Application of Agglomeration Multigrid Method in GSM-CFD Solver

†*Jianyao Yao[1] and G.R. Liu[2]

[1]College of Aerospace Engineering, Chongqing University, Chongqing, China
[2]Department of Aerospace Engineering and Engineering Mechanics, University of Cincinnati, USA

†*Presenting and corresponding author: yaojianyao@cqu.edu.cn

Abstract

The GSM-CFD solver was firstly proposed by Liu and Xu [1] in 2008 for solving compressible flow problems using unstructured triangular meshes. Strategies for constructions of smoothing domains and algorithms for gradient approximations were introduced in detail for 2D problems. Different from the conventional numerical methods for gradient approximation, the first- and second-order spatial derivatives are calculated using GSM in a compact and conservative manner. Very good results can be achieved efficiently by properly choosing the smoothing functions and quadrature schemes on even highly distorted unstructured meshes. In a later paper [2], an adaptive GSM-CFD solver was developed to better capture the shock location in 2D inviscid transonic flows. More recently, this adaptive solver was improved to solve the shock-wave boundary-layer interaction problems [3]. In both methods, the GSM-CFD solver is coupled with a solution-based adaptive mesher to automatically generate finer mesh around the shock-wave zone.

Besides the compressible flows, the GSM-CFD solver can also be used to solve the steady-state and transient incompressible flow problems [4, 5, 6] using the artificial compressibility method [7]. The incompressible GSM-CFD solver was used to solve 2D pulsatile blood flow within rigid vessel, and the flow phenomena in stenosed vessels and arteries with aneurysm were investigated [5]. The 3D incompressible GSM-CFD solver was firstly introduced in [6], and the simulations of blood flow in carotid bifurcation were used as the numerical example. Same as the compressible flow problems, the incompressible GSM-CFD solver inherits the favorable features including being stable, accurate and efficient, and more importantly, insensitive to mesh qualities. These features make the newly developed GSM-CFD solver a competitive alternative to conventional finite volume or finite element methods.

Since we have flexible choices of smoothing functions and integration schemes, it is always an important issue to find a reasonable balance between efficiency and accuracy. The piecewise constant smoothing function with edge-midpoint integration scheme is mostly used in the previous studies due to its simplicity, and it works very well on the unstructured triangular meshes. However, recent research [3] indicated that the accuracy of this method on hybrid mesh was not as accurate as on triangular mesh, and an one-point quadrature scheme was devised to improve the accuracy of GSM by adding more integration points along the smoothing domain boundaries. A matrix-based algorithm and the corresponding generalized edge-based data structure were also devised [3, 6] to further improve the numerical efficiency of gradient approximation using GSM. The usage of piecewise linear smoothing function was introduced for triangular mesh and compared with conventional piecewise constant one [8]. It was concluded that the accuracy had improved significantly with the application of linearly-weighted smoothing operation instead of linear interpolation.

To further improve the numerical efficiency of the GSM-CFD solver, the geometric multigrid method [9, 10, 11] is adopted to accelerate convergence. The coarser mesh is generated by

Table 1: Effects of multigrid technique

Grid level	Time/iteration	Total number of iteration	Total computation time(s)
0	0.078	4219	329
1	0.177	1700	302
2	0.195	1092	212
3	0.199	817	163
4	0.203	729	148

merging neighboring control volumes around a seed point, and new vertex-based control volumes are created correspondingly. Therefore, there is no need to generate independent meshes for the coarse levels. There are two advantages of using multigrid method: (1) larger time steps can be used on the coarser meshes to reduce the numerical effort; and, (2) the low frequency components of the solution error are effectively damped on coarse meshes, and the convergence is significantly accelerated.

In the proposed solver, the first-order accurate discretization of the convective terms is employed, and the viscous terms are calculated using the average of gradients on the coarse meshes. The GSM approximations of flow variables are only employed on the finest mesh to ensure the accuracy of the final results.

The laminar flow around NACA 0012 airfoil is used as numerical example to show the effectiveness of the multigrid GSM-CFD solver. The comparisons among different mesh levels are summarized in Table 1. We can see that the multigrid can greatly accelerate convergence and reduce the total computational time.

Acknowledgments

This research is supported by National Natural Science Foundation of China under Grant No. 11502037.

References

[1] G. R. Liu and G. X. Xu. A gradient smoothing method (gsm) for fluid dynamics problems. *International Journal for Numerical Methods in Fluids*, 58(10):1101–1133, Dec 2008.

[2] G. X. Xu, G. R. Liu, and A. Tani. An adaptive gradient smoothing method (gsm) for fluid dynamics problems. *International Journal for Numerical Methods in Fluids*, 62:499–529, 2010.

[3] J. Yao, T. Lin, GR Liu, and CL Chen. An adaptive gsm-cfd solver and its application to shock-wave boundary layer interaction. *International Journal of Numerical Methods for Heat & Fluid Flow*, 25(6):1282–1310, 2015.

[4] G. X. Xu, E. Li, V. Tan, and G. R. Liu. Simulation of steady and unsteady incompressible flow using gradient smoothing method (gsm). *Computers & Structures*, 90-91:131–144, Jan 2012.

[5] E. Li, G.R. Liu, G. X. Xu, T. Vincent, and Z.C. He. Numerical modeling and simulation of pulsatile blood flow in rigid vessel using gradient smoothing method. *Engineering Analysis with Boundary Elements*, 36(3):322–334, Mar 2012.

[6] J. Yao and G. R. Liu. A matrix-form gsm-cfd solver for incompressible fluids and its application to hemodynamics. *Computational Mechanics*, 54(4):999–1012, 2014.

[7] A. J. Chorin. A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics*, 135(2):118–125, 1997.

[8] E. Li, V. Tan, G. X. Xu, G. R. Liu, and Z. C. He. A novel linearly-weighted gradient smoothing

method (lwgsm) in the simulation of fluid dynamics problem. *Computers & Fluids*, 50:104–119, 2011.

[9] D. J. Mavriplis. Multigrid solution of the two-dimensional Euler equations on unstructured triangular meshes. *AIAA Journal*, 26(7):824–831, Jul 1988.

[10] D. J. Mavriplis and V. Venkatakrishnan. Agglomeration multigrid for two-dimensional viscous flows. *Computers & Fluids*, 24(5):553–570, 1995.

[11] T. D. Economon, F. Palacios, J. J. Alonso, et al. Towards high-performance optimizations of the unstructured open-source SU2 suite. *AIAA Paper*, 1949:2015, 2015.