Volume 5, Issue 12, December 2015 ISSN: 2277 128X
International Journal of Advanced Research in Computer Science and Software Engineering
Research Paper
Available online at: www.ijarcsse.com

Numerical Modeling bi-jet at the Bottom of a Channel

M. Loukili, S. Mordane
Polymer Physics and Critical Phenomena Laboratory Sciences Faculty Ben M.sik, P.O. Box 7955, university hassan II, Casablanca Morocco

Abstract: **In** ***this work, we focus on numerical solutions of the Navier Stokes equations. These equations are nonlinear and in the case of strong re-circulations of the velocity field near the wall, these nonlinearities dominate and define all the physical processes associated with the problem. To illustrate these types of nonlinearities, we chose the b-jet problem from the bottom in an open channel of an incompressible viscous fluid high Reynolds number. The numerical solutions of the Navier Stokes equations are made by the decomposition method of the field into two sub domain; the first is treated by finite differences high accuracy, and the second by the particle method. We present the results for analyzing the validity of the proposed numerical approach in this work and also for refining the description of the flow particularly in the vicinity of the diffuser as the significance of this problem for any industrial application.***

Keyword---Navier Stokes equations. Particular Methods, Method of Finished Differences, Decomposition of the Domain, bi-jet, nonlinearities

I. INTRODUCTION

In this study, we present a complete numerical solution of flow map generated by a bi-jet emitted from the bottom in an open channel. The viscous fluid is considered incompressible. The resolution was made by the method of domain decomposition, digital processing of the equations modeling the behavior of flowing fluid mechanics, and can introduce complexities to the nature of these equations. Indeed, these equations are generally in the form of a balance of momentum and mass: Temporal variation and convection are balanced by diffusion and source term. The Reynolds number which is a number that shows the ratio between convection and diffusion can characterize the non-linearity of such material balance equations. Indeed, for many of the Reynolds convective term dominates over diffusion which makes delicate digital processing. The objective is to study a bi-jet representing strong recirculation of the velocity field near the wall view of the importance of this problem for technical and industrial application.

This work concerns the study of dynamic phenomena present when a vortex structure moves near a solid wall. A numerical simulation is performed for the configuration of two-dimensional of a viscous incompressible fluid. This is to characterize the instabilities present in the flow to moderate Reynolds number between 100 and 1000. Interaction mechanisms are then described. It seeks to highlight the emergence and development of the boundary layer in the near-wall region, creating vortices, the winding of these structures, and their eventual ejection.

II. STATEMENT OF THE PROBLEM

Consider a two-dimensional unsteady flow of an incompressible viscous fluid of positive buoyancy in a limited physical area by two imaginary planes, a flat free surface (SL) and an impermeable horizontal bottom (F). On the bottom was placed two broadcasters of the same diameter which immersed the pollutant concentration C.

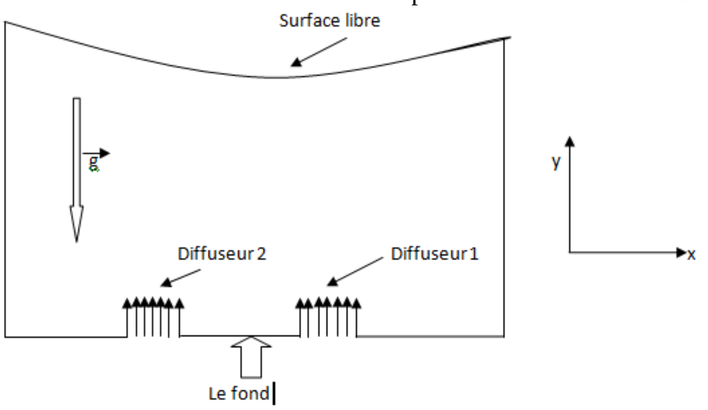

Fig1: Geometric configuration of the field of study.

This problem can be formulated by the Navier-Stokes equations and mass transport equation for the two-ply. In terms of vortex function$\boldsymbol{\omega}$, current function $\boldsymbol{\psi}$ and concentration C, these equations are written as follows:

$$\frac{\partial \omega}{\partial t} + (\vec{U} \cdot \vec{\nabla})\omega = \frac{1}{R_e}\Delta\omega + \frac{1}{Fr^2}\left[\vec{\nabla} C \wedge \frac{\vec{g}}{\|\vec{g}\|}\right] \cdot \vec{k} \quad (1)$$

$$\Delta\psi = -\omega \quad (2)$$

$$\vec{U} = \vec{\nabla}\Lambda(-\psi\vec{k}) \quad (3)$$

$$\frac{\partial C}{\partial t} + (\vec{U} \cdot \vec{\nabla})C = \frac{1}{R_e P_r}\Delta C \quad (4)$$

With the following notations

$\vec{\nabla}$ and Δ are respectively the gradient operator and the Laplacian operator

t : is the time variable..

$\vec{k}$:is c the directly perpendicular vector in the plan of the flox.

$\vec{U}$: velocity vector

C : is the concentration of the pollutant

ψ : is the function of current

ω : is the vortex function, such $\vec{\omega} = \omega\vec{k} = \vec{\nabla}\Lambda\vec{U}$

$\vec{g}$: Acceleration of gravity.

$\Re e, \Pr, Fr$: Represent respectively the numbers of: Reynolds, Prandtl and Froude.

This formulation must be completed by the given initial conditions and boundary conditions, we will now clarify:

II.1. Initial conditions

At the initial instant, all variables are zero except on the diffuser where C = 1.

II.2. Boundary condition

In terms of boundary conditions, there are two types of conditions: physical conditions which will be imposed on the variable speed and the particular concentration on the bottom, and those necessary to the closing of the problem are those which impose on the vortex vector on the solid walls and on all open borders on variables.

- On the lateral boundaries, we can write that

$$\frac{\partial C}{\partial n} = 0 \text{ , } \frac{\partial(\vec{U} \cdot \vec{n})}{\partial n} = 0 \text{ , } \vec{U} \cdot \vec{\tau} = 0 \text{, } \frac{\partial \omega}{\partial n} = 0$$

With $\vec{n};\vec{\tau}$ respectively normal vectors and tangent to the border.

- It is assumed that the free surface is flat, there will therefore be:

$$\frac{\partial C}{\partial n} = 0; \ \frac{\partial(\vec{U} \cdot \vec{\tau})}{\partial n} = 0 \text{, } \vec{U} \cdot \vec{n} = 0 \text{, } \omega = 0$$

- In the channel bottom :

On the one hand, on each broadcast, a prescribed concentration in a uniform manner with a uniform output rate:

$$\vec{U} \cdot \vec{n} = 1; \ C = 1. \ \omega = 0$$

- Moreover, far from the diffuser, we impose a zero flow and zero speed.

$$\frac{\partial C}{\partial n} = 0 \text{ ; } \vec{U} = \vec{0}$$

- When the boundary condition for the vorticity, it is calculated from the Poisson equation:

$$\frac{\partial^2 \psi}{\partial n^2} = \omega$$

III. PRESENTATION OF METHOD

The application of the decomposition method of the domain for a numerical solution of the Navier-Stokes equations is done in several steps:

III.1. Step 1: Solving laplacienne equation

The resolution is based on the use of hermitian relations 4th order, involving the function and its second derivative at three points. In any node within the domain, we write the following algebraic equations, expressed not with uniform [2]:

$$\left(\frac{\partial^2 \psi}{\partial^2 x}\right)_{i,j} + \left(\frac{\partial^2 \psi}{\partial^2 y}\right)_{i,j} = \omega_{i,j}$$

$$\frac{12}{h^2}(\psi_{i+1,j} - 2\psi_{i,j} + \psi_{i-1,j}) = \left(\frac{\partial^2 \psi}{\partial x^2}\right)_{i+1,j} + 10\left(\frac{\partial^2 \psi}{\partial x^2}\right)_{i,j} + \left(\frac{\partial^2 \psi}{\partial x^2}\right)_{i-1,j}$$

$$\frac{12}{h^2}(\psi_{i,j+1} - 2\psi_{i,j} + \psi_{i,j-1}) = \left(\frac{\partial^2 \psi}{\partial y^2}\right)_{i,j+1} + 10\left(\frac{\partial^2 \psi}{\partial y^2}\right)_{i,j} + \left(\frac{\partial^2 \psi}{\partial y^2}\right)_{i,j-1}$$

The result is a system of order$3(N_x - 1)(N_y - 1)$, where N_x, N_y each node numbers are according to x and y. This system is solved by a method for alternating directions implicit (ADI). The analysis and details of the resolution are explained in reference [3].

The given boundary conditions on the edges to calculate the first derivatives throughout the internal field by the following relationships:

$$\frac{3}{h^2}(\psi_{i+1,j}-\psi_{i-1,j}) = \left(\frac{\partial\psi}{\partial x}\right)_{i+1,j} + 4\left(\frac{\partial\psi}{\partial x}\right)_{i,j} + \left(\frac{\partial\psi}{\partial x}\right)_{i-1,j}$$

$$\frac{3}{h^2}(\psi_{i,j+1}-\psi_{i,j-1}) = \left(\frac{\partial\psi}{\partial y}\right)_{i,j+1} + 4\left(\frac{\partial\psi}{\partial y}\right)_{i,j} + \left(\frac{\partial\psi}{\partial y}\right)_{i,j-1}$$

III.2. Step 2: Resolution transport equations on ω and C

Transport equations are solved with a diagram of the second order, combined with the ADI method [1-5]. The ADI method is used is that introduced by Peacemen Rachford [4]. The terms of transport of the vortex vector and concentration are treated on the basis of a discretization offset upstream - downstream depending on the sign of the speed. This procedure allows linking the convective terms to the directions of speed and letting the diffusive terms act in all directions. The preponderance of the diagonal of the matrix associated with the linear system is strengthened. This method is robust and we chose it because of the experience gained in its application.

III.3. Step3 Subdivision of domaine Ω

The bi-jet is the bottom (Domain Ω1) and the flow is dominated by the effect of the walls. The resolution selected in this subdomain is the method of finite differences [3]. It is the method Eulerienne which possesses the advantage to facilitate the writing of the conditions in the limits on the solid walls . Far from walls (Domain Ω 2) (Figure 2), the flow is external in large number of Reynolds and in big recirculation of speed. In this domain, we use the particular method [4],It is the lagrangienne method which consists to discretize the transportable variables by means of a number of particles which will be followed in their movement.. Between both sub-domains the interface is the place of transmission the results of resolution by a method which uses a meshing (the method differences finis on Ω 1) and a method without meshing (the particular method on Ω 2), requires special treatment. A first algorithm is to transmit information according to the sign of the velocity normal to the boundary between the two sub domains Ω1, Ω2 without recovery domain. This is an explicit approach but is used here for the implementation of the method presented in this work .for all the domain of Study by the finite difference method.

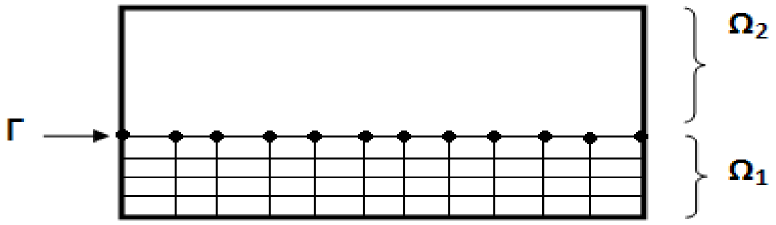

Fig 2. Subdivision du domaine Ω.

IV. RESULTS AND DISCUTION

On the bottom of a rectangular channel (length = 6 m, depth L = 4 m) was placed a 1m diameter diffuser. The area of study is limited by two lateral boundaries, a flat free surface and a raincoat horizontal bottom. (Figure 1). The equations to be solved are the equations 1-4. The chosen dimensionless numbers are F_r= 18, P_r= 1.5. The Reynolds number is a parameter study. In applying the decomposition method of the field, Ω is partitioned into two sub areas (Figure 2). In the subdomain Ω1, we choose to solve the equations 1-4 by a finite difference method. In Ω2, we solve the problem by particle method. This is justified by the aspect of flow. Indeed, in this subdomain, it is the aspect of high recirculation speed dominates and the finite difference method is less suited to deal with this aspect of the movement. The particle method is a Lagrangian method adapted to process the convection problems in the turbulent regime (& CHOQUIN HUBERSON, 1988). The coupling is made by a method of particle mesh.

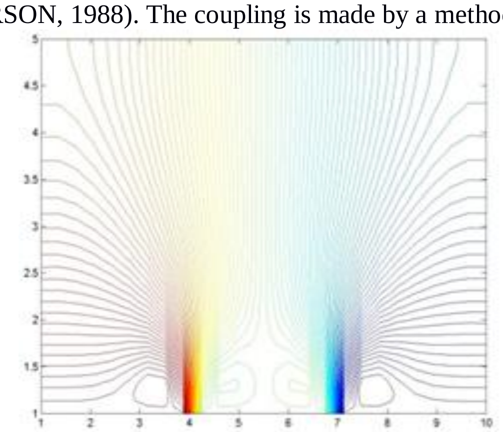

Fig 3a: ISO - values of the of the current function by direct calculation Witt t=1, Re=100

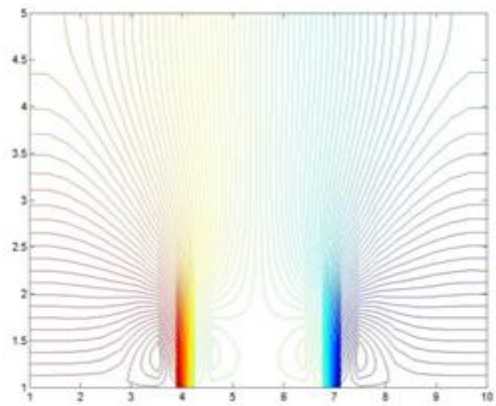

Fig 3b: ISO - values of the current function by direct calculation Witt t=1, Re=1000

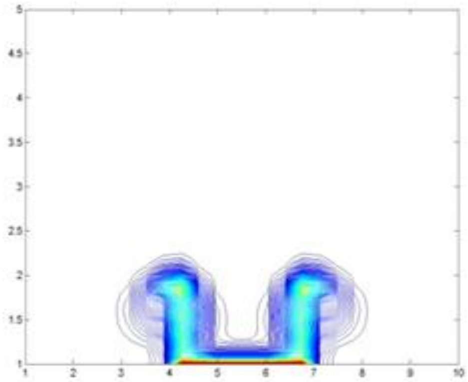

Fig 4a: ISO - values of the of the Concentration by direct calculation Witt t=1, Re=100

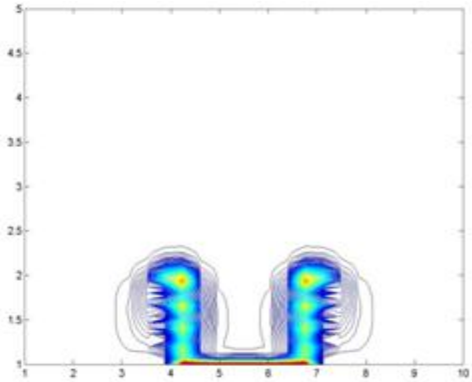

Fig 4b: ISO - values of the Concentration by direct calculation Witt t=1, Re=1000

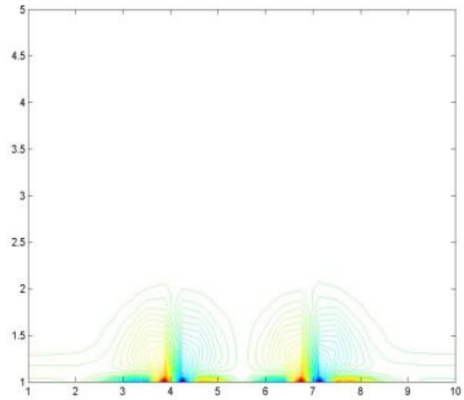

Fig 5a: ISO - values of the of the vortex function by direct calculation Witt t=1, Re=100

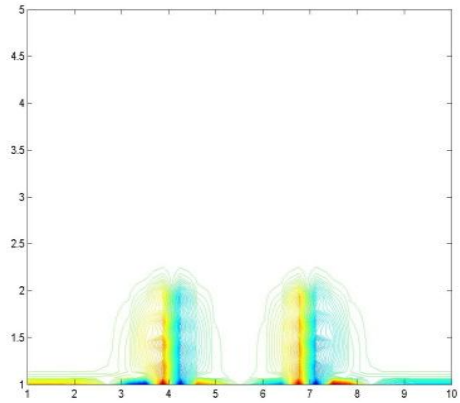

Fig 5b: ISO - values of the vortex function by direct calculation Witt t=1, Re=1000

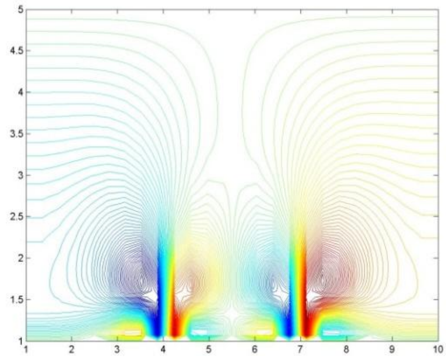

Fig 6a: ISO - values of the of the velocity by direct calculation With t=1, Re=100

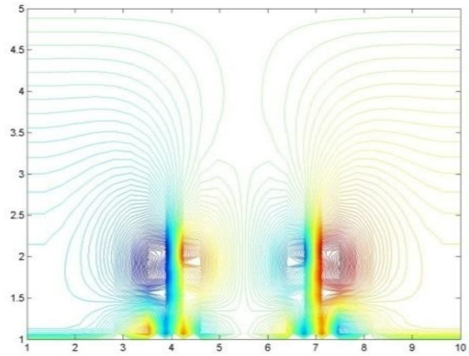

Fig 6b: ISO - values of the velocity by direct calculation With t=1,Re=1000

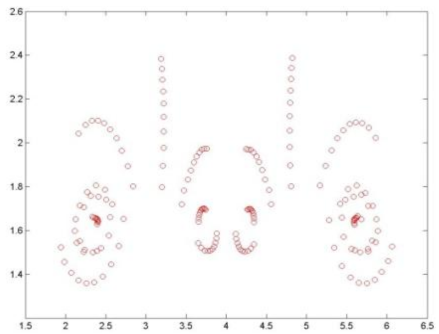

Fig 7a: I Movement of fluid particles in the subdomain Ω2 , Re=100

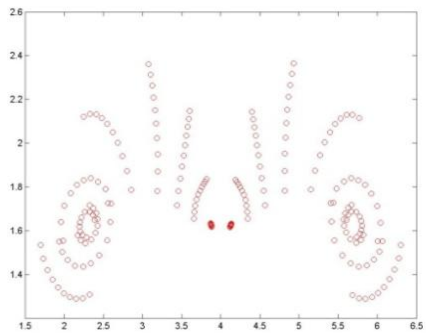

Fig 7b: Movement of fluid particles in the subdomain Ω2 , Re=1000

The fields of the current function and *concentration* and *the vortex function*, within the flow, are simultaneously represented in the figure (3-a, 3-b, 4-a, 4-b, 5-a, 5-b, 6-a, 6-b we presented the results of direct calculation). The maximum speed corresponds to a high vortex intensity. The flow widens progressively as the distance of the ejection section. The flow is characterized by the development of the boundary layer on the wall. The velocity field reflects the influence of the boundary layer and the lower limits of the conditions of flow. A recirculation zone appears in the shear layer. Induces the sliding speed at first the generation of a number of vortices which is associated a circulation amount. These vortices are then disseminated but they are not convected because the velocity is zero at the wall. Dice the next step, these vortices are both disseminated and convected. In the figure (7-a, 7-b) we presente the results of (the particle method in the outdoor area) .the record is by conditions of trasmession using a model for creating particle-fluid redistribution ,taking information on the interface between the two vertuel subdomain. this method it's more useful for the high Reynolds number ,can reproduces the jet flow, with fewer difficulties than those encountered with the direct method

IV.1. Behaviour of the error over time

In this section we are interested in the behavior of the error made on the swirl and the current and the concentration depending on time; these errors are given by the following formulas:

$$\varepsilon_{\psi} = max_{i,j} \left|\psi_{ij}^{n+1} \_ \psi_{ij}^{n}\right|$$
$$\varepsilon_{\omega} = max_{i,j} \left|\omega_{ij}^{n+1} \_ \omega_{ij}^{n}\right|$$
$$\varepsilon_{C} = max_{i,j} \left|C_{ij}^{n+1} \_ C_{ij}^{n}\right|$$

The figures below give the curves of the respective errors:

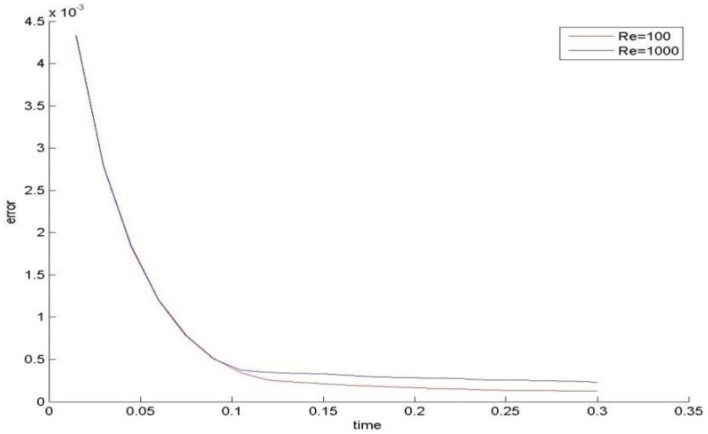

Fig 8: The error of the current function

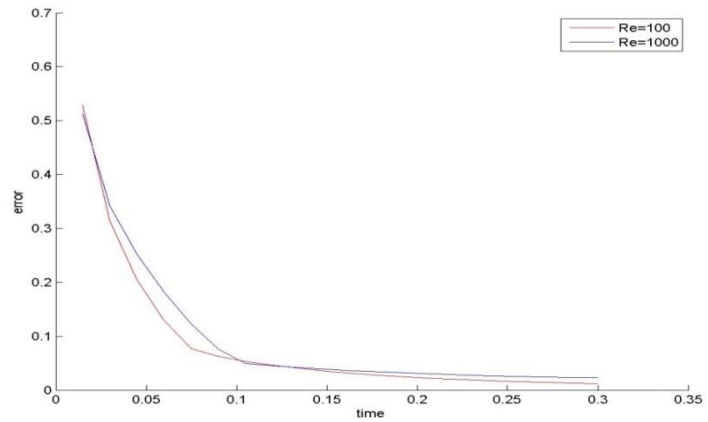

Fig9: The error of the vorticity function

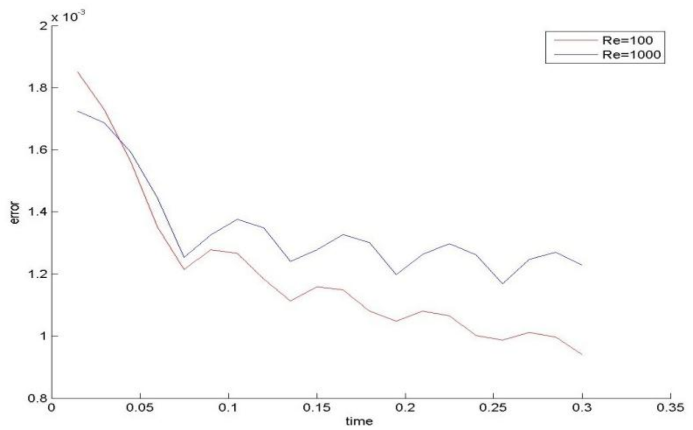

Fig10: the error of the concentration function C

The curves obtained as a function of time for Re = 1000 show a numerical instability due to poor approximation derived by finite differences and ill-conditioned algebraic systems,

The results show that the finite difference methods for resolution the system (1 and 2) is inadequate for the number of Reynolds high, to have digital reliable results the method of decomposition is at stake

VI. CONCLUSIONS

The phenomena studied locally in terms of fields of speeds and *current* lines and concentration, allowed us to realize the importance of interaction and evolution of flow depending on the Reynolds number, and mechanisms of instabilities flow were detected by numerical simulation. The influence of Reynolds number needs further study. It should further model the interactions of Reynolds number even higher in order to see if other structures may be created in the shear zone of the flow. Moreover the domain decomposition method we proposed can analyze and simulate problems for large Reynolds number. It is well suited to the problems of confined flow and external flows in the presence of the walls with a high recirculation speed fields.

REFERENCES

[1] A. Jolles, Résolution des équations de Navier Stokes par une méthode particules-maillage, Thèse de Doctorat Université Paris-VI, Décembre 1989.

[2] G. Mangoub, Etude numérique de l'interaction tourbillon paroi par une méthode particules-maillage, Thèse de Doctorat Université Paris-VI, Décembre 1992.

[3] O. Daube – T. P. Loc, Méthodes hermitiennes compactes d'ordre $H^2 – H^4$, Note et documents LIMSI.

[4] D. W. Peacemen et H. H. Rachford, The numerical solution of parabolic and elliptic differential equation, J. Scoc. Ind. Appl. Math., Vol 3, p 28-41, 1955.

[5] M. Chagdali, contribution numérique à l'étude de l'écoulement turbulent externe.

