UNIVERSITA' DEGLI STUDI DI ROMA TorVergata

Facoltà di Ingegneria
Dipartimento di Ingegneria Meccanica

Laurea Triennale in Ingegneria Meccanica

ESTIMATION OF THE IMMERSED BOUNDARY TECHNIQUE FOR INDUSTRIAL FLUID DYNAMIC SIMULATIONS

Relatore:
Prof. Ing. Roberto Verzicco

Candidato:
Paolo Marafini

Triennio accademico 2009-2012

Abstract

The purpose of this work is to corroborate a new computational fluid dynamics software produced by Karalit CFD.
This program aims to provide reliable results for industrial applications with CFD, overcoming limitations that other codes have in front of complex geometries.
The Immersed Boundary (IB) technique can satisfy the previous requests; this method adds to the Navier-Stokes equations a body-force f and maintains a Cartesian computational domain.
Two and three dimensional cases were analyzed with this method, the flow conditions were: steady flow or buoyancy-induced flow.
Several three-dimensional examples of flow over a sphere for some Reynolds numbers were simulated and then compared with experimental results.
In order to show the potential of Karalit CFD, in this paper, there are applications that present significant difficulties for the traditional methods: so, the software has been used with the same ease and satisfaction for objects with very complex geometry.

Abstract

Il presente lavoro è nato con l'intento di validare un nuovo software di fluidodinamica computazionale prodotto dall'azienda Karalit.
Il programma in questione mira a fornire alle industrie risultati attendibili nella CFD, superando i limiti che i programmi precedentemente impiegati riscontravano di fronte a oggetti con geometrie complesse.
Il metodo usato per fare ciò è il metodo dei contorni immersi, il quale interviene nelle equazioni di Navier-Stokes aggiungendo un termine di forza di volume f mantenendo però un dominio di calcolo basato su griglie cartesiane.
Sono stati analizzati con questo metodo casi in due e tre dimensioni, sia in condizioni di moto del fluido stazionario sia con moto dovuto a gradienti di temperatura.
In particolare, per il caso 3D, si sono svolte più simulazioni, variando solo il numero di Reynolds, riguardanti il flusso intorno ad una sfera, e si sono poi confrontati i risultati con dati di letteratura.
Per mostrare ulteriormente le potenzialità di Karalit CFD, si è pensato di studiare anche casi che, contrariamente alla sfera immersa in un fluido, presentano difficoltà ingenti di analisi con i metodi tradizionali: ebbene, la stessa facilità di applicazione del programma si è potuta riscontrare con soddisfazione anche per oggetti a geometria molto complessa.

1 Karalit CFD

In the past the Immersed Boundary method was considered a mere research tool, but now thanks to its *"simplicity without compromise"* is also used for industrial applications.
In this context the Karalit's software has been created and implemented, thanks to cooperation between the Karalit CFD and the University of Rome "Tor Vergata" it has been possible to perform independent testing validation.
This thesis was developed in three phases:

1. The simulation of several two-dimensional cases with the integration of equations on structured grids, since the code used only this type of grids in the early stages.

2. The use of the software to reproduce laboratory experiments following the possibility of using unstructured grids, and compare the simulation results with the data available in the literature; in this way it was possible to validate the data obtained by computational fluid dynamics.

3. The use of Karalit to simulate applications that present significant difficulties for the traditional methods, to emphasize Karalit's versatility and speed of use.

1.1 Karalit CFD Introduction

Before presenting the tests performed, it is useful to understand how to create a simulation and how to manage this simulation with the GUI.
You can see in the foreground of the Fig.(1) the window that will be used to create the case called `Sfera_Re_150`. Against the background of Fig.(1) there is the main window of the program, where you can see the contents of our "working directory" and a series of commands to manage them.
As you can see, the interface is divided into two main areas: the first, on the left, is the descriptive summary of our test, the second contains four tabs through which you can choose and define all the parameters of interest.
Regarding the GUI, we need to look at this in detail.The first part allows us to enter a brief description of the case and immediately below we can choose the type of application (flow outside/inside, wind tunnel etc.) next, we can specify: the number of bodies immersed, the fluid and possible meshes.
You can choose the boundary conditions on the surface of the body in the first tab of the second zone, as you can see in Fig.(1), it is possible to manipulate the body through a set of editing tools (*Edit, Divide e Surface selection*); the size of the object and its position relative to Cartesian reference system are given in the section titled "Bounding box".

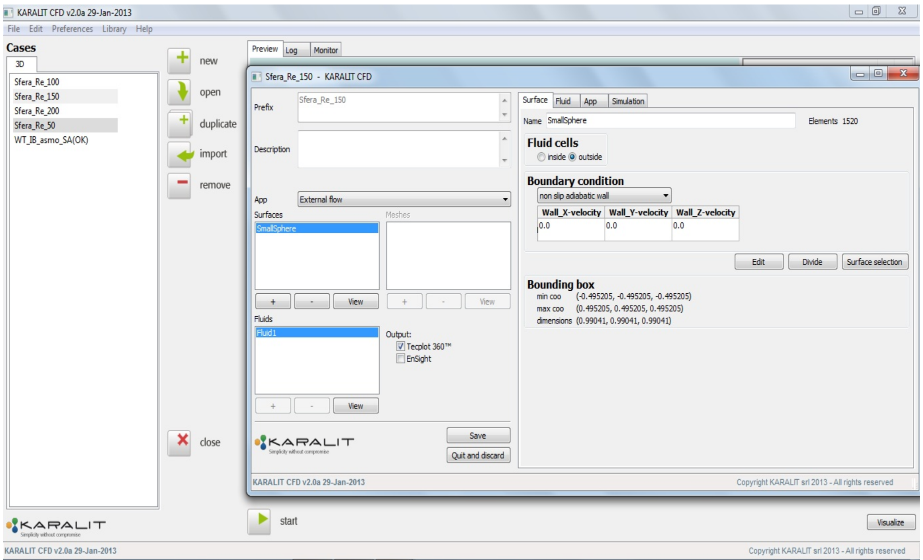

Figure 1: User interface.

2 The flow past a sphere at varying Reynolds numbers

For the validation of Karalit CFD has been relied on this kind of simulations since for this problem have been made many experiments and therefore there are many laboratory data. All the simulations were made with the following assumptions:

1. Steady axisymmetric flow $5 < Re < 200$
2. Incompressible fluid
3. Characteristics of the simulations
   - Density $\rho = 1000\ \frac{Kg}{m^3}$
   - Components of the velocity vector $V_x = 2,225\,10^{-4}\ \frac{m}{s}$, $V_y = V_z = 0$
   - Diameter of the sphere $D = 1\,m$
   - Pressure $p = 101325\ Pa$
   - Temperature $T = 228.15\ K$
   - $CFL = 100$
   - Maximum number of iterations equal to 500
   - *Monitoring x-momentum residual, stop when convergence reaches* 10^{-4}

The viscosity of the fluid μ has been changed in order to obtain the variation of the Reynolds number. The sphere was positioned in the center of a computational domain, which is of cubic shape and twenty times larger Fig.(2).

The characteristics of the grid are:

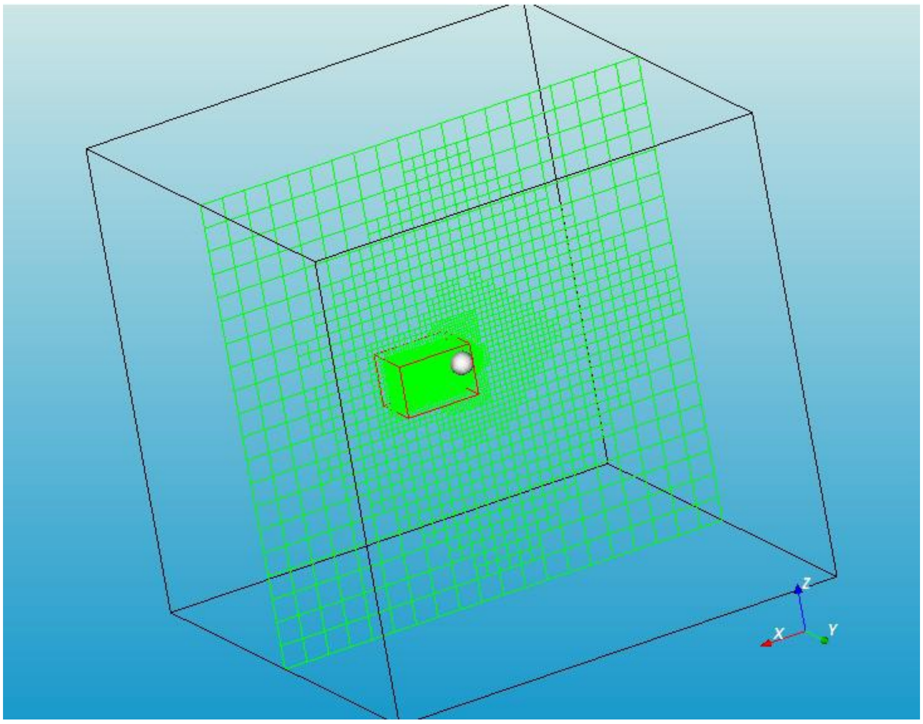

Figure 2: Computational domain.

- Initial size of the grid to the extreme of the domain equal to the characteristic size of the sphere D;
- number of layers equal to 10;
- number of refinement equal to 5, so the minimum size of the cell is $d = 0.03125$

The integration was performed according to the method of Gauss-Seidel and using the scheme *2nd order symmetric TVD*.

2.1 Simulations

In this work, four simulations are shown and the Reynolds numbers selected are: $Re = 50$, $Re = 100$, $Re = 150$, $Re = 200$. The software has been validated by the results obtained from these cases. The geometric characteristics considered are shown in Fig.(3)

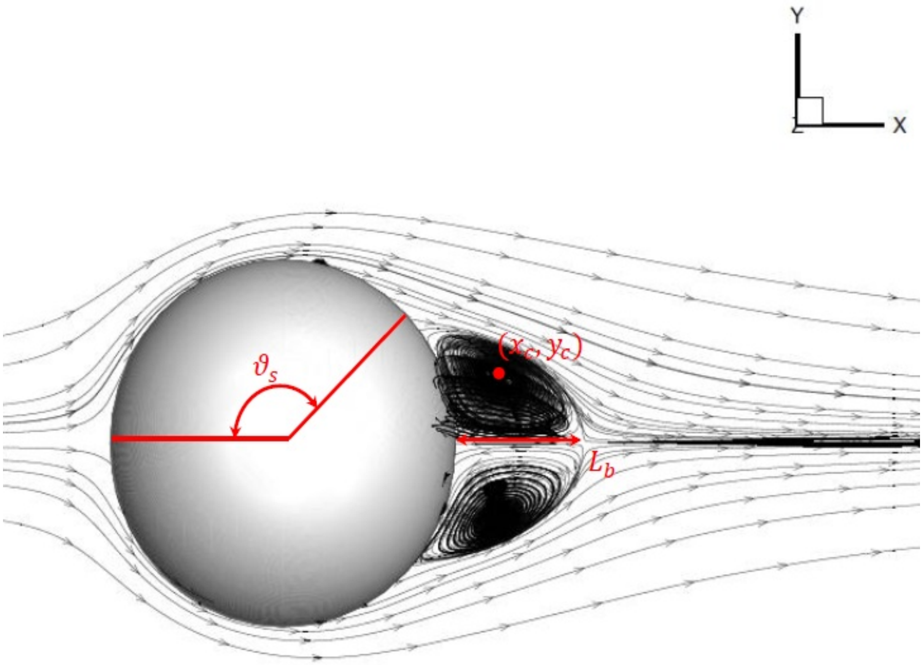

Figure 3: Flow at $Re = 50$.

As illustrated in Fig.(3), the flow is seen to separate from the surface of the sphere at an angle ϑ_s from the front stagnation point, it forms a closed separation bubble (length L_s) and a toroidal vortex centred at (x_c, y_c). The comparison between the literature data and the simulation data has been done on these parameters. In the Fig.s(3-7) are shown the results of simulation, while in Tab.(1) are reported the number of total cells for each simulation.

Re	number of cells
50	194216
100	265420
150	614552
200	614552

Table 1: Re and number of cells

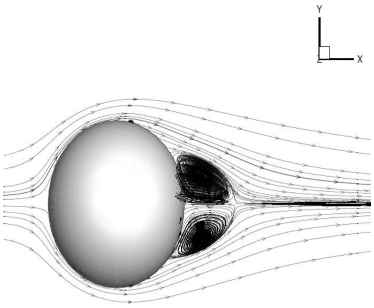

Figure 4: Flow at $Re = 50$.

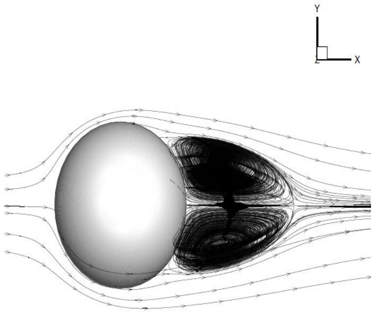

Figure 5: Flow at $Re = 100$.

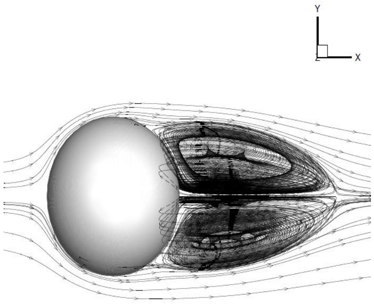

Figure 6: Flow at $Re = 150$.

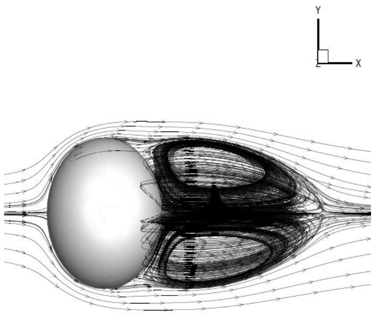

Figure 7: Flow at $Re = 200$.

2.2 Comparison between simulations and data from the literature

In addition to the geometric characteristics in Sec.(2.1) was also considered the drag coefficient C_D.

$$C_D = \frac{F_x}{\frac{1}{2}\rho {V_x}^2 S}$$

S is defined as $S = \pi\frac{D^2}{4}$
The comparison result is shown in Fig.s(8-11).

The Fig.(9) was considered significant because the data reported by Johnson and Patel and the data obtained from Magnaudet differ between them of 11.5%, exceeded the threshold of $Re = 150$, while those obtained from Karalit CFD are placed approximately in the middle between the two: in fact these results deviate by 5% from the results of Johnson and Patel and 6.9% from the results of Magnaudet.

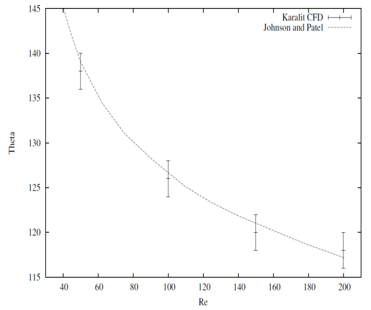

Figure 8: ϑ_s.

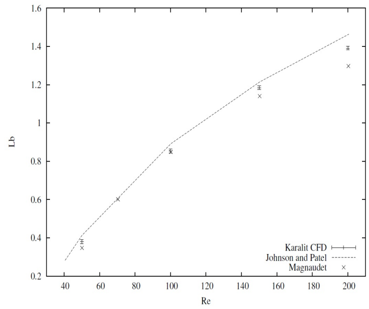

Figure 9: L_b.

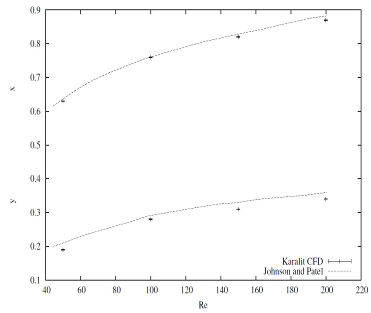

Figure 10: (x_c, y_c).

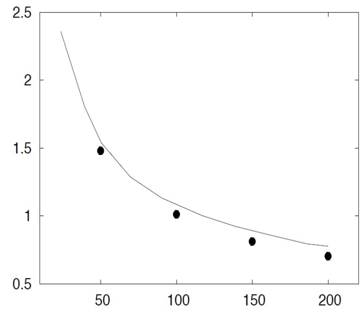

Figure 11: C_D.

Conclusions

After analyzing the results of simulations made with Karalit, a selection of which is shown in this paper, the software can be considered validated. In fact, the comparison between the data obtained from the simulator and those found in the literature have always shown a substantial agreement.
The immersed boundary method proved to be excellent also in simulations that were performed on objects of very complex form, which would cause considerable difficulties when faced following the traditional methods.
Another important factor is the ease with which Karalit CFD can be used through its interface, so the user should not program, in fact, the user will only have to enter the parameters simulation will be based on and choose the setting provided by the software which he considers best for his test.
The immersed boundary method, in combination with KARALIT, can break down considerably the setup time, in fact the methods normally used would have required also a month of preparation, instead with this software only takes a few seconds to switch to simulation.
The conclusion that can be drawn is that the usability of the immersed boundary method through a GUI turned out to be an excellent alternative to methods that are normally used in the simulations of industrial applications.

References

[1] Vieceli JA (1969), *A method for including arbitrary external boundaries in the MAC incompressible fluid computing technique*, J.Comput. Phys.

[2] Vieceli JA (1971), *A computing method for incompressible flows bounded by moving walls*, J. Comput. Phys.

[3] Peskin CS (1972), *Flow patterns around heart valves: A digital computer method for solving the equations of motion*, PhD thesis, Physiology, Albert Einstein College of Medicine.

[4] Peskin CS and McQueen DM (1989), *A three-dimensional computational method for blood flow in the heart: (I) immersed elastic fibers in a viscous incompressible fluid*, J. Comput. Phys.

[5] McQueen DM and Peskin CS (1989), *A three-dimensional computational method for blood flow in the heart: (II) contractile fibers*, J. Comput. Phys.

[6] Briscolini M and Santangelo P (1989), *Development of the mask method for incompressible unsteady flows*, J. Comput. Phys.

[7] Saiki EM and Biringen S (1996), *Numerical simulation of a cylinder in uniform flow: Application of a virtual boundary method*, J. Comput. Phys.

[8] L. J. Fauci and A. L. Fogelson (1993), *Truncated newton method and modeling of complex immersed elastic structures*, Comm. Pure Appld. Math.

[9] L. J. Fauci and C. S. Peskin (1988), *A computational model of acquatic animal locomotion*, J. Comp. Phys.

[10] R. Dillon, L. J. Fauci, and D. Gaver (1995), *A microscale model of bacterial swimming, chemotaxis and substrate support*, J. Theor. Biol.

[11] Goldstein D, Handler R, and Sirovich L (1993), *Modeling no-slip flow boundary with an external force field*, J. Comput. Phys.

[12] J. Mohd-Yusof (1997), *Combined Immersed Boundaries/B-Splines Methods for Simulations of Flows in Complex Geometries*, CTR Annual Research Briefs, NASA Ames/Stanford University.

[13] E. A. Fadlun, R. Verzicco, P. Orlandi, and J. Mohd-Yusof (2000), *Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations*, J. Comput. Phys.

[14] De Zeeuw D and Powell KG (1993), *An adaptive Cartesian mesh method for the Euler equations*, J. Comput. Phys.

[15] Pember RJ, Bell BJ, Colella P, Crutchfield WJ, and Welcome ML (1995), *An adaptive Cartesian grid method for unsteady compressible flow in irregular regions*, J. Comput. Phys.

[16] D. Angeli, P. Levoni, G. S. Barozzi (2008), *Numerical predictions for stable buoyant regimes within a square cavity a heated horizontal cylinder*, International Journal of Heat and Mass Transfer.

[17] T. A. Johnson, V. C. Patel (1999), *Flow past a sphere up to a Reynolds number of 300*, Iowa Institute of Hydraulic Research and Department of Mechanical Engineering.

[18] R. Verzicco (2006), *Appunti di Turbolenza*, Università degli studi di Roma "Tor Vergata" Dipartimento di Ingegneria Meccanica.