Aerospace Science and Technology 64 (2017) 161–170

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

An adaptive fully-Lagrangian meshless method for incompressible laminar flow airfoil studies

Sh. Khorasanizade, J.M.M. Sousa *

IDMEC, Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais, 1049-001 Lisboa, Portugal

A R T I C L E   I N F O

Article history:
Received 29 July 2016
Received in revised form 26 January 2017
Accepted 29 January 2017
Available online 7 February 2017

Keywords:
Meshless method
Spatial adaptivity
Lagrangian approach
Smoothed Particle Hydrodynamics
Airfoil studies
Incompressible laminar flow

A B S T R A C T

Laminar incompressible flow around a NACA0012 airfoil placed in a free-stream at various incidences has been revisited with the use of a fully-Lagrangian meshless method based on a Smoothed Particle Hydrodynamics (SPH) formulation. Spatial adaptivity has been incorporated in the scheme via splitting and merging of SPH particles employing zonal criteria. In addition, a novel algorithm has been proposed here so that particle merging may be achieved, ensuring the robustness, accuracy and efficiency of the computational simulations. The results obtained have been benchmarked against available data from mesh-based methods. Good agreement has been found both in steady and unsteady flow regimes. Overall, the present work demonstrated the effectiveness and competitiveness of this meshfree approach for detailed studies in aerodynamics.

1. Introduction

Over the past twenty years, the role of numerical investigations of incompressible aerodynamic flows in industrial applications has exponentially grown in importance. During this period, many methods have been made available based on the solution of the Navier–Stokes equations using calculation meshes [1–5]. In this class of schemes, mesh generation for complex geometries often represents a considerable task by itself. Moreover, in this case, handling moving boundaries in general body motion is not as straightforward as one might expect beforehand. Contrastingly, and among other benefits, such difficulties are generally easier to cope with by using the more recently developed meshfree schemes. Detailed comparisons between both classes of methods can be found in the literature [6–8].

Amongst the known panoply of meshfree methods, Smoothed Particle Hydrodynamics (SPH) is nowadays one of the most widely used, both in fluid and solid mechanics studies. As a consequence of its intrinsic meshless nature, SPH is well-suited for problems involving complex geometries, as well as moving bodies. By following this approach, the computational domain is discretized using moving calculation nodes, denoted as particles. These Lagrangian entities carry both fluid and flow information. The method was firstly developed for the study of astrophysical phenomena [9], but later it was applied to incompressible flow problems [10] also. Many researchers have succeeded to take advantage of SPH for detailed investigations of various industrial problems, ranging from the analysis of free surface flows [11] to aerosol dispersion [8], and bird strikes on jet-engines [12]. A comprehensive review on the applications and future perspectives of SPH has been very recently presented by Shadloo et al. [13]. However, clear deficiencies have been identified by these authors when aiming to employ this method to obtain accurate predictions of thin boundary layers, such as those occurring in the flow around an airfoil.

Aerodynamic characteristics and the underlying physics of air flow past a NACA0012 airfoil have been studied earlier by various researchers [1–4,14]. The foregoing studies have mostly focused on the laminar flow regime, thus avoiding additional uncertainties concerning turbulence modeling. Here, the proposed SPH formulation is to be validated for the numerical simulation of laminar flow around the above-mentioned airfoil placed in a free-stream at various angles of attack. A variety of procedures may be followed to adequately deal with the different types of boundaries [15] to be considered in such flow problem. The open boundary treatment proposed by Khorasanizade and Sousa [16] for inflow/outflow, together with the solid boundary implementation also recommended by the same authors to handle the no-slip condition [17], will be used in the present work.

Nevertheless, the large computational domains demanded by the study of free-stream configurations push the use of an equally large number of particles in SPH. It has been shown that the required number of particles may be reduced by the use of an

* Corresponding author. Fax: +351 21 849 9242.
E-mail addresses: shahab.khorasanizade@ist.utl.pt (Sh. Khorasanizade), msousa@ist.utl.pt (J.M.M. Sousa).

http://dx.doi.org/10.1016/j.ast.2017.01.027
1270-9638/

adaptive algorithm [18,19], without sacrificing local spatial resolution. By this procedure, each particle may be successively split into four smaller ones where demanded for accuracy requirements. However, as the SPH particles are convected throughout the domain, certain flow regimes may eventually lead to the interaction of minute particles with massive ones, which ultimately may compromise the stability and accuracy of the method. In addition, this process results in a continuous increase of the total number of particles during the simulation. On the other hand, merging fine particles into coarser ones, where appropriate, refrains the steady rise in computational cost resulting from the growing number of calculation points. Some researchers have proposed coalescing two adjacent particles [20], whereas others suggested merging of equally sized particles in the vicinity of one central particle [21]. A different approach was proposed by Barcarolo et al. [19], in which particles that were created within regions of interest are erased once these are no longer needed, but its usage may face difficulties when dealing with flow problems where high mixing of particles occurs.

It may be anticipated that retrieving the original arrangement before splitting into four particles would be beneficial from both stability and efficiency viewpoints. However, the scheme presented by Xiong and Xu [21] for this purpose occasionally leads to undesirable particle clumping, as the resultant merged particle might end up being placed too close to a large one in its neighborhood. In order to avoid this shortcoming, a novel algorithm based on the interaction of particles is presented in this work to efficiently handle wide-range adaptive spatial resolution in SPH, while maintaining the computational effort as low as possible.

The paper is organized as follows. In section 2, a general description of the SPH methodology and governing equations is given before the explanation of the merging algorithm. In section 3, the geometrical configuration and simulation characteristics are defined, followed by a detailed presentation and discussion of the results. The main goal of the present work consists in the validation of the state-of-the-art meshfree numerical tool HAdynaSPH [16–18] for the study of steady and unsteady two-dimensional laminar flow problems in aerodynamics. This code has been developed in Fortran and compiled with Intel® Fortran Composer XE v12.1 for Linux. All the results reported herein have been obtained on a single core of a desktop machine equipped with an Intel Core i7-2600K @ 3.4 GHz quad-core CPU.

2. Numerical tool

2.1. Methodology and governing equations

The meshless nature of SPH is acquired by making use of the interaction of each calculation node with its neighbors rather than their connectivity (as in mesh-based methods). This is achieved by employing a weighting function (the kernel), which has a compact support of size h, to calculate domain values at each discrete fluid particle by means of the information contained in surrounding particles [6,22]. From a physical perspective, h represents a smoothing length, expressing the characteristic length of the domain of influence of a fluid particle at a position $\boldsymbol{r}$. Accordingly, the gradient of the kernel is used to calculate the gradients of domain values at a particle location from its neighbors. Hence, a function f and its gradient ∇f at any discrete point i are obtained using the following convolution summations:

$$f_i = \sum_j \frac{m_j}{\rho_j} f_j w_{ij}, \qquad \nabla f_i = \sum_j \frac{m_j}{\rho_j}(f_j - f_i)\nabla w_{ij}, \tag{1}$$

where i and j denote the target and its neighboring domain particle, respectively, m and ρ stand for the particle mass and density, respectively, and w_{ij} is the discrete kernel function centered on i as calculated at j. Khorasanizade and Sousa [23] took advantage of the so-called "negative formula", as given in Eq. (1), to ensure high accuracy and robustness of their numerical scheme in divergence-free flow simulations. Although it has been shown that this formulation does not strictly conserve global momentum, it is the preferred choice in ISPH calculations [24] due to the resulting numerical stability. In the present work, a Wendland C6 kernel [25] has been used as weighting function, with a smoothing ratio (i.e., the quotient of the particle spacing and the smoothing length) equal to 1.95.

A Lagrangian description of the flow is also obtained in SPH as the discrete particles move throughout the fluid domain by satisfying Newton's second law. The acceleration of the fluid is calculated in the present study by solving the incompressible, laminar Navier–Stokes equations in Lagrangian form following the SPH formalism. Hence, the complete set of governing equations is given by:

$$\begin{cases} \nabla \cdot \boldsymbol{u}_f = 0, \\ \dfrac{d\boldsymbol{u}_f}{dt} = -\dfrac{\nabla p}{\rho} + \nabla.(\nu \nabla \boldsymbol{u}_f) + \boldsymbol{F}, \\ \dfrac{d\boldsymbol{r}}{dt} = \boldsymbol{u}_f, \end{cases} \tag{2}$$

where $\boldsymbol{u}_f$ denotes the fluid velocity vector, p is the pressure, ρ and ν stand for the density and viscosity of the fluid, respectively, $\boldsymbol{F}$ represents the external forces per unit mass (e.g., gravity or other driving forces) and $\boldsymbol{r}$ is the position vector. Air at standard atmospheric temperature and pressure conditions was selected as working fluid in the present study. Flow incompressibility is enforced here by solving Eq. (2) employing a projection method. In addition, the time integration scheme used is of prediction – correction type [26].

For more details on the current SPH algorithm, including the particle shifting (resettlement) procedure and a suitable treatment of boundaries, the reader is referred to previous work [16–18,23]. However, owing to its relevance in this study, the treatment of sharp edges deserves additional explanations because it is a nontrivial issue in particle methods as pointed out, e.g., by Meringolo et al. [27] and Rossi et al. [28]. This is facilitated here by the use of the Segmented Boundary Algorithm (SBA), proposed by Khorasanizade and Sousa [17] for the treatment of solid boundaries of complex shape. In this approach, the concept of "image particles" to minimize the consequences of kernel truncation occurring in vicinity of boundaries [6,22] is used together with segmentation algorithms.

Fig. 1 illustrates the application of SBA to the treatment of a sharp edge, such as the trailing edge of an airfoil. A central section of the *corner* region in the fluid, denoted by $corner_c$, is established by the line extensions of the segments defining the vertex of the edge. The remaining sections of the *corner* region, namely $corner_u$ and $corner_l$, are found by using also the lines normal to the aforementioned segments. For all fluid particles located in any of these three sections, only those within $corner_c$ produce "image particles", as this is the condition leading to the placement of such particles inside of the solid body (shaded area) only. Hence, the fluid particles in the whole *corner* region only interact with "image particles" created from those at $corner_c$, besides the fluid particles inside their own compact support surrounding the edge. In the case of the fluid particles located in the vicinity of the sharp edge, but outside the *corner* region, a "+" side and a "−" side of the normal to the line segment are defined with respect to each particle describing the edge geometry, according to the counterclockwise direction