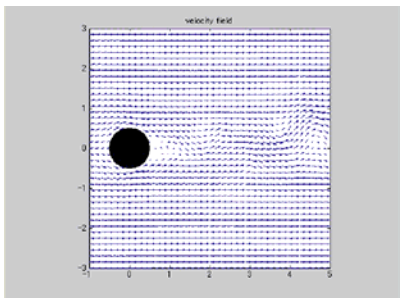
3.4 Vortex Shedding

Now these corrections were applied to calculate the vortex shedding from a circular cylinder. Vortex elements were shed from each element on the wall. For this case, the vortex elements were shed only once and the convection of these elements were evaluated. Then the case for periodic shedding was tested. Instead of shedding one set of vortex elements at the beginning of the calculation, they were shed every time step. This causes a vortex cloud to form around the cylinder as shown in Fig. 4(a). The shedding was successful but the consideration of the viscous diffusion and dissipation is still insufficient. This is another matter of controversy in the vortex method. The two main ways to account for the viscous effects are the deterministic method and the random walk method. The deterministic method was adopted for this test case. A cut-off function was used to account for the viscous diffusion. A core spreading method was used to account for the dissipation (sec. 2.1).

The corresponding velocity field represents in figure 4(b). The velocity field behave as follows. When circulation is zero, the uniform stream approaching from the right divides into two symmetric flows, one going over the cylinder, the other flowing under it. The two flows connect again downstream of the cylinder. The flow field is symmetric with respect to the *x*-axis. Two fluid particles immediately above and below the upstream stagnation point travel the same distance around the cylinder and then meet again at the downstream stagnation point. When circulation is increased, the stagnation points move towards the lower half of the cylinder so that the two companion fluid particles follow different routes to reach the downstream stagnation point.



(a) Vortex convection



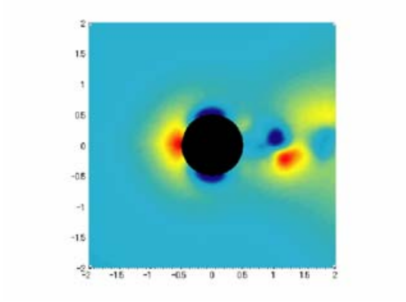
(b) Velocity field

Fig.4: Vortex convection and velocity field

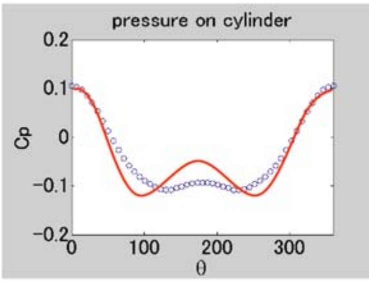
In particular the fluid particle that travels above the cylinder makes a longer route with respect to its companion, but it travels fast enough to arrive at the same time at the downstream stagnation point.

3.5 Pressure Analysis

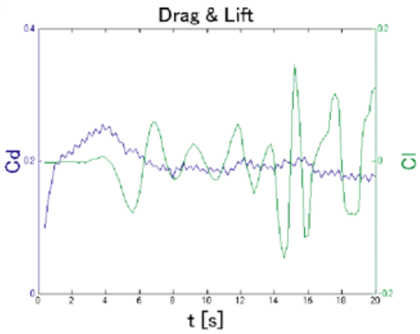
The cylinder is a bluff body whereas a wing that is well-oriented with respect to flow is a slender body. In reality (in the sense of a real viscous fluid) separation of the boundary layer with the formation of a wake will be unavoidable for the cylinder. The irrotational solution can not predict such phenomenon and the resulting flow field does not resemble the real flow around a cylinder.



(a) Pressure field



(b) Pressure distribution



(c) Drag and lift coefficients

Fig.5: Pressure Analysis

When circulation is absent, the pressure field is symmetric with respect to both the x and the y axes (Fig. 5(a)). On the stagnation points the excess pressure is positive (which means an action directed towards the body) while on the upper and lower points the excess pressure is negative (which means an action directed away from the body). The pressure field was calculated from the discrete vortex elements.

The pressure distribution is shown in Fig. 5(b). The solid line represent the analytic results and the point circle represent the simulated results. It can be observed that the both results have good agreement in this case. From the pressure distribution on the cylinder, the drag and lift were calculated. The area above the obstacle is at high pressure, while the area below the obstacle is at low pressure.

Drag coefficient (Cd) is a mechanical force generated by a solid object moving through a fluid. The lift coefficient (Cl) is a dimensionless coefficient that relates the lift generated by an airfoil/body, the dynamic pressure of the fluid flow around the body, and the platform area of the body. It may also be described as the ratio of lift pressure to dynamic pressure. Figure 5(c) represents the drag and lift coefficient of the flow around a circular cylider. Both results have good agreement compare to other experimental works (not shown here).

4. Conclusion

Vortex method has been successfully introduced to calculate the flow around circular cylinder. It was shown that, the vortex method is capable of calculating the accurate pressure distribution around a circular cylinder. The accuracy of the calculation can be improved by introducing more elements or by considering other physical models. These results will be used for calculating the deformation of the cylinder as a consequent work.

Acknowledgments

The author would like to thank Prof. Shinnosuke Obi, Keio University, Japan for the fruitful discussion and encouragement during this work. He would like to also thank Yoshida Scholarship Foundation for financial support to study in Japan.

References

[1] M. M. Zdravkovich, Flow around Circular Cylinder: Vol 1: Fundamentals, *Oxford University Press*, Oxford, UK, (1997)

[2] P. W. Bearman, and M. M. Zdravkovich, Flow around a Circular Cylinder near a Plane Boundary, *J. Fluid Mech.*, 89 (1978), 33-47.

[3] C. Lei, L. Cheng, and K. Kavanagh, Re-Examination of the Effect of a Plane Boundary on Force and Vortex Shedding of a Circular Cylinder, *J. Wind Eng. Ind. Aerodyn.*, 80 (1999), 263-286.

[4] P. Chatelain, Contributions to the Three-Dimensional Vortex Element Method and Spinning Bluff Body Flows, *PhD Thesis*, *California Institute of Technology*, USA (2005).

[5] G. S. Winckelmans, and A. Leonard, Contributions to Vortex Particle Methods for the Computation of Three-Dimensional Incompressible Unsteady Flows. *J Comput Phys*, 109 (1993), 247-273.

[6] T. K. Sheel, K. Yasuoka, and S. Obi, Fast vortex method calculation using a special-purpose computer. *Computers and Fluids*, 36 (2007), 1319-26.

[7] T. K. Sheel, R. Yokota, K. Yasuoka, and S. Obi, The study of colliding vortex rings using a special-purpose computer and FMM. *Transactions of the Japan Society for Computational Engineering and Science*, 2008 (2008), 20080003.

[8] A. J. Chorin, Numerical Study of Slightly Viscous Flow, *J. Fluid Mech.*, 57 (1973), 785-796

[9] K. Kuwahara, and H. Takami, Numerical Studies of Two-Dimensional Vortex Motion by a System of Point Vortices, *J. Phys. Soc. Japan*, 34:1 (1973), 247-253.

[10] A. Leonard, Vortex Methods for Flow Simulations, *J Comput Phys.*, 37 (1980), 289-335.

[11] L. F. Rossi, Resurrecting Core Spreading Vortex Methods: A New Scheme that is Both Deterministic and Convergent, *SIAM J. Sci. Comput.*, 17:2 (1996), 370-397.

[12] L. A. Barba, A. Leonard, and C. B. Allen, Advances in Viscous Vortex Methods-Meshless Spatial Adaption Based on Radial Basis Function Interpolation, *Int. J. Num. Meth. Fluids*, 47:5 (2005), 387-421.

[13] L. A. Barba, Discussion: "Three-Dimensional Vortex Method for Gas-Particle Two-Phase Compound Round Jet"(Uchiyama, T., and Fukase, A., 2005, ASME J. Fluids Eng., 127, pp. 32-40), *ASME J. Fluids Eng.*, 128 (2006), 643-645.

[14] M. J. Huang, Diffusion via Splitting and Remeshing via Merging in Vortex Methods, *Int. J. Num. Meth. Fluids*, 48:2 (2005), 521-536

[15] P. Degond, and S. Mas-Gallic, The Weighted Particle Method for Convection-Diffusion Equations. Part 1: The Case of an Isotropic Viscosity, *Math. Comput.*, 53:188 (1989), 485-507.

[16] P. Chatelain, and A. Leonard, Face-centred cubic lattices and particle redistribution in vortex methods, *J. Turb.*, 3 (2002), 046.

[17] S. Shankar, and L. V. Dommelen, A New Diffusion Procedure for Vortex Methods, *J. Comput. Phys.*, 127:2 (1996), 88-109.

[18] Y. Ogami, and T. Akamatsu, Viscous Flow Simulation Using the Discrete Vortex Model-the Diffusion Velocity Method, *Comput. Fluids*, 19 (1991), 433-441.

[19] J. R. Grant, and J. S. Marshall, Diffusion Velocity for a Three-Dimensional Vorticity Field, *Theor. Comput. Fluid Dyn.*, 19:6 (2005), 377-390.

[20] S. Koshizuka, H. Tamako, and Y. Oka, A Particle Method for Incompressible Viscous Flow with Fluid Fragmentation, *Comput. Fluid Dyn. J.*, 4 (1995), 29-46.

[21] R. Yokota, T. K. Sheel, and S. Obi, Calculation of Isotropic Turbulence Using Pure Lagrangian Vortex Method, *Journal of Computational Physics*, 226

T.K. Sheel received his PhD in Engineering from Keio University, Japan in 2008. He is an Associate Professor at the Department of Mathematics, Shahjalal University of Science & Technology, Sylhet, Bangladesh. His current research interest focuses on languages, compilers and tools for special-purpose computers with parallel programming and include OpenMP performance analysis and measurement, compiler analysis, and parallel algorithm. He is currently working as a postdoctoral researcher at Université Catholique de Louvain, Belgium. (2007), 1589-1606.