Application of the *Ghost Cell Method* to motoring flows

Tommaso Stilo

Introduction

The *Ghost Cell Method* belongs to the cathegory of the *Immersed Boundary Methods*. It can handle the typical difficulties of the study of flows in complex geometries - that is, the generation of a non-structured grid - retaining the accuracy of the simulations performed on regular grids.

The purpose of this work will be the validation of this method by comparing the results of simulations carried out using this technique with results obtained with standard methods usually adopted in industry. The Ghost Cell Method will be enforced through KARALIT CFD software.

1. Setting the case

This work addresses the study of the flow within the airbox of Formula SAE 2013 single-seaters. The airbox is an intake manifold capable of guaranteeing the best aspiration condition for the engine cylinders. The manifold is characterized by a *dual plenum* geometry, which allows the decoupling of the two couples of cylinders (1-4 and 2-3 in Fig. 1), providing them with two separate air masses.

The geometry has two fluidodynamical criticalities: a restrictor, with the aim of reaching supersonic speeds in the duct, and a butterfly valve, used to regulate the flow.

Fig. 1 Formula SAE 2013 single-seater intake manifold.

The description of the geometry used by the software is a STL file composed of 20,006 elements.

The first step in the setting is the generation of a grid. KARALIT can produce a locally refined regular grid; the grid, in this geometry, is composed of 1,508,328 cells. The initial grid spacing is 0.01m; in the proximity of the surface, the tangential dimension of the cell is 0.01m e and the normal dimension is 0.0008 (the grid is anisotropic).

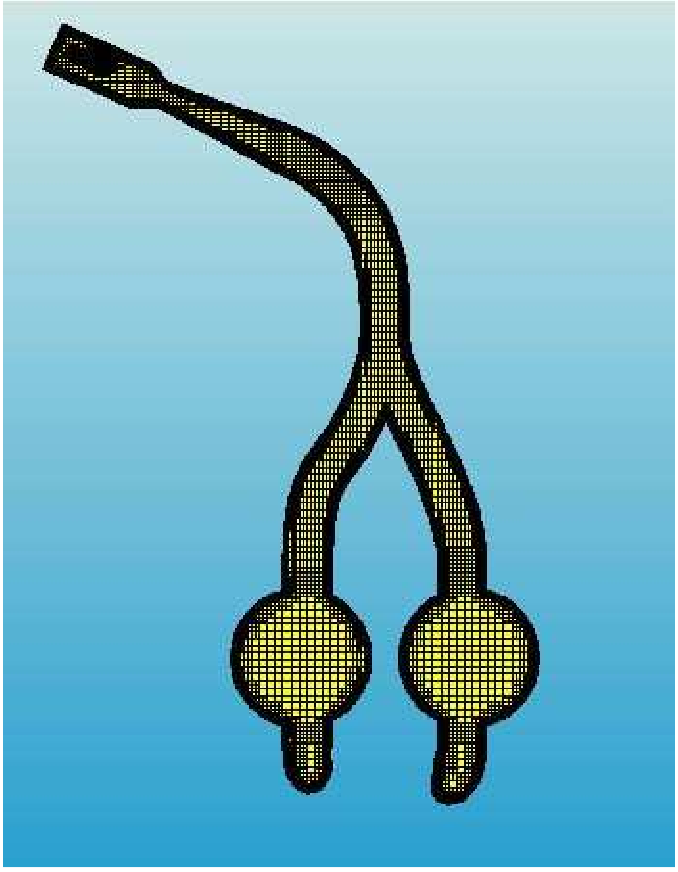

Fig. 2 The grid on the plane of symmetry.

Concerning the boundary conditions, the imposed differential pressure between the inlet and the outlet is $\Delta p = 20{,}000\ Pa$.

The most important parameters of the simulation are the CFL condition and the Under Relaxation Factor. They affect the computational step of the simulation and must be assessed in such a way that the stability of the simulation is maintained. The chosen values are 100 and 0.5, respectively.

In the next section, the results will be discussed and compared with data obtained using Fluent software.

2. Results

It took the solution about 5,000 iterations to reach convergence. Post-processing of the data was executed with KARALIT internal visualization system. Given the geometry of the manifold, the calculation was carried out only for the outlets 1 and 2, expecting symmetrical results for the outlets 4 and 3, respectively.

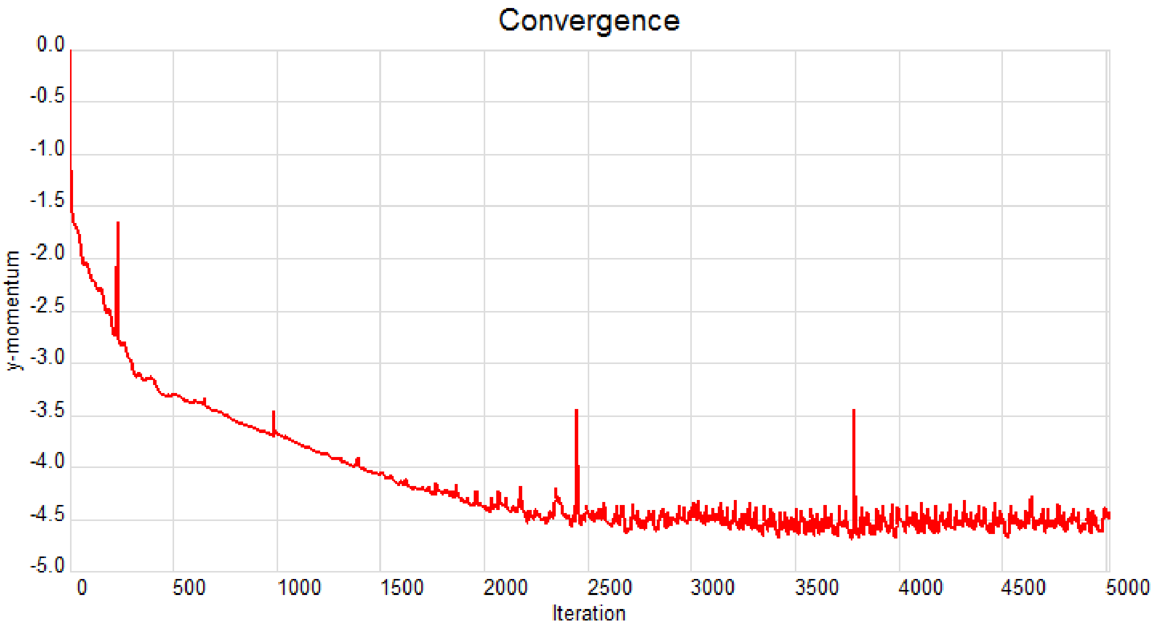

Fig. 3 Residual trend in the simulation.

The computed mass flow rate differs from the data obtained using Fluent by 9.81%. The results are shown in the following table.

	KARALIT	**Fluent**
Mass flow rate outlet 1 [kg/s]	0,0640	0,0709
Mass flow rate outlet 2 [kg/s]	0,0638	0,0708

Mass flow rate: KARALIT Vs Fluent results.

The second result concerns the velocity field. Both KARALIT and Fluent obviously show the maximum at the restrictor, where $Ma > 1$. Notice that the wake generated by the valve closes before entering the restrictor.

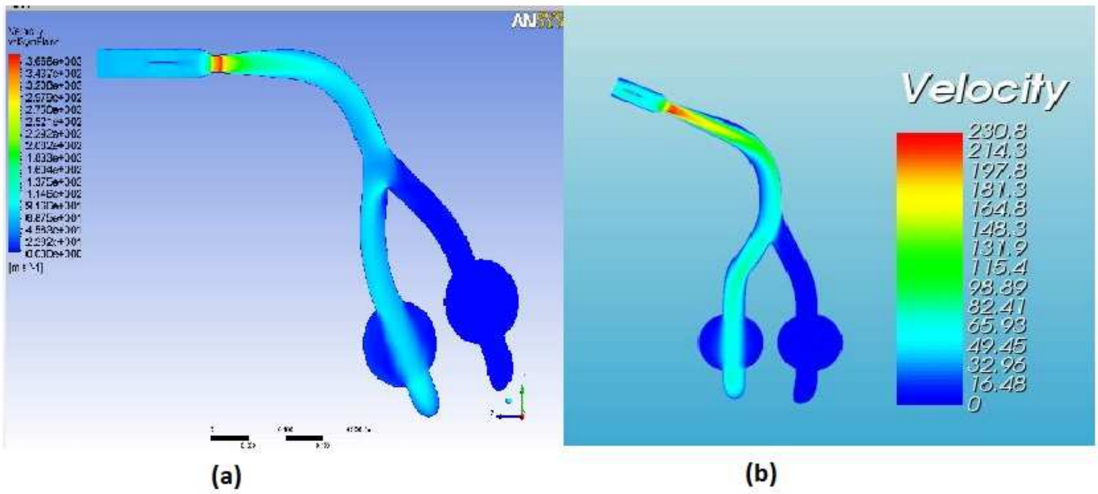

Fig. 4 Velocity field: Fluent (a) Vs KARALIT (b) results.

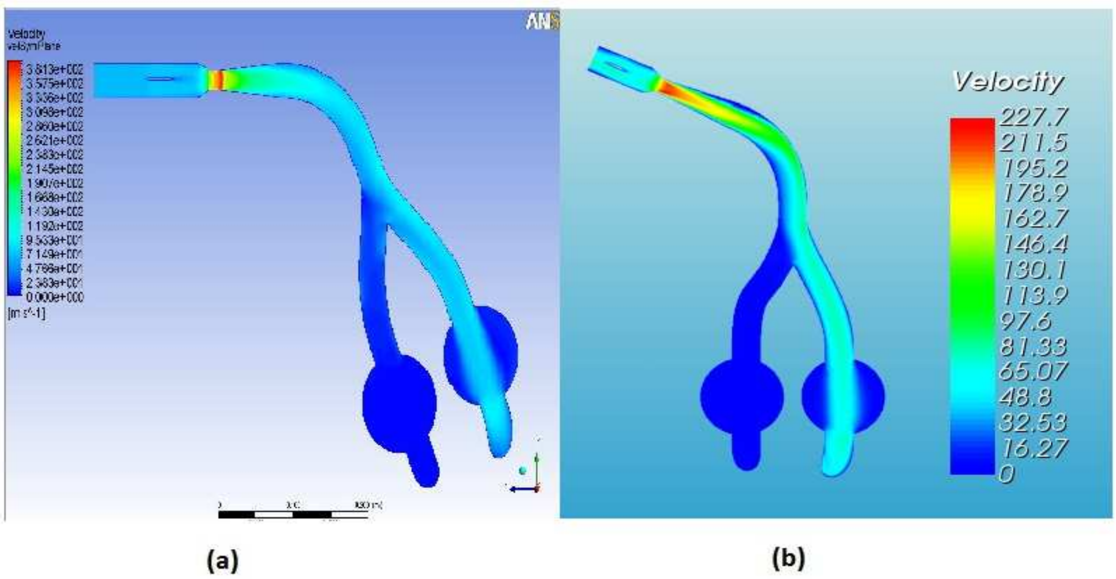

Fig. 5 Velocity field: Fluent (a) Vs KARALIT (b) results.

Conclusions

In the last section, the reported data show a nearly perfect correspondence between the results obtained using KARALIT and the ones obtained using Fluent, that is a standard software used in industry. Therefore, it's possible to affirm that the Ghost Cell Method is validated.

However, there are a number of points that still need a detailed investigation, such as the implementation of new turbulence models and more realistic boundary conditions (e.g. rough surfaces), but this work encourages the use of the present approach for industrially relevant applications.

References

[1] G. Iaccarino, R. Verzicco. «Immersed boundary technique for turbulent flow simulations» in *Applied Mechanics Reviews, Volume 56 Issue 3* (2003).

[2] Y. H. Tseng, J. H. Ferziger. «A ghost-cell immersed boundary method for flow in complex geometry» in *Journal of Computational Physics 192* (2003).

[3] E. A. Fadlun, R. Verzicco, P. Orlandi e J. Mohd-Yusof. «Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations» in *Journal of Computational Physics 161* (2000).

[4] R. Mittal, G. Iaccarino. «Immersed Boundary Methods» in *Annual Review of Fluid Mechanics* (2005).

[5] André Bakker. «Lecture 10 – Turbulence Models» in *Applied Computational Fluid Dynamics*. 2002. URL: `http://www.bakker.org/dartmouth06/engs150/10-rans.pdf`

[6] KARALIT srl. *Extended User Guide*. 2013.

[7] R. Verzicco. *Appunti di fluidodinamica*. Notes. 2006.

[8] R. Verzicco. *Appunti di turbolenza*. Notes. 2006.

[9] G. Topazio. *Fluidodinamica dei condotti di aspirazione di una vettura Formula SAE. Analisi ed ottimizzazione.* Bachelor of Science thesis. 2012.

[10] KARALIT srl. *KARALIT CFD 2.1 TRAINING*. Webinar by M. Talice. 2014.