Advances in Computer Science Research, volume 50
4th International Conference on Electrical & Electronics Engineering and Computer Science (ICEEECS 2016)

Solver of Multi-GPU Compressible Turbulence Parallel Simulations Used in Aerodynamic Teaching

LUO Kai[a, *], CAO Wenbin [b], LI Song[c] and SONG Limin[d]
Aviation University of Air Force, Changchun 130022, China
[a]1564095655@qq.com, [b]caowenbin08@163.com, [c]Lisong4885@126.com, [d]liminsong_2001@163.com, * corresponding author

Keywords: GPU; computational fluid dynamics; aerodynamic teaching

Abstract. The focus of the current paper is the development of a Solver named multi-GPU compressible turbulence parallel simulations, and its application in aerodynamic teaching. Test of a space shuttle is chosen to demonstrate the accuracy and acceleration performance of the solver. Some teaching examples in aerodynamics which students may have difficulties in understanding are presented to show the advantages of this solver on helping the teachers and students in aerodynamics.

Introduction

Computational fluid dynamics (CFD) is playing a more and more important role in science research and engineering technology [1-2], if we import CFD in classroom teaching, the teaching efficiency must be doubled. We have already established aerodynamic lab which is operating well, but only with low speed wind tunnel, we cannot teach and research efficiently on high speed aerodynamics. Now we have developed a CFD solver to well extend the speed range which can used in sub-sonic, trans-sonic and super-sonic fields, and is considerable accuracy, efficient and low-cost. The solver will strongly promote aerodynamic teaching efficiency, not only in science research.

The Solver

We developed a finite volume computational fluid dynamics solver on the Graphical Processing Unites (GPUs) using CUDA Fortran [3] for compressible turbulence simulations. The solver is implemented with an AUSMPW+ scheme for the spatial discretization, the k-ω SST model for turbulence model, and MPI communication for parallel computing.

We calculate a space shuttle model [4] with Mach number 4.95, AOA-5 degree, and Reynolds number 5.26x107. The length of the space shuttle is 0.29m, and highly accurate experimental results were presented. Considering a steady turbulent state, the wall surface adopted the condition of heat-insulating. The grid size is 0.134 billion with 26 zones, the height of first wall surface grid is $1\text{x}10^{-6}$ m. 4GPUs has been used in the parallel computing, each of which has a 33.50 million grid size load.

Figure 1 shows the pressure distribution along the upper surface of the body symmetric central line, we can see that the computational results revealing the variation of pressure were basically coincide with those got from experiments, which confirm the accuracy and flexibility of the solver.

When implementing parallel computing with 4GPUs, every iteration costs 0.667s, and every single GPU costs 0.611s, the parallel efficiency is 91.6%, so the computational efficiency is considerably high.

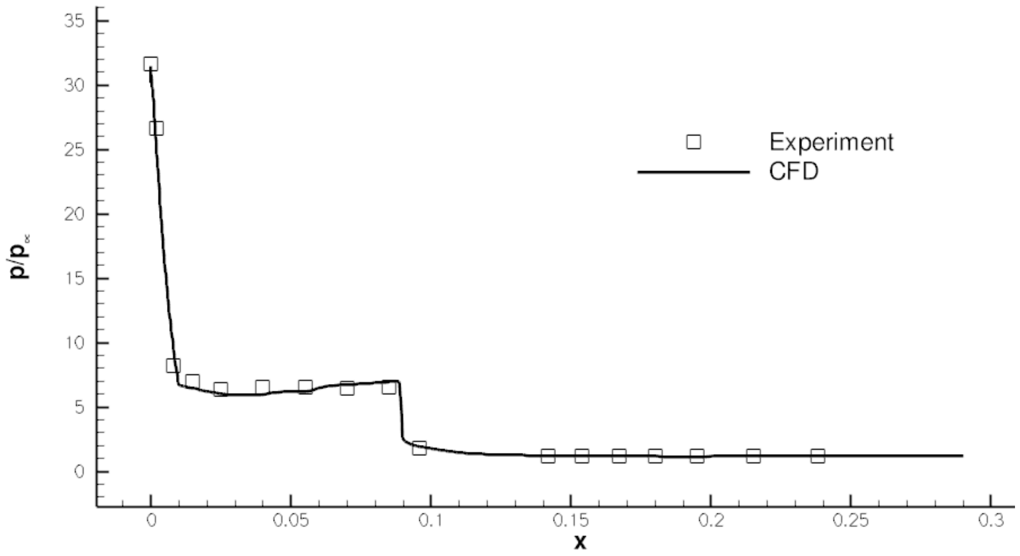


Fig. 1 pressure distribution along the upper surface of the space shuttle symmetric plane

Applications in Teaching

Among the aerodynamic teaching, we notice that trans-sonic aerodynamic properties of airfoil and wing are difficult for students to understand, meanwhile, the lack of researching methods imposes restriction on teachers, so they tend to repeat books about this part without lively and vivid explanation. Using our solver, we can give exact results quickly, and present the visual pictures for students to understand this part of courses.

Development of Local Shock Wave on Airfoil. Figure 2 shows the development of local shock wave on NACA0012 with AOA 2 degree. We can see clearly that the variation of local shock wave against increasing Ma, so that the traditional saying "first above, extend slower; second below, extend quicker" will be more visual and concrete.

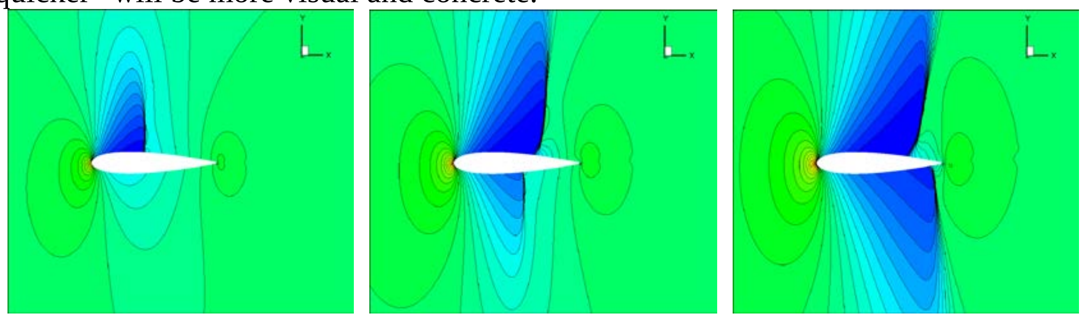
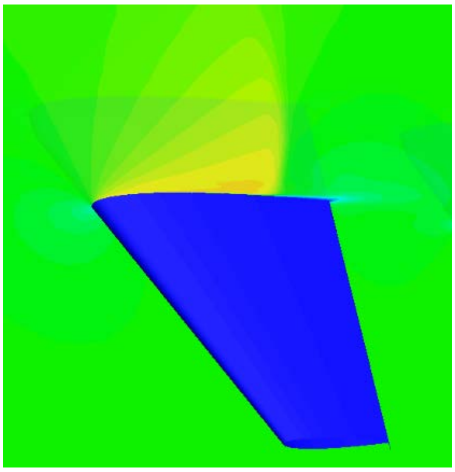
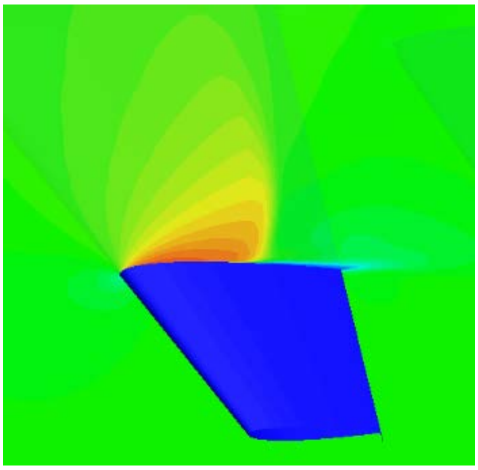
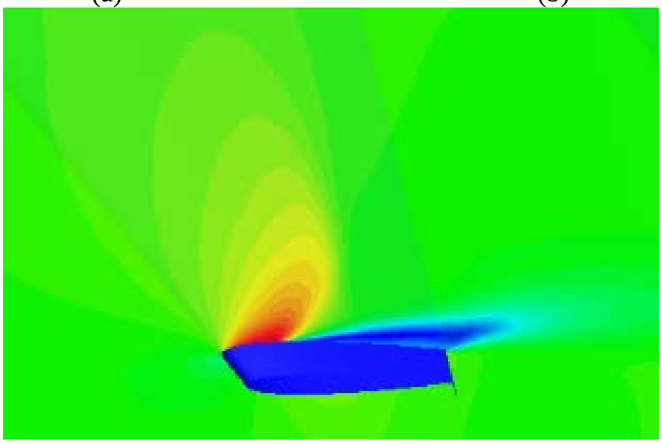

Ma=0.75 Ma=0.83 Ma=0.87

Fig.2 Development of Local Shock Wave on Airfoil

Development of Local Shock Wave on Wing. The development of local shock wave on three-dimensional wing is hard to explain, take swept-back wing for example, textbook [5] says "because of the effects on wing tip, the local super-sonic range may first appear nearby the wing tip, and develop local shock wave", but this depends on the type of airplane. Some advanced civil airplane, like B737, adopted unique wing design, and will first develop local shock wave nearby wing root; while some advanced 3G fighter is opposite. This situation need us to explain the basic reason of developing local shock wave, at the same time, we had better show the actual phenomenon to persuade students more effectively. Figure 4 shows the span section Mach clouds of some 3G fighter wing of which Ma=0.9, AOA is 5degree. We can easily conclude that, the more approaching wing tip, the more prominent of local super-sonic range, so obviously the local shock wave first developed nearby wing tip.

(a) (b)



(c)

Fig.3 Development of Local Shock Wave on Wing

Shock Wave Stall. Shock wave stall is caused by boundary layer separation induced by shock wave. Figure 4 shows the same wing of 3G fighter with 0.9Ma and 5 degree AOA, chosing the section near the wing tip. Rear part of upper surface forms clearly the reverse stream line, locating just behind the shock wave, this is boundary layer separation, which takes place from the lower laminar boundary, and because of the rise in pressure cross shock wave, comes out boundary separation.

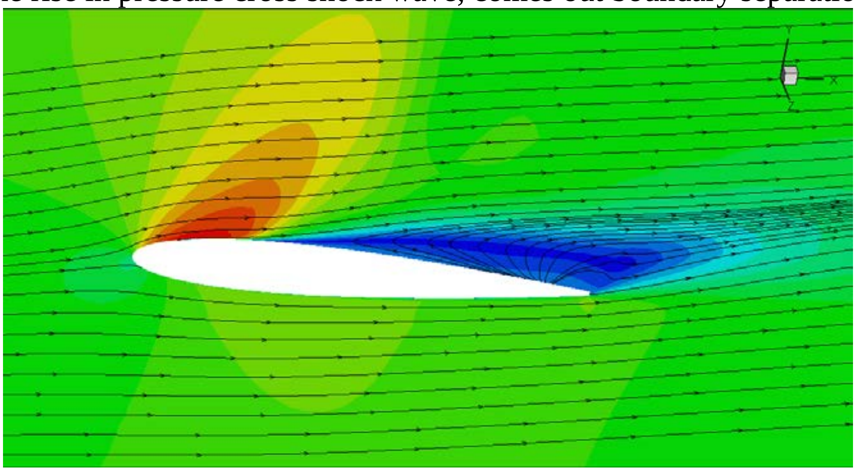


Fig.4 shock wave stall

Variation of Pressure Center among Sub and Trans-Sonic Range. Figure 5 shows the variation of CP of some 3G fighter with AOA 5 degree and Ma=0.3-1.3, the original point is the leading point of the wing. From the picture we can see that among sub-sonic range, the CP will move slightly forwards; when entering trans-sonic range, the CP will drastically move rearwards. This graph accompanied by theoretical analysis can give students a direct explanation which is more convenient to understand.

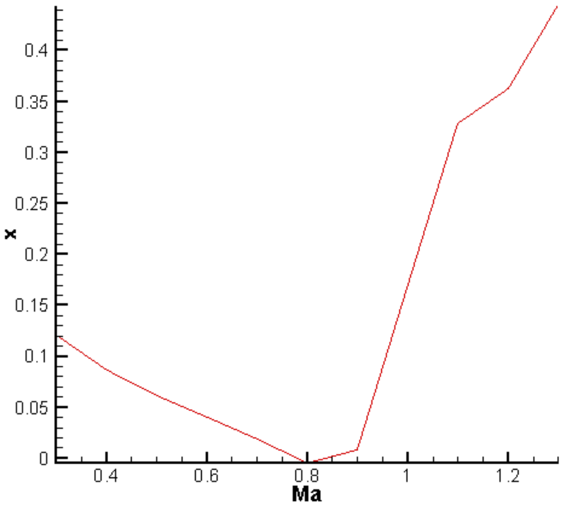


Fig.5 variation of CP

Summary

From the above we can see that, this solver can considerably perfect the analyzing means of teachers, extend the methods of solving complicated problems. But it must be pointed out that, we completed the teaching example above on our personal computer (include 1 high performance GPU), the hardware limited the full computational capability. If allocating high performance hardware, we can give satisfactory solutions to pilot-concerning problems such as "aerodynamic properties among large AOA flight".

References

[1] Yan Chao. On the Achievements and Prospects for the Methods of Computational Fluid Dynamics [J]. Advances in Mechanics, Vol.41, No.5, 2011.

[2] Wei Cao, Chuanfu Xu, Zhenghua Wang, et al. CPU/GPU computing for a multi-block structured grid based high-order flow solver on a large heterogeneous system[J]. Cluster Computer, 2014.

[3] NVIDIA Corporation. NVIDIA CUDA C Programming Guide Version 5.5, 2013.

[4] Li Suxun. Hypersonic Flow Properties of Typical Configuration [M]. Beijing: Press of Defense Industry, 2007.

[5] Liu Yongxue. Aeridynamics [M], Press of Aviation University of Air Force, 2016.