SIMULATION OF PREMIXED COMBUSTION FLOW AROUND CIRCULAR CYLINDER USING HYBRID RANDOM VORTEX

G. Heidarinejad[*]

Department of Mechanical Engineering, Tarbiat Modares University, P.O. Box 14115-143, Tehran, Iran
Tel +9821 82883361
gheidari@modares.ac.ir

S. Shahriarian

Department of Mechanical Engineering, Tarbiat Modares University, P.O. Box 14115-143, Tehran, Iran
Tel +9821 82883361
sshahriarian@yahoo.com

*Corresponding author

Abstract: This research describes unsteady two-dimensional reacting flow around a circular cylinder. The numerical solution combines random vortex method for incompressible two-dimensional viscous fluid flow with a Simple Line Interface Calculation (SLIC) algorithm for propagation of flame interface. To simplify the governing equations, two fundamental assumptions namely Low Mach Number and Thin Flame Thickness are used. Numerical and graphical representation of vorticity field, velocity variation on the wake axis, the effect of combustion on stream line pattern and location of vortex element at Reynolds numbers of 3000 and 9500 are discussed. The numerical results for the non-reacting flows fall within the range of the experimental measurements while the results of the reacting case are qualitatively following the physics.

Keywords: Random Vortex Method, Simple Line Interface Calculation (SLIC), Low Mach Number Model, Premixed Combustion, Reynolds Number

چکیده   در این تحقیق جریان احتراقی ناپایدار دوبعدی در اطراف یک استوانه مدور بررسی می‌شود. روش عددی به کار رفته ترکیبی از روش گردابه‌های تصادفی برای حل جریان سیال لزج دو بعدی و الگوریتم SLIC برای شبیه سازی پیشرفت سطح شعله می‌باشد. برای ساده سازی معادلات حاکم دو فرض اساسی عدد ماخ کوچک و ضخامت شعله نازک اعمال می‌شود. نمایش عددی و تصویری میدان چرخش، تغییرات سرعت بر روی محور جریان برگشتی پشت استوانه، اثرات احتراق بر روی خطوط جریان و موقعیت المان‌های گردابه‌ای در اعداد رینولدز ۳۰۰۰ و ۹۵۰۰ نشان داده شده‌اند

1. INTRODUCTION

Most flames in practical applications burn under turbulent conditions to enhance the rate of heat release and to meet the loading requirements in small size combustors. This study focuses on the continuous combustion of premixed gases where the fuel and oxidizer were well homogenized prior to burning. To stabilize the burning process, flow separation induced to maintain a continuous recirculation of products, which, in turn provides the energy, required to sustain the ignition of reactant. Turbulent combustion involves a complex set of interactions between essentially unstable fluid flow phenomena and transient exothermic chemical reactions that produce significant increase in the specific volume of the following mixture. In general, these interactions influence the structure of the fluid flow as well as the progress of chemical process and lead to a highly un-tractable situation, especially when basic mechanism of both phenomena represent substantial challenges in their analysis. Random

Vortex Method is an effective method for solving the vorticity field [1].

Chorin used this method for solving the Navier-Stockes equations in 1973 for the first time [2]. Chorin and Ghoneim used Random Vortex Method for modeling turbulent flow in a combustion tunnel in 1981[3]. Beale and Majda worked on the convergence of this method in 1985[4]. Ghoniem et al. introduced Transport Element Method in 1986 to extend the method to transport of scalar and energy equations.

In the current work, random vortex method and a modified SLIC algorithm were combined to compute the flow field and the flame propagation. Flow around a circular cylinder used as the test problem and the effects of Reynolds number on burning and combustion were analyzed.

2. GOVERNING EQUATIONS

In this method, it is essential to use a set of differential equations that describe both fluid flow and flame propagation in an open domain. Governing equations for a combustion flow are complicated so the following assumptions are used:

1. First order, irreversible, unimolecular, single step reaction.
2. Arrhenius rate reaction expression as a source term.
3. Burnt and unburnt gas mixtures behave as perfect gas with the same molecular weight, same constant heat conductivities and specific heats.
4. The thin flame thickness.
5. A Low Mach number model [5, 6].

$$\frac{D\omega}{Dt} = \partial\omega / \partial t + u.\nabla\omega = (1/\text{Re})\nabla^2\omega \tag{1}$$

$$\omega = \partial u_2 / \partial x - \partial u_1 / \partial y \tag{2}$$

$$u = v + u_{\omega'}, \quad v = \nabla\phi, \quad u_\omega = (\partial_2\psi, \partial_1\psi) \tag{3}$$

$$\nabla^2\psi = -\omega \tag{4}$$

$$\nabla^2\phi = \frac{QS_u}{T_u}\delta(x - x_f) \tag{5}$$

$$\frac{\partial F}{\partial t} + u_u.\nabla F = S_u|\nabla F| \tag{6}$$

2.1. Numerical Solution A hybrid numerical scheme which is used in this study combines the Random Vortex Method for incompressible, viscous and two dimensional flow with the Simple Line Interface Calculation (SLIC) algorithm for the propagation of the flame interface [7, 15, 16]. Two solutions are coupled by:

1. The advection part of the flame motion, which uses the velocity field computed by random vortex method used to propagate the flame interface.
2. The expansion field due to the heat release resulting from burning at the flame front, which contributes to the flow field computation by the random vortex method.
3. A third coupling mechanism, in which the generation of vorticity at the flame front is neglected.

2.2. Fluid Flow-Random Vortex Method The random vortex method solves the vorticity transport equation instead of momentum equation. In addition, conservation mass equation and the identity $\nabla\rho \times \nabla P = 0$ are used.

In random vortex method solution, the vorticity field is discretized among a finite number of vortex elements whose configuration is updated every time according to the flow equations. A Lagrangian formulation is employed for advection expressed in terms of a set of coupled ordinary differential equations while an appropriate random walk algorithm simulates diffusion. Numerical diffusion is minimized by avoiding Euleran discretization of computational grid. Therefore, numerical scheme is grid free and the computations are self adaptive since vortex elements move to capture zones of large velocity gradients with concentration of vorticity [8, 17]. In the following, we proceed to demonstrate the application of the method to the fluid flow equations on the computational domain listed below. The governing equations are as follow:

$$\frac{D\omega}{Dt} = \partial v / \partial t + u.\nabla\omega = (1/\text{Re})\nabla\omega$$

$$\omega = \partial u_2 / \partial x - \partial u_1 / \partial y$$

$$u = v + u_{\omega'}, \quad v = \nabla\phi, \quad u_\omega = (\partial_2\psi, \partial_1\psi)$$

$$\nabla^2\psi = -\omega$$

The solution of the flow equations proceeds differently to region as follow:

1. Out of wall region
2. Wall region

Also, in random vortex method, each time step is divided into two fractional steps.

During each fractional step one of the advection or diffusion mechanisms is considered.

2.3. Out of Wall Region   The method of viscous splitting is applied to the vorticity transport equation in this region. The governing equations are solved, and the vorticity field is updated at every time step, using two fractional steps: an advection step and diffusion step. In the former, the vorticity field is updated according to the inviscid version of the vorticity transport equation, namely:

$$\frac{\partial \omega}{\partial t} + \mathbf{u}.\nabla \omega = \mathbf{0} \tag{7}$$

Then, in the diffusion step, vorticity field is updated solving related equation:

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re}\nabla^2 \omega \tag{8}$$

This is equation 1 but with u=0.
Combining the two solutions provides a solution for Eq. 1 in the out of wall region.

2.4. Advection   Inviscid term of the transport equation is written as follows [9]:

$$\frac{\partial \omega}{\partial t} + \mathbf{u}.\nabla \omega = \mathbf{0}$$

According to Eq. 7 the vorticity advects along particle paths. Thus, vortex blobs and vortex sheets advects according to Eq. 7. Given the locations X_i^n at time $t = n\Delta t$, one may compute the intermediate locations due to the advection alone, $X_{i,adv}^{n+1}$ at time $t = (n+1)\Delta t$. In this research Euler's method is used for time integration:

$$X_{i,adv}^{n+1} = X_i^n + \Delta t u_i$$
$$Y_{i,adv}^{n+1} = Y_i^n + \Delta t v_i$$

2.4. Diffusion   The random walk algorithm is based on the fact that the Green functions of one dimensional form of Eq. 8 given by the following equation:

$$G(x,t) = (Re/4pt)^{1/2} \exp(-x^2 Re/4t)$$

is identical to the probability density function of a Gaussian random variable h with a zero mean and a standard deviation of $\sigma = \sqrt{\dfrac{2\Delta t}{Re}}$, namely:

$$P(h,t) = (1/2s^2 p)^{1/2} \exp(-h^2/2s^2)$$

So, the diffusive transport of vortex elements is simulated stochastically by adding to their convective motion an extra displacement drawn from a Gaussian population with zero mean and a standard deviation σ. The total transport of vortex elements obtained by adding the two fractional displacements is:

$$X_i^{n+1} = X_{i\,adv}^{n+1} + h_i \tag{9}$$

where, $h = (h_1, h_2)$ is the random displacement vector, and $X_i^{n+1} =$ is the new location of blob at t= (n+1) D t.

Note that wall region diffusion takes place only in the direction normal to the wall. Without diffusion term, the sheets would never leave the boundary, since u= (0, 0) at the wall, by construction. Therefore, Eq. 9 will change to the following equations:

$$X_i^{n+1} = X_{i,adv}^{n+1}$$
$$y_i^{n+1} = y_{i,adv}^{n+1} + h_i$$

where h_i is the random jump.

3. Combustion Solution

Now, the numerical description of the motion of the flame interface is considered. The motion of the flame can be split into two components: advection and burning. Advection is simply the motion of the flame interface as a passive material surface with the local flow velocity. Burning, on the other hand, is the motion of the flame into the reactants with respect to an observer moving with

the reactants immediately ahead of the flame. The reactants burn, and are converted into products. As a result of the exothermic combustion reaction, heat is generated at the flame front. The products of combustion are at a higher temperature, and lower density, than the reactants. The fuel and air mixture expands upon burning. In thin flame model, this expansion of the fluid crossing the flame zone occurs only at the flame interface. The flame expansion field discretized here into a finite number of two-dimensional point sources, which are distributed along the flame interface. The implementation of these sources is described below.

3.1. SLIC Algorithm Various methods exist for the numerical description of the shape and evolution of interfaces between fluids. The algorithm implemented in this research is based primarily on the SLIC (Simple Line Interface Calculation) method introduced by Noh and Woodward [11], and later implemented by Chorin [12] for flame propagation and advection with increased spatial resolution. This method uses a finite set of horizontal and vertical straight-line segments on a two dimensional rectangular grid to construct the flame interface, and an alternating direction scheme to advect the interface.

The information provided by SLIC includes the amount of products, or reactants, in each cell of the computational grid, and, for cells containing the flame interface. The flow field is divided for this purpose into cells. The fraction of volume (V) occupied in a given cell by the burned medium is expressed in terms of a number [13]:

$$f = \frac{V_b}{V_c} \qquad (10)$$

Thus, f=0.1 means that there is, respectively, either unburnt or burnt medium in the cell, while fractional values of f indicate cells containing the interface. The particular geometry of the interface is deduced, depending on the f-number of neighboring cells. In this connection, as illustrated in Figure1, proper provisions are included in the algorithm for four possibilities:

a) Vertical interface
b) Rectangular corner
c) Horizontal interface
d) Neck

(a)   (b)

(c)   (d)
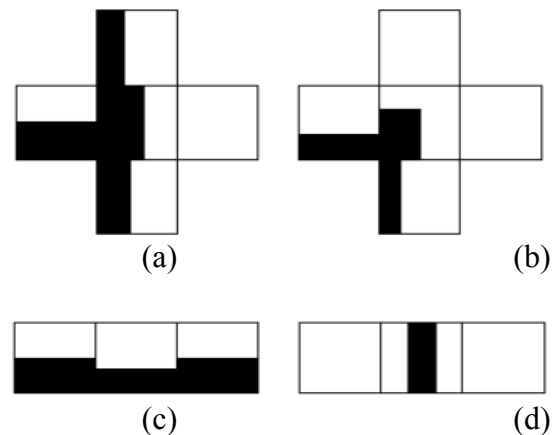
Figure1. Elementary components of an interface recognized by the algorithm a) Vertical interface, b) Rectangular corner, c) Horizontal interface, d) Neck

3.2. Advection We begin by discretizing the domain using a non uniform grid with cell size given by $\Delta r \& \Delta \theta$. As shown in Figure 2, each cell is assigned a fractional index f_{ij} which indicates the amount of products in the cell. Hence, $f_{ij} = 1$ for a cell full of products, $f_{ij} = 0$ for a cell full of reactants, and $0 < f_{ij} < 1$ for fractional cells, which contain some reactants and some products. These fractional cells contain the flame interface. The orientation of the interface inside a cell is determined by using SLIC algorithm.

After the interface is located, its velocity is evaluated by interpolating between the velocities on both sides of the cell. The interface velocity is used to move the interface.

The velocity due to convection is related to the stream function which is the solution of Poisson's equation:

$$\nabla^2 \Psi = -\omega(\mathbf{x})$$
$$u = -\frac{\partial \Psi}{\partial y}, v = \frac{\partial \Psi}{\partial x}$$

3.3. Wall Region In the context of the Random Vortex Method, the "wall region" refers to a very thin region extending only a few σ's (Random walk standard deviations) from the domain walls, where the discretization of the vorticity field uses vortex sheets instead of blobs, the latter being used in the bulk of the flow domain. Vorticity is
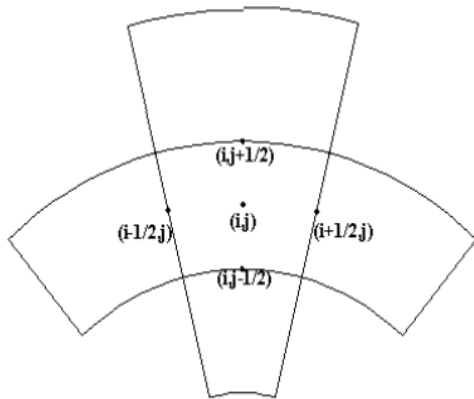
Figure 2. polar grid

generated at the walls to satisfy the no-slip boundary condition. Chorin used scheme of vorticity generation in the random vortex method to satisfy the viscous boundary condition at the solid walls [8]. The vorticity generated along the wall is discretized into elements separated by a distance "h". To improve desired resolution, required circulation generated at each wall is distributed among several elements (sheets) such that each has a certain maximum circulation. These sheets leave the wall by diffusion, to become part of the out of wall vorticity field at later times, when they are converted to vortex blobs of appropriate core radius to satisfy compatibility between the out of wall and wall regions. The wall region (also referred to below as the sheet region) has a thickness of Δ_S.

By the nature of the solution in the wall region, we will use the wall velocity computed from out of wall flow calculation, as specified for the boundary layer equations. We will not need to use the Helmholz decomposition, Eq. 3 of the velocity field. It will be sufficient to write down the curl of the Navier-Stokes equations and the continuity equation as starting points. Further, since Δ_S is typically much less than the cell size used for the flame propagation algorithm, we will not consider the expansion field of any flame source in the sheet region, rather the effect of the expansion field will be presented via U.

3.4. Combustion   The motion of the flame in the direction normal to itself due to combustion is calculated in the exact manner as advection but

the normal burning speed is used in all possible directions of propagation instead of the advection velocity, since the direction of propagation of the flame due to combustion is not known a priori, S_u gives all possible directions and the actual flame motion corresponding to all directions is calculated, thus the flame speed is taken as [14]:

$$S_f = (S_u \cos\theta_i, S_u \sin\theta_i)$$

where $\quad \theta_i = \dfrac{(i-1)\pi}{4}$

And i=1, 2 … 8.
According to Huygens principle, the actual flame propagation is the maximum advancement in all directions, thus f is updated as:

$$f(t + \Delta t) = \underset{0 \le i \le 8}{Max} f_i$$

where $\quad f_0 = f(t)$

3.5. Exothermicity   Figure 3 explains the dynamic effect of burning due to the increase in specific volume in which a particle moves away from the flame front by a distance of $2u_s$ after burning. To account for this effect in the two-dimensional calculations of the velocity field, changes in f duo to flame advancement in the combustion step Δf_c are used to calculate the strength of the volumetric sources according to the following equation:

$$\Delta = 0.5(\upsilon - 1)r\Delta r\Delta\theta \frac{\Delta f_c}{\Delta t_c}$$

Where υ is the specific volume ratio across the flame. For each cell with $\Delta f > \Delta f_{min}$, where Δf_{min} is negligible change in volume, a source blob is generated and placed at the center of this cell. The core size of this blob is:

$$r_0 = \frac{\Delta}{2\pi u_s}$$

Where $u_s = 0.5(\upsilon - 1)S_u$ is the maximum velocity generated by the flame propagation.
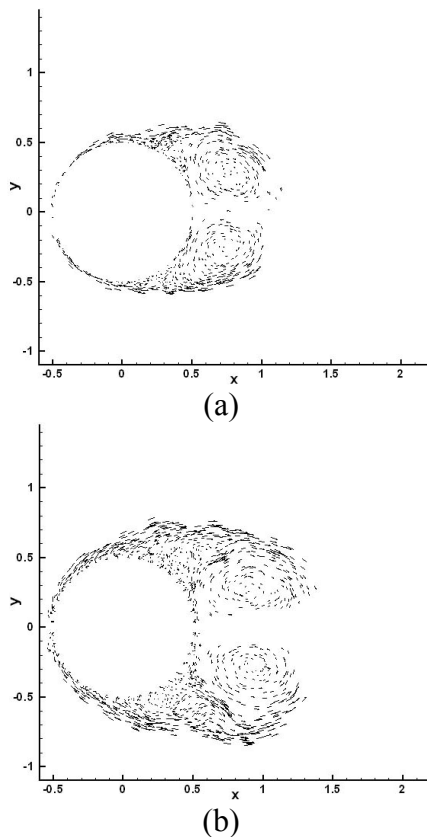
4. RESULTS

In Figure 3, vortex locations around a circular cylinder are shown at $t = 2$. Case (a) shows non-reacting case with $S_u = 0$ and, case (b) shows the reacting case with $S_u = 0.02$.

As shown in Figure 3 (b) in the reacting case, vortex generation is more than Figure 3 (a) which is for non-reacting case.
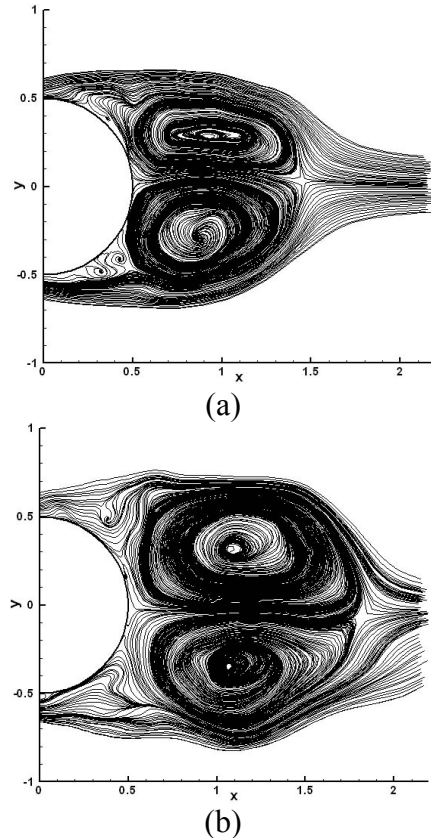
Figure 4 shows the streamlines behind the cylinder at Re =3000.

As shown in Figure 4 (b), the recirculation zone in the reacting case is stronger than non-reacting case as shown in Figure 4 (a).

The u-velocity on the x-axis behind the cylinder is plotted in Figures 5 and 6 for non-reacting case and reacting case, respectively at Re =3000. The portion of the curve that is positive corresponds to the positive velocity and the negative portion is back flow on the wake axis. Note that the rear stagnation point is situated at point (0.5, 0.0) and

(a)

(b)

Figure 4. Plot of streamline around a cylinder at t=3 , (a) non-reacting, (b) reacting

the diameter of the cylinder is 1. Velocity on the wake axis for calculations is in satisfactory agreement with known experimental and numerical results. As shown in Figure 6, the minimum value of the velocity in the reacting case at t<2 second is less than non-reacting case. At t≥2 the flow field is more unsteady than non-reacting case, so the minimum value of the velocity in the reacting case is more than non-reacting case.

The u-velocity on the x-axis behind the cylinder is plotted in Figures 7 and 8 for non-reacting case and reacting case, respectively at Re =9500. A comparison between Figures 5 and 7 indicates that the minimum velocity at Re =9500 is less than its value at Re=3000. Also, by increasing the Re number of the flow field, the recirculation zone will be decreased and after t =2 seconds the flow field will be unstable.

Plots of the flame front are shown in Figures 9 (a) for Re=150 and 9 (b) for Re=3000 and as shown

(a)

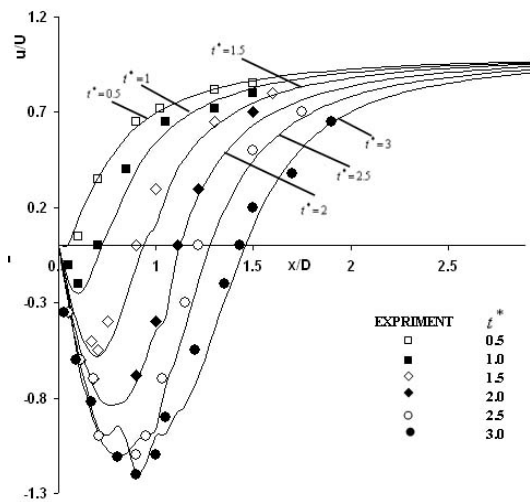
(b)

Figure 3. Plot of the velocity vector field, (a) without combustion, (b) with combustion

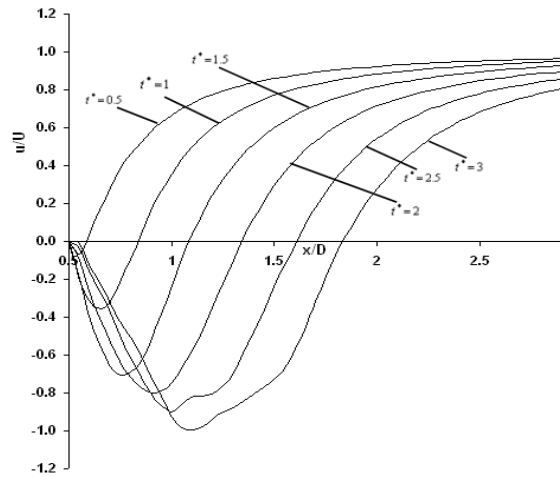**Figure 5.** Velocity on the wake axis, Re=3000

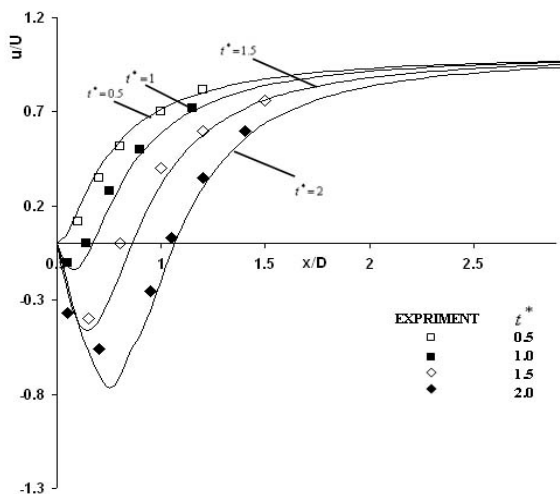**Figure 6.** Velocity on the wake axis with combustion, Re=3000

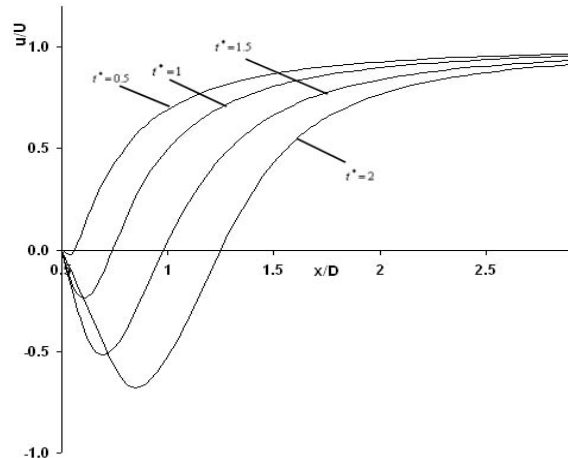**Figure 7.** Velocity on the wake axis, R=9500

Figure 8. Velocity on the wake axis with combustion, R=9500

the Re Number value imposes some changes on the flame front geometry. The plots identify the flame as the interface between blue area, depicting reactants, and red area, representing the products.

In Figure 10 the vortex shedding is shown in accordance with time. After initial time steps, main vortexes began to dissipate periodically. Those vortexes developed the Von Karman vortex street. Additionally, it is shown that in the reactant case stronger eddies are generated behind the cylinder.

5. CONCLUSIONS

The model used is based on the Random Vortex Method of computing the turbulent flow field. The method is grid free and avoids numerical diffusion; therefore, it is suitable for modeling the intrinsic physical properties of flows at large Reynolds numbers. It can analyze large gradients produced by the small eddies of turbulent motion, and it allows perturbations to grow into possible instabilities without damping or dissipation. The combustion of premixed gases is idealized by the
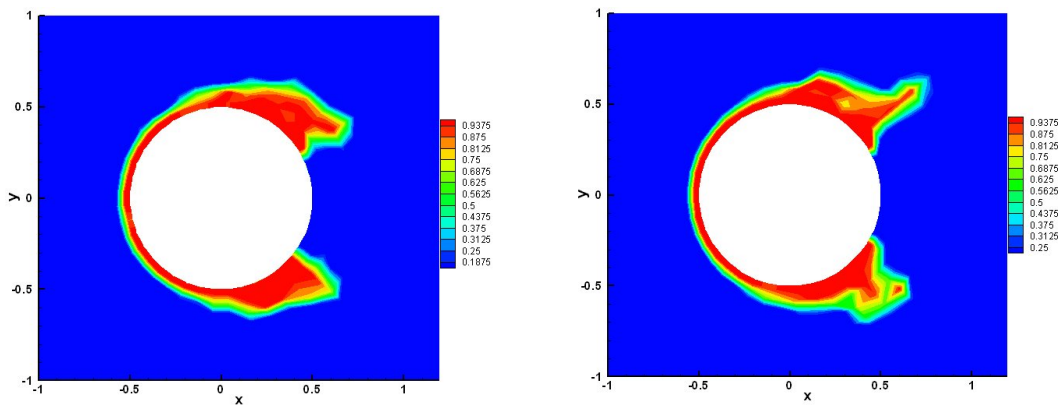
Figure 9. Flame front contours, represented by interface between reactants and products, for $S_u = 0.02$ at different values of Re numbers. (a) Re=150 (b) Re=3000
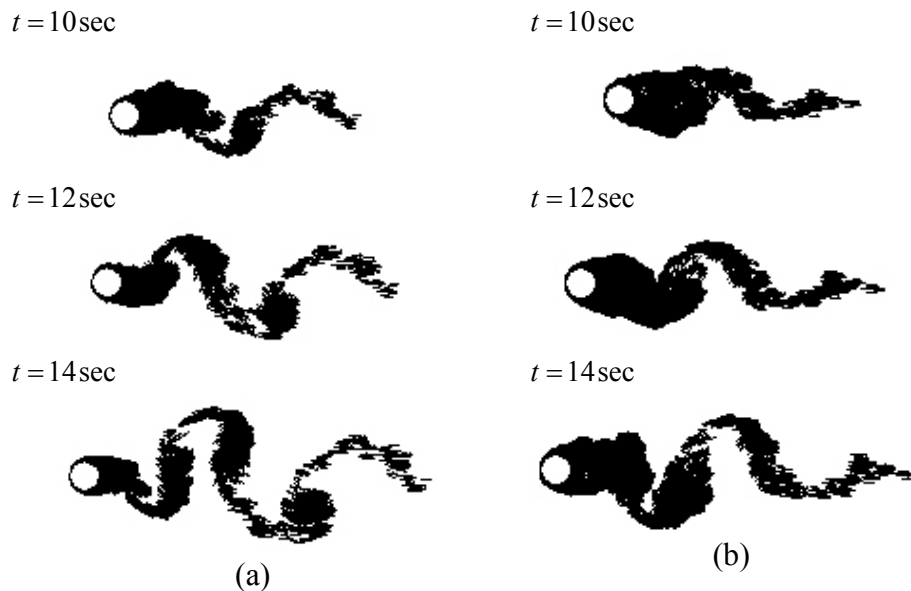
Figure 10. Vortex shedding (a) non-reacting case (b) reacting case

propagation of a flame interface, located between reactants and products, due to convection and self-propagation. The expansion of the flow, across the flame, due to the exothermicity of the reaction is taken into account using a distribution of volumetric sources within the burning zone. The formation of large scale eddy structures in the turbulent flow down-stream of the cylinder persists in the reacting case but also the flame front almost surrounds the vorticity field and thus produces a situation in which the flame propagates along a streamline instead of normal to

it. The formation of a recirculation zone behind the cylinder provides a continuous source of ignition by enhancing the contact between the products and the reactants.

6. REFERENCES

1. Barba, L. A. "Vortex method for computing high-Reynolds number flows: increased accuracy with a fully mesh-less formulation", California institute of technology Pasadena, thesis, (1994).
2. Chorin, A. J. "Numerical study of slightly viscous flow", *Journal of fluid mechanics*, Vol. 57, (1973), 785-796,

3. Ghoniem,A.F., Chorin, A. J. Oppenheim, A. K. "Numerical modeling of turbulent flow in a combustion tunnel", *Journal of comp. fluid mech.* (1981), 303-325.

4. Beale, J. Majda, T. "A. High-order accurate vortex method with explicit velocity kernels", ***Journal of computational physics***, Vol. 58, (1985), 188-208.

5. Lakkis,L. A.F. Ghoniem, "Axisymmetric vortex method for low-mach number, diffusion – controlled combustion", *J. comp. phys*, Vol.184, (2003), 435-475,

6. Muller, B., "Low Mach number asymptotic of the Navier-Stokes equations and numerical implications", Von Karman Institute for Fluid Dynamics, (1999)

7. Ghoniem, A.F., "Analysis of flame deformation in turbulent field, effect of Reynolds number of burning rates", American Institute of Aeronautics & Astronautics, (1985)

8. Chorin, A. J., "Vortex Sheet Approximation of boundary layers", ***Journal of computational physics***, Vol. 27, (1978), 428-4480.

9. Anderson, C. R., "A vortex method for flows with slight density variations, ***Journal of computational physics***, Vol. 61, (1985), 417-444.

10. Cheer, A.Y., "Unsteady separated wake behind an impulsively started cylinder in slightly viscous fluid", ***Journal of fluid mechanics***, (1988), 485-505,

11. W. F. Noh, P. Woodward, SLIC, Simple line interface calculation, 15th international conference on numerical method in fluid dynamics, Springer-verlag, Berlin, (1976), 330-339,

12. Chorin, A. J. "Flame advection and propagation algorithms", ***Journal of computational physics***, Vol. 35, (1980), 1-11,

13. Hirt, C. W., Nichols, B.D., "Volume of fluid method for the dynamics of free boundaries", ***Journal of computational physics***, Vol. 39, (1981), 201-225.

14. Calzada, M. E., "Numerical study of flame propagation in boundary layers", ***Journal of pure and applied mathematics***, (1997), 1-7,

15. Shahriarian, S., Heidarinejad, G., Delfani, Sh. "Investigation of the Reynolds Effects on the Premixed Combustion Flow around the Circular Cylinder", Presented in the 11th Fluid Dynamics Conference, in 2008

16. Shahriarian, S., Heidarinejad, G., Delfani, Sh., "Simulation of the premixed combustion flow around the circular cylinder at initial times for moderate and high Reynolds", Presented in 16th International Conference on Mechanical Engineering; ISME 2008

17. Heidarinejad, G., Delfani, S., "Direct numerical simulation of the wake flow behind a cylinder using random vortex method in medium to high Reynolds numbers", ***International Journal of Engineering***, Vol. 13, NO. 3, (2000), 33-50,