Advances in Engineering Software 000 (2016) 1–9

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

A local numerical solution of a fluid-flow problem on an irregular domain

G. Kosec*

Jožef Stefan Institute, Department of Communication Systems, Jamova 39 1000 Ljubljana, Slovenia

ARTICLE INFO

Article history:
Received 21 December 2015
Revised 7 March 2016
Accepted 16 May 2016
Available online xxx

Keywords:
Navier-Stokes
Meshless local strong form method
Local pressure-velocity coupling
Random geometry
Lid driven
Backward facing step

ABSTRACT

This paper deals with a numerical solution of an incompressible Navier-Stokes flow on non-uniform domains. The numerical solution procedure comprises the Meshless Local Strong Form Method for spatial discretization, explicit time stepping, local pressure-velocity coupling and an algorithm for positioning of computational nodes inspired by Smoothed Particles Hydrodynamics method. The presented numerical approach is demonstrated by solving a lid driven cavity flow and backward facing step problems, first on regular nodal distributions up to 315,844 (562×562) nodes and then on domain filled with randomly generated obstacles. It is demonstrated that the presented solution procedure is accurate, stable, convergent, and it can effectively solve the fluid flow problem on complex geometries. The results are presented in terms of velocity profiles, convergence plots, and stability analyses.

1. Introduction

Computational fluid dynamics (CFD) is a field of a great interest among researchers in many fields of science, e.g. studying mathematical fundaments of numerical methods, developing novel physical models, improving computer implementations, and many others. Pushing the limits of all the involved fields of science helps community to deepen the understanding of several natural and technological phenomena. Weather forecast, ocean dynamics, water transport, casting, various energetic studies, etc., are just few examples where fluid dynamics plays a crucial role. The core problem of the CFD is solving the Navier-Stokes Equation [1] or its variants, e.g. Darcy or Brinkman equation for flow in porous media. This paper focuses on a solution of the Navier-Stokes equation in a randomly generated domain with a local numerical approach.

Usually, numerical methods such as the Finite Volume Method (FVM), Finite Difference Method (FDM), or the Finite Element Method (FEM) are typically used for solving fluid flow problems. Although classical methods, especially FEM, offer several advanced features, the meshing of realistic domains still remains one of the most cumbersome and time-demanding step in the entire numerical solution process, since it often involves a significant user's assistance. In past few years the coupling of Computer Aided Design (CAD) and FEM analysis [2] alleviates that burning problem. The approach is also referred to as an isogeometric analysis and is focused on integration of FEM into conventional Non-Uniform Rational Basis Splines (NURBS) based CAD environments. On the other hand, the most intuitive and straightforward to implement is definitely the FDM approach that performs excellent as long as the treated domain can be described with an equidistant orthogonal mesh, which unfortunately covers only limited spectra of problems.

A promising alternative is a class of meshless methods (MM) that are based on scattered discretization nodes. MMs originate in the seventies with Smoothed Particles Hydrodynamics (SPH) [3] and develop further with the Diffuse Element Method (DEM), the Meshless Petrov-Galerkin method (MPG), the Element Free Galerkin method (EFG), etc. [4]. The SPH, an Eulerian kernel based approximation method, is an effective tool for simulations of problems where mesh-based method fail, for example breaking waves, gas problems and many more. However, SPH suffers from inconsistency due to the combination of Eulerian kernel and Lagrangian description of motion. The more consistent particle method with Lagrangian kernels has been later introduced for solution of solid mechanics problems [5].

In this paper, one of the simplest class of MMs, Meshless Local Strong Form Method (MLSM), a generalization of methods which are in literature also known as Diffuse Approximate Method (DAM) [6], Local Radial Basis Function Collocation Methods (LRBFCM) [7], Generalized FDM [8], Collocated discrete least squares (CDLS) meshless [9], etc., is used. Although each of the named methods poses some unique properties, the basic concept of all local strong form methods is similar, namely, to approximate treated

* Corresponding author.
E-mail address: gkosec@ijs.si

http://dx.doi.org/10.1016/j.advengsoft.2016.05.010
0965-9978/

Please cite this article as: G. Kosec, A local numerical solution of a fluid-flow problem on an irregular domain, Advances in Engineering Software (2016), http://dx.doi.org/10.1016/j.advengsoft.2016.05.010

fields with nodal trial functions over the local support domain. The nodal trial function is then used to evaluate various operators, e.g. derivation, integration, and after all, approximation of a considered field in arbitrary position. The MLSM could easily be understood as a meshless generalization of the FDM, however much more powerful. The MLSM has an ambition to avoid using pre-defined relations between nodes and shift this task into the solution procedure. The final goal of such an approach is higher flexibility in complex domains, moving boundaries and nodal adaptivity.

There are several publications regarding adaptive MM. The h-refinement, i.e. the adaptivity in terms of adding and/or removing nodes on/from the domain has been demonstrated with the global Radial Basis Function Collocation Method [10] in solution of nearly singular Partial Differential Equations (PDE), as well as local MMs in solution of coupled Burgers' equation [11] and torsion problem [12]. The meshless r-refinement approach, where the positions of the nodes are adjusted to obtain an optimal approximation with the total number of the nodes unchanged, has been demonstrated in solution of phase field model [13]. In general, the meshless adaptivity has been thoroughly demonstrated in crack propagation problems [14–17]. An important part of the adaptivity is the error estimate that determines the nodal density that has been discussed in [11,18].

Although the meshless methods do not require any topological relations between nodes and even randomly distributed nodes could be used [19], it is well-known that using regularly distributed nodes leads to more accurate and more stable results [20–22], which is also confirmed in this paper. Therefore, despite meshless seeming robustness regarding the nodal distribution, a certain effort has to be invested into the positioning of the nodes and this paper, to some extent, deals with this problem.

The rest of the paper is organized as follows; in Section 2 the MLSM principle is explained, in Section 3 the lid driven cavity and backward facing step problems together with base elements of the solution procedure are presented, Section 4 is focused on discussion of results, and finally, paper offers some conclusions and guidelines for future work in last section.

This paper is extension of results presented on Ninth International Conference on Engineering Computational Technology [23].

2. Numerical methodology

2.1. Meshless LOCAL STRONG FORM METHOds (MLSM)

The core of MLSM presented in this paper is a local approximation of a considered field over the overlapping local support domains, i.e. in each node a considered field is approximated over a small local sub-set of neighbouring N_S nodes. The trial function is thus introduced as

$$\theta(\mathbf{p}) = \sum_{n=1}^{N_B} \alpha_n \Psi_n(\mathbf{p}), \tag{1}$$

with N_B, α_n, Ψ_n, $\mathbf{p}(p_x, p_y)$ standing for the number of basis functions, approximation coefficients, basis functions and the position vector, respectively. The type of approximation, the size of support domain, and the type and number of basis function can be general.

Although the selection of basis function Ψ_n is general, several researchers follow the results from Franke's analysis [24] and use Hardy's Multiquadrics, however in this work the monomials are used based on the results presented in [25]. The goal here is to solve a Navier-Stokes equation, i.e. a second order PDE, and to obtain non-trivial first and second derivatives a minimal basis of five monomials $(1, p_x, p_y, p_x^2, p_y^2)$ is used. Therefore, to determine corresponding coefficients at least five support nodes are required. In such setup, i.e. support domain size is the same as the number of basis functions $(N_S = N_B)$, the determination of coefficients α_n simplifies to solving a system of linear equations that results from expressing Eq. (1) in all support nodes. The system can be written in vector form as

$$\boldsymbol{\theta} = \boldsymbol{\Psi}\boldsymbol{\alpha}, \tag{2}$$

where $\boldsymbol{\theta}$ stand for field values in support nodes, $\boldsymbol{\Psi}$ basis matrix $(\Psi_{ij} = \Psi_i(\mathbf{p}_j))$ and $\boldsymbol{\alpha}$ vector of coefficients. The LRBFCM that has been recently used in various problems [26, 27] uses such collocation on different sizes of support domain, depending on the problem tackled.

If the number of support nodes is higher than the number of basis functions $N_S > N_B$ Weighted Least Squares (WLS) approximation is used to solve over-determined system (2), again, constructed by expressing (1) in all support nodes. An example of such approach is DAM [6] that was originally formulated to solve fluid flow in porous media. DAM uses six monomials for basis and nine nodded support domains to evaluate first and second derivatives of physical fields required to solve problem at hand. Note that WLS with a Gaussian weighting

$$\mathrm{W}(\mathbf{p}) = \exp\left(-\left(\frac{\|\mathbf{p}\|}{\sigma\, p_{\min}}\right)^2\right) \tag{3}$$

is used, where σ stands for weight parameter and $p_{\min}$ for the distance to the first support domain node.

Our goal is to apply partial operator on a considered field

$$L\theta(\mathbf{p}) = \sum_{n=1}^{N_B} \alpha_n L\Psi_n(\mathbf{p}), \tag{4}$$

where L stands for general differential operator. Considering Eq. (4) by using explicit computation of approximation coefficients $\boldsymbol{\alpha} = \boldsymbol{\Psi}^{-1}\boldsymbol{\theta}$ results in

$$L\theta(\mathbf{p}) = \sum_{n=1}^{N_B} \left(\sum_{m=1}^{N_S} \Psi_{nm}^{-1}\theta_m\right) L\Psi_n(\mathbf{p}). \tag{5}$$

Using merely few summation rules the Eq. (5) can be rewritten in a more convenient form

$$L\theta(\mathbf{p}) = \sum_{m=1}^{N_S} \chi_m^L(\mathbf{p})\theta(\mathbf{p}_m), \tag{6}$$

where the shape function χ_m^L is introduced as

$$\chi_m^L(\mathbf{p}) = \sum_{n=1}^{N_B} \Psi_{nm}^{-1} L\Psi_n(\mathbf{p}), \tag{7}$$

with $\boldsymbol{\Psi}^{-1}$standing for inverse/pseudo inverse of the approximation system matrix.

The presented formulation is convenient for implementation since most of the complex operations are performed only when nodal topology changes, i.e. when the system (2) has to be re-evaluated. In the main simulation, the pre-computed shape functions are then convoluted with the vector of values in the support to evaluate the desired operator, refer to Eq. (16) for example. The presented MLSM approach is even easier to handle than the FDM, however despite its simplicity it offers many possibilities for treating challenging cases, e.g. nodal adaptivity to address regions with sharp discontinuities or p-adaptivity to treat obscure anomalies in physical field. The stability versus computation complexity and accuracy can be regulated simply by changing number of support nodes, etc. All these features can be controlled on the fly during the simulation.

Please cite this article as: G. Kosec, A local numerical solution of a fluid-flow problem on an irregular domain, Advances in Engineering Software (2016), http://dx.doi.org/10.1016/j.advengsoft.2016.05.010