Accepted Manuscript

Title: Developing a parallel density-based implicit solver with mesh deformation in OpenFOAM

Author: Xiang Gao Chuanfu Xu Yidao Dong Min Xiong Dali Li Zhenghua Wang Xiaogang Deng

PII: S1877-7503(18)30161-3
DOI: https://doi.org/doi:10.1016/j.jocs.2018.07.006
Reference: JOCS 901

To appear in:

Received date: 12-2-2018
Revised date: 27-7-2018
Accepted date: 27-7-2018

Please cite this article as: Xiang Gao, Chuanfu Xu, Yidao Dong, Min Xiong, Dali Li, Zhenghua Wang, Xiaogang Deng, Developing a parallel density-based implicit solver with mesh deformation in OpenFOAM, *<![CDATA[Journal of Computational Science]]>* (2018), https://doi.org/10.1016/j.jocs.2018.07.006

Developing a parallel density-based implicit solver with mesh deformation in OpenFOAM

Xiang Gao[a], Chuanfu Xu[a,*], Yidao Dong[b], Min Xiong[a], Dali Li[b], Zhenghua Wang[a,c], Xiaogang Deng[a,b]

[a]*College of Computer, National University of Defense Technology, Changsha 410073, China*
[b]*College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China*
[c]*State Key Laboratory of Aerodynamics, P.O. Box 211, Mianyang 621000, China*

Abstract

Numerical simulations like the aeroelastic computation and aerodynamic shape optimization usually involve moving boundaries, and are always carried out with supersonic compressible flows. In this paper, based on the OpenFOAM platform, we present a parallel density-based implicit solver with mesh deformation to address this kind of problems, and fill the gap for the deficiencies of implicit solvers. The core implementation details of the solver based on OpenFOAM are introduced. The Godunov type schemes in ALE (Arbitrary Lagrangian Eulerian) formulation and the dual-time stepping LU-SGS (Lower-Upper Symmetric Gauss-Seidel) algorithm are applied to solve the Navier-Stokes equations with turbulence model. For dynamic meshes, the novel parallel mesh deformation approach based on the Support Vector Machine is implemented. Four typical cases are tested to demonstrate the validity and parallel efficiency of the proposed solver. The result shows that the scalability of the solver reaches its limit at around 128 MPI (Message Passing Interface) tasks, and the cost of MPI communication is the bottleneck for large-scale simulations. Overall, our solver is applicable for various dynamic mesh compressible problems and all types of grids.

Keywords: OpenFOAM, Density-based, Mesh deformation, Support vector

*Corresponding author
Email address: `xuchuanfu@nudt.edu.cn` (Chuanfu Xu)

Preprint submitted to Journal of Computational Science *July 28, 2018*