A Unified Simulation Framework for Compressible Flows at All Speeds in Industrial Applications

o ChungGang Li, RIKEN, 7-1-26, Minatojima-minami-machi, Chuo-ku, Kobe, chung-gang.li@riken.jp
Makoto Tsubokura, RIKEN, 7-1-26, Minatojima-minami-machi, Chuo-ku, Kobe, mtsubo@riken.jp
Keiji Onishi, RIKEN, 7-1-26, Minatojima-minami-machi, Chuo-ku, Kobe, keiji.onishi@riken.jp
Niclas Jansson, Riken, 7-1-26, Minatojima-minami-machi, Chuo-ku, Kobe, leifniclas.jansson@riken.jp
Rahul Bale, RIKEN, 7-1-26, Minatojima-minami-machi, Chuo-ku, Kobe, rahul.bale@riken.jp

Computational fluid dynamics (CFD) is a major tool of aid in the design and analysis for industrial applications owing to its cost effectiveness. However, this topic accompanies with several critical issues including the variable density at low Mach numbers such as combustion, rapid turnaround time and complex geometry. We use Roe scheme with preconditioning and dual time stepping to tackle slow flows with variable densities. Building Cube Method (BCM) is adopted to make our method suitable for massive parallelization systems to tremendously reduce the calculation and turnaround time. An immersed boundary method (IBM) for compressible flows with a fast, easy to implement and robust interpolation method is developed to handle flows with complex immersed geometries. The results show that the present program is suitable for aid in design and analysis for practical products due to its high performance and wide availability.

1. Introduction

CFD is widely adopted as a major tool of aid in design and analysis such as vehicle shape, combustion and heat sink in LED owing to its cost effectiveness. So, a new CFD program with the capabilities of high performance computing (HPC) for rapid turnaround, dealing with the complex geometries and obtaining accurate results, is highly demanded.

Therefore, the aim of this study is to develop a sophisticated and robust program for heat transfer and fluid flow phenomena on industrial applications. A program which can tremendously reduce the turnaround time is needed. BCM [1] is adopted to make the program suitable for massive parallelization and high performance computing to reduce the turnaround time. Besides, it is obvious that in practical applications, the variable density at low Mach numbers such as combustion, vehicle aeroacoustic or chimney design should be always taken into consideration. So, we use Roe scheme with preconditioning method [3], which allows the program is also applicable for the above situation. Generally speaking, the practical applications always accompany with the issue of the complex geometry. In order to handle this issue, an immersed boundary which is suited to compressible flows and enables to treat the infinitely thin structure such as fins is developed.

2. Governing equation and numerical method

The governing equations are the original Navier–Stokes equations with the source term to calculate the effect of the buoyancy force.

$$\frac{\partial U}{\partial t}+\frac{\partial F_1}{\partial x_1}+\frac{\partial F_2}{\partial x_2}+\frac{\partial F_3}{\partial x_3}=S\ . \qquad (1)$$

The quantities included in U , F_i , and S are

$U=(\rho,\rho u_1,\rho u_2,\rho u_3,\rho e)$,

$$F_i=\begin{pmatrix}\rho u_i\\ \rho u_i u_1+P\delta_{i1}-\mu A_{i1}-g u_1\\ \rho u_i u_2+P\delta_{i2}-\mu A_{i2}\\ \rho u_i u_3+P\delta_{i3}-\mu A_{i3}\\ (\rho e+P)u_i-\mu A_{ij}u_j-k\dfrac{\partial T}{\partial x_i}\end{pmatrix},\quad \forall i=1,2,3$$

and

$$S=\begin{pmatrix}f_1\\ f_2\\ f_3\\ f_4\\ f_5\end{pmatrix}$$, where f_i is a source tem.

At extremely low Mach numbers, the fluid speed is several orders of magnitude lower than the sound speed. The Weiss and Smith preconditioning method [3] is then adopted to resolve the governing equation given by Eq. (1).

In order to have the better load balancing and higher performance computing to save the product turnaround time for massively parallel systems such as ExaFlops computers, BCM is adopted. When the calculation is carried out, the same number of cubes is assigned to each CPU.

For the sake of representing the complex geometry by uniform meshes in BCM, a fast, easy implement and robust immersed boundary method (IBM) for compressible flows is proposed and shown in Figure 1. the Dirichlet condition and Neumann condition for can be represented as Eq. (2) and (3), respectively.

$$\phi_{IC}=\frac{w_1\phi_1+w_2\phi_2+w_3\phi_3+\phi_W}{2-w_{IC}}\text{, Dirichlet condition;} \qquad (2)$$

$$\phi_{IC}=\frac{w_1\phi_1+w_2\phi_2+w_3\phi_3}{1-w_{IC}+\varepsilon}\text{, Neumann condition.} \qquad (3)$$

Where w_1 , w_2 , w_3 , and w_{IC} are the weight coefficients calculated by using the Vandermonde matrix corresponding to the bilinear interpolation scheme [4] and ε is a small value to prevent divergence from the denominator in Eq. (3).

Copyright © 2015 by JSFM

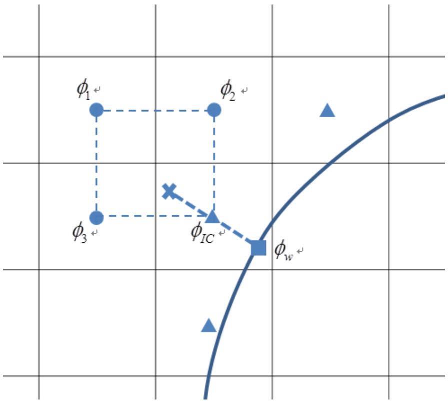


Fig. 1 Compressible IBM configuration

3. Result and discussion

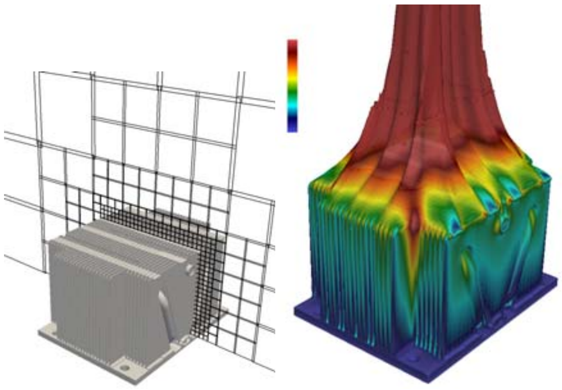
Based on [5], a steady-state natural convection around a heated sphere under the condition of that, the Grashof number (Gr) based on the radius of the sphere is 10^4, is conducted to validate the present program.

Comparisons of the averaged Nusselt number (*Nu*), drag coefficients caused by pressure ($C_{D,p}$) and viscous ($C_{D,\mu}$) with [5] are tabulated in Table 1. The results are in good agreement with [5] and show the accuracy and availability of our program for dealing with the complex geometry and heat transfer problems.

Table 1. Comparisons *Nu*, $C_{D,p}$ and $C_{D,\mu}$

	Nu	$C_{D,p}$	$C_{D,\mu}$
[5]	8.74	0.46	0.62
Present	8.77	0.46	0.59

Besides, our program is also applied to a heat sink with fins and heat pipes to show its availability on practical products shown in Fig. 2(a). The temperature of the heat sink is set to 370K. Fig. 2(b) shows the T = 360K isothermal surface contoured by velocity magnitude. The distribution of the isothermal surface with higher temperature 360K is near the wall, so the shape of the heat sink can be clearly identified. The velocity distribution can be also observed. For example, in the center of the gap between fins, the velocity will be faster, which shows our program has great capability of handling complex geometry even in extremely severe situations.



(a) (b)

Fig. 2 T=360 K isothermal surface contoured by velocity magnitude (m/s).

The parameters and the results of the strong scaling test for 500 iterations in the artificial time are listed in Table 2. The test is run on the Kcomputer, which has massive multi core processors of 2.0 GHz 8-cores SPARC64 VIIIfx developed by Fujitsu. In the case of 48 processors, the load balancing can be almost ignored and our program shows excellent peak performance 29.0 %. Besides, the usage of SIMD also achieves 77.9%. In the case of 384 and 1536 processors, because the situation of the load balancing is more and more server, the lower peak performance can be expected. However, our program still show very good peak performance around 21%. Additionally, the scaling also shows good performance 70%. In an extreme case of 3072 processors, the peak performance and scale performance are still acceptable, which shows our program is also suitable for massively parallel systems such as ExaFlops computers.

Table 2. Strong scaling test

Proc.	N_{cube}	N_{cell}	*T(s)*	*Comm.(s)*
48	473.8	1,940,821	366.0	42.5(11.6%)
384	59.2	242,602	63.2	20.9(32.6%)
1536	14.8	60,650	16.1	3.54(22.0%)
3072	7.4	30,325	11,7	3.58(30.6%)

Proc.	*Peak(%)*	*SIMD(%)*	*Scale*	*Per.(%)*
48	29.0	77.9	1	100
384	20.9	75.4	5.8	72.4
1536	21.0	75.4	22.7	70.9
3072	15.8	68.9	31.3	48.9

4. Conclusion

The computational method and numerical scheme developed in the present study can be widely used for analyzing heat transfer and fluid flow phenomena in industrial applications. Roe scheme with preconditioning method can make the program available in the situation of the variable density at low Mach numbers. BCM is suitable for the massively parallel systems to tremendously reduce the turnaround time. The new interpolation method for IBM can handle the complex geometry. The program is firstly validated by a steady-state natural convection around a heated sphere at $Gr=10^4$. The simulation of a heat sink shows its capability on practical applications. Finally, from the strong scaling test, it can be expected that our program is also suitable for the next generation supercomputer- ExaFlops scale

5. Bibliography

(1) Nakahashi, K. and Kim, L. S., AIAA Paper, (2004) 2004-0423
(2) Gray, D. D. and Giorgini, A., International Journal of Heat and Mass Transfer, 19 (1976), pp. 545–551.
(3) Weiss J. M. and Smith W. A., AIAA Paper (1995), pp. 2050–2056.
(4) Ghias, R. Mittal, R. and Dong, H., Journal of Computational Physics, 225 (2005) pp. 528-553.
(5) Jia, H. and Gogos, G., Int. J. Heat Mass Transfer, 19 (1996) pp. 1603-1615.

Copyright © 2015 by JSFM