Implicit fully mesh-less method for compressible viscous flow calculations

Y. Hashemi, A. Jahangirian*

Department of Aerospace Engineering, Center of Excellence in Computational Aerospace, Amir Kabir University of Technology, 424 Hafez Avenue, Tehran, Iran

ARTICLE INFO

Article history:
Received 1 November 2009
Received in revised form 27 May 2010

Keywords:
Dual time implicit scheme
Mesh-less method
Compressible viscous flow

ABSTRACT

A dual-time implicit mesh-less scheme is developed for solution of governing viscous flow equations. The computational efficiency of the method is enhanced by adopting accelerating techniques such as local time stepping and residual smoothing. The Taylor series least square method is used for approximation of derivatives at each node which leads to a central difference spatial discretization. The capabilities of the method are demonstrated by flow computations about standard cases at subsonic and transonic laminar flow conditions. Results are presented which indicate good agreements with the alternative numerical and experimental data. The computational time is considerably reduced when using the proposed mesh-less method compared with the explicit mesh-less and finite-volume counterparts using the same distribution of points.

© 2010 Elsevier B.V. All rights reserved.

1. Introduction

The main problem of computational fluid dynamics (CFD) for numerical flow simulation around complex geometries is the generation of a quality mesh. In general, the numerical mesh generation methods are classified as structured and unstructured methods [1-3], both of which enable the generation of grid points and their connections. Each of these methods has its own advantages and disadvantages; the answer to which method is preferable depends on the problem to be solved. Difficulties in generating quality meshes, particularly for viscous flow simulations have recently attracted much interest towards the so-called mesh-less methods. These methods only use clouds of nodes in the influence domain of every node. Thus, they don't require the nodes to be connected to form a mesh and decrease the difficulty of meshing particularly around complex geometries. The flow derivatives are calculated using different approximation methods like least squares. Some attempts have been made in order to use the mesh-less idea locally for computation of boundary values within a mesh based framework [4,5]. Lohner has shown that generation of a finite-point mesh is an order of magnitude faster as compared to an unstructured mesh for a 3D configuration [6]. Thus, development of fully mesh-less methods for flow simulations has shown great promise. Additionally mesh-less methods have advantages regarding the moving boundary and large deformations compared with mesh based algorithms. Besides these advantages, they have some drawbacks i.e. shape functions are rational functions which require a high-order integration scheme to be correctly computed. The mesh-less method is neither a finite-difference nor a finite-volume type approach. Therefore, metrics, areas, or volumes are not computed. To minimize the lack of conservation such as the finite difference method, a higher-order representation is used to improve solution accuracy. In particular, the computational cost of mesh-less methods is usually higher than mesh based ones at any iteration. Thus, researchers have focused on decreasing the computational cost of mesh-less methods by convergence accelerating techniques such as multi-cloud approach [7]. Several mesh-less methods have been proposed in the literature [8,9]. Explicit time discretizations have been commonly used in most mesh-less methods.

* Corresponding author.
E-mail address: ajahan@aut.ac.ir (A. Jahangirian).

0377-0427/ - see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.cam.2010.08.002