International Journal of Innovation and Scientific Research
ISSN 2351-8014 Vol. 17 No. 1 Aug. 2015, pp. 114-119

http://www.ijisr.issr-journals.org/

The Method of decomposition domain for the numerical modeling of a jet by the particle-mesh method

Khalid Adnaoui, Noureddine Tounsi, Mohamed Chagdali, and Soumia Mordane

Faculté des Sciences Ben M'Sik,
Casablanca, Morocco

ABSTRACT: This work concerns to the digital treatment of the problems with strong not linearities during the resolution of the equations of Navier-Stokes in particular those due to the recirculation strong in turbulent regime. The idea developed is to use the method of subdomains: The domain in which took place the flow is decomposed several subdomains separated by imaginary boundary. In each of these subdomains, we use the best adapted digital method. The passage in all the domain is made by digital connecting. This connecting is made by covering of domain. The results are presented in the case of a jet of rejection emitted by the bottom in a rectangular canal. In this application, we divided the domain of study into two parts: Near the boundary layer, we use the finished difference method and in the outside zone the resolution is made by the method Particular. The fictitious interface between these two subdomains is processed by the method particles - meshing. A validation of this approach is made by a comparison with a direct calculation in all the domain.

KEYWORDS: Equations of Navier-Stokes, numerical methods, particulars Methods, Method of finished differences, Decomposition of the domain, Scheme T.S.C of projection interpolation.

1 INTRODUCTION

In this work we present a complete numerical solution of flow plan generated by a jet emitted from the bottom in an open canal. The fluid is considered incompressible viscous. The direct resolution of such a problem by the methods of differences finished puts number difficulties for the large numbers of Reynolds. Indeed, numerical instabilities due to the treatment of the not linear terms in zones with strong variations of speed appear what require refining the meshing. But this type of methods used to carefully consider the boundary conditions on solid walls. For high Reynolds numbers, other methods are better suited to handle viscous fluid flows in the case of strong recirculation of velocity vectors, mention particle methods. But they have other types of drawbacks on treatment of the boundary conditions at the wall. We present a numerical approach which combines the advantages offered by each of both methods quoted previously. For that purpose, the domain of the flow is divided into two subdomains in each of which is used the most appropriate method: in the neighborhood of walls, we use a method of the finished differences and somewhere else the method particular. But there raises the problem of connecting both subdomains, that is of the transmission of the information enter a method which uses a meshing (the method of the finished differences) and another one without meshing (the method particular). To solve this problem, we used the following technique: It makes the emission of particles after a test on the sign of the normal speed at the interface between the two subdomains. The particles which by effect of recirculation enter the first one subdomain close to the solid wall are distributed on the meshing of differences finished by a technique particles-meshings and new particles are created in exchange [1][2].

2 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:

STEP 1: GEOMETRIC DECOMPOSITION OF THE DOMAIN:

The domain resolution is decomposed into several subdomains. This decomposition is based on physical and numerical considerations: Nature of the flow regime, ease of writing the conditions in the limits and treatment of nonlinearities in the equations of movement.

For connected domain, the decomposition is based on the partitioning of the domain Ω into n (n ≥ 2) subdomain Ωi :

$$\Omega = \bigcup_{1 \le i \le n} \Omega_i$$

The intersection between the subdomain is limited to the interfaces (Figure 1):

$$1 \le i < j \le n, \Omega_i \cap \Omega_j = \partial\Omega_i \cap \partial\Omega_j$$

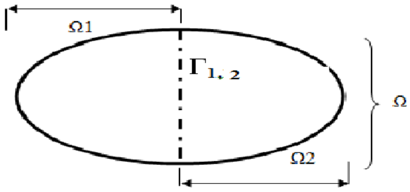

Figure 1: Partition into two subdomains Ω1 et Ω2

STEP 2: CHOICE OF METHOD FOR SOLVING IN THE SUBDOMAIN:

The choice of the method of resolution in every subdomain is made on the basis of the capacity of the digital method in treated the equations of the movement. In particular, we consider the calculation time, the adaptation to the treatments of non-linearity and to the geometrical configuration.

STEP 3: WRITING CONDITIONS INTERFACES:

It is a question of writing conditions of continuity of variables between both under geometrical domains. In particular it is necessary to assure the implicit character of the resolution of the equations of movements.

3 APPLICATION

We tried to solve the equations of Navier-Stokes and the equation of transport of mass of a jet at the bottom of a canal. In terms of vortex function ⍰ , function of current ψ , and concentration C, these equations are [3]:

$$\frac{\partial \omega}{\partial t} + (\vec{U}.\vec{\nabla})\omega = \frac{1}{\mathcal{R}e}\Delta\omega + \frac{1}{\mathcal{F}r^2}\left[\vec{\nabla}C \wedge \frac{\vec{g}}{\|\vec{g}\|}\right].\vec{k} \quad (1)$$

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

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

$$\frac{\partial C}{\partial t} + (\vec{U}.\vec{\nabla})C = \frac{1}{\mathcal{R}eSc}\Delta C \quad (4)$$

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 flow.
- $\vec{U}$ is the velocity Victor.

$\Omega = \bigcup \Omega_i.$

- ψ is the function of current
- ω is the vortex function , such a $\vec{\omega} = \omega\vec{k} = \vec{\nabla} \wedge \vec{U}$
- C is the concentration of the pollutant.
- $\vec{g}$ acceleration of gravity.
- $\mathcal{R}e$, Sc, Fr Represent respectively the numbers of: Reynolds, Schmidt and Froude.

The connected rectangular domain Ω is decomposed into two subdomains Ω1 and Ω2 separated by a border Γ (figure 2):

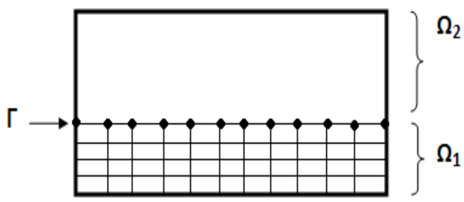

Figure 2: Subdivision of the domain Ω.

The jet is the bottom (Domain Ω1) and the flow is dominated by the effect of the walls. The resolution method selected in this subdomain is the method of finished differences [2]. 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), the flow is external in large number of Reynolds and in big recirculation of speed. In this domain, we use the particular method [5], 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 subdomains 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. If some particles of Ω2 come in Ω1, their intensities (vorticity and concentration) are distributed on the nodes of the mesh used for the method of finished differences, one using the particle mesh method.

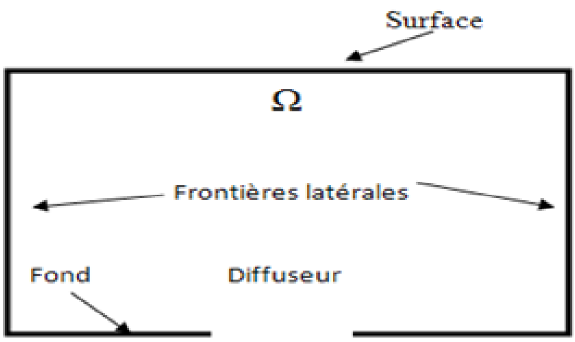

Figure 3: Domain of study Ω.

4 RESULTS

Geometric Data: Length 6 m and 4 m high. The fictitious interface between the two domains Ω1 and Ω2 under fixed at

y = 1.6.

The results will be presented in terms of the iso-values tourbillon w, function, the concentration C, the lines of current, in all the domain of study Ω.

Given physical: Fr = 18, Sc = 1.5.

A direct calculation was made in all the domain of study by the finished difference method. Figures 4a, 4b, 4c and the results are presented for a Reynolds number Re = 100.

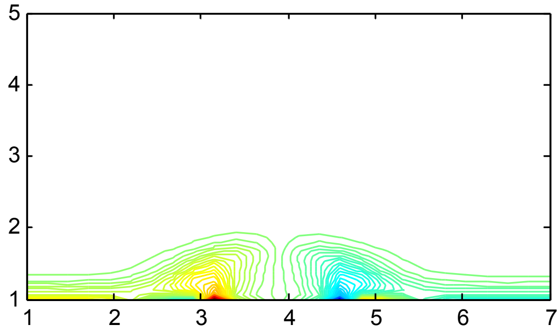

Figure 4a : Iso-vortices by direct calculation at t = 1.

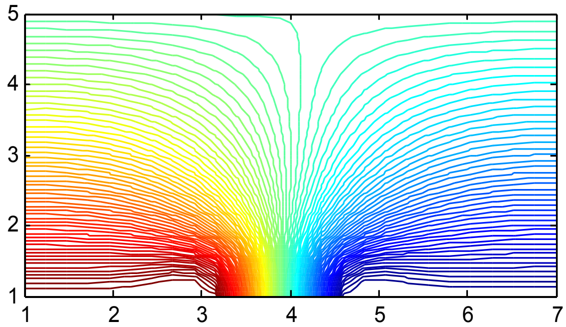

Figure 4b : The current lines by direct calculation at t = 1.

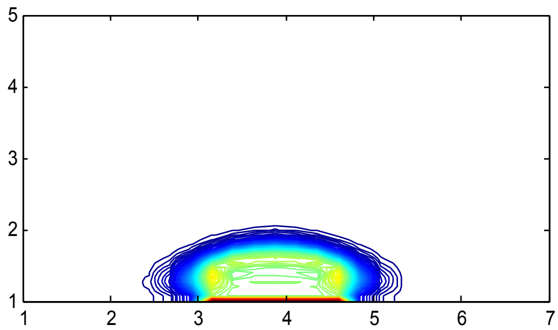

Figure 4c : Iso-concentrations by direct calculation at t = 1.

Direct calculation becomes unstable from Re = 1000. In the same geometrical configuration, it solved the complete Navier-Stokes equations by the method of the subdomains.

The fictitious interface separating the two subdomain Ω1 and Ω2 is set at y = 1.6. It is from this interface that are emitted particles representing the flow of the jet in the subdomain Ω2 between y = 1.6 and y = 5. We present on the figure 5a calculation for the number of Reynolds Re=100, and on the figure 5b a calculation for the number of Reynolds Re=1000.

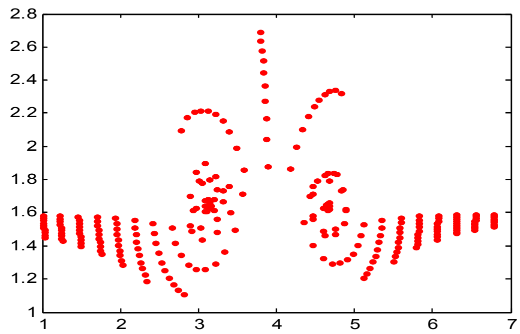

Figure 5a : Movement of fluid particles in the subdomain Ω2 at t = 1.

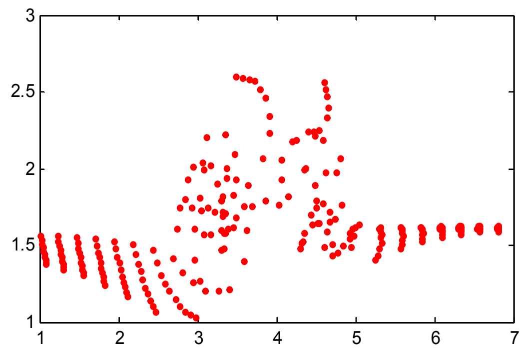

Figure 5b : Movement of fluid particles in the subdomain Ω2 at t = 1.

The particles that pass from the subdomain Ω2 to Ω1 will project their information carried (vortex and concentration) on the nodes of the closest mesh with the projection Scheme TSC.

On the figure 6a and 6b, we present the results for a number of Reynolds Re 100.

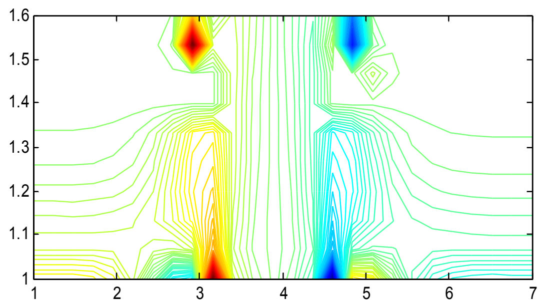

Figure 6a : Iso-vortices in the sub domain Ω1 at t = 1.

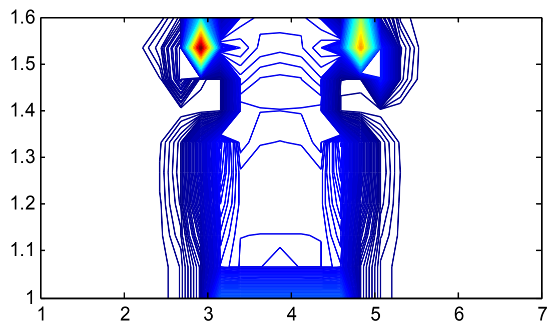

Figure 6b : Iso-concentration in the subdomain Ω1 at t = 1.

5 CONCLUSION

From the results presented, we can conclude that the method of multi-domain has removed the difficulties posed by the finite difference method and are related to numerical instabilities associated with high Reynolds numbers. Based on the combination of several methods.

Multi domain method allows the advantages of a numerical method given in the region where it is most effective.

REFERENCES

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

[2] O. DAUBE – T. P. LOC," Etude numérique d'écoulements instationnaires de fluides visqueux incompressibles autour de corps profilés, par une méthode combinée d'ordre O(h2) et O(h4) Journal de mécanique Vol 17 n°6 1978.

[3] G. Mangoub, Méthode de sous domaine et de décomposition de champs de vitesse : Application aux écoulements à surface libre, Thèse de Doctorat D'état à l'Université Hassan II Mohammedia, Faculté Ben M'Sik, Juillet 2007.

[4] K. Adnaoui, N. Tounsi, M. Chagdali, S. Mordane, The method of decomposition domain for the digital modelling of a jet, American Journal of Applied Mathematics 2014; 2(2): 64-68.

[5] HUBERSON S. (1986).Modélisation asymptotique et simulation numérique d'écoulements tourbillonnaires. Thèse de doctorat d'état, Université Paris 6.