Engineering Analysis with Boundary Elements 50 (2015) 59–68

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

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

A least squares based meshfree technique for the numerical solution of the flow of viscoelastic fluids: A node enrichment strategy

Mohsen Lashkarbolok [a], Ebrahim Jabbari [b,*], Jerry Westerweel [c]

[a] Department of Engineering, Golestan University, Golestan, Iran
[b] School of Civil Engineering, Iran University of Science and Technology, Iran
[c] Laboratory for Aero and Hydrodynamics, Delft University of Technology, Netherlands

ARTICLE INFO

Article history:
Received 1 January 2013
Received in revised form
5 July 2014
Accepted 31 July 2014
Available online 3 September 2014

Keywords:
Meshfree method
Least squares techniques
Viscoelastic fluid
Radial point interpolation methods
Artificial compressibility

ABSTRACT

A fully implicit least-squares-based meshfree method is used to solve the governing equations of viscoelastic fluid flow. Here, pressure is connected to the continuity equation by an artificial compressibility technique. A radial point interpolation method is used to construct the meshfree shape functions. The method is used to solve two benchmark problems. Thanks to the flexibility of meshfree methods in domain discretization, a simple node enrichment strategy is used to discrete the problem domain more purposefully. It is shown that the introduced enrichment process have a positive effect on the accuracy of the results.

1. Introduction

There are numerous investigations about the numerical simulation of viscoelastic fluid flow. Townsend used a finite difference technique to simulate the flow of viscoelastic fluid past stationary and rotating cylinders [1]. Viriyayuthakorn and Caswell simulated the flow of viscoelastic fluid using finite element method [2]. Darwish and Whiteman presented a staggered-grid, finite-volume method for the numerical simulation of isothermal viscoelastic liquids [3]. A comprehensive review in the field of numerical simulation of the viscoelastic fluid flow is performed by Owens and Timothy [4]. Most of the studies in this field are carried out by finite element and finite volume methods. These methods require mesh or grid to discretize domain of a problem. The subdivision of the domain into such components is laborious and difficult necessitating complex mesh or grid generation. Further, if adaptivity processes are used, generally large areas of the problem have to be remeshed [5]. The main feature of a meshfree method is its ability to more easily discretize the domain of a problem using some scattered nodes instead of elements or grids. This ability is a promising approach to perform an effective refinement procedure. In the present study, a least-squares-based meshfree technique referred to as Collocated Discrete Least Squares (CDLS), that was presented in [6], is used to solve the governing equations. A Radial Point Interpolation Method (RPIM) using Multi-Quadratic Radial Basis Functions (MQ-RBF) is used to construct meshfree shape functions. By this kind of function approximation we suppose that the exponential behavior of the stresses can be captured better in comparison with polynomial basis functions usually used in conventional numerical methods [4]. Here, the equations are considered to be solved implicitly. It means that the evolution of the pressure, velocity and stresses are computed simultaneously at each time step. To connect the pressure to the continuity equation, conventional artificial compressibility technique is used. Although the problem is assumed to be steady state, the governing equations are solved in time to the point that a steady-state solution is obtained. In this paper, a node enrichment strategy is used to discretize the domain of the problem according to some information from a prior solution. Some researchers have used adaptive refinement techniques to obtain more accurate solution in the simulation of viscoelastic fluid flow (for example [7,8]). However most of the efforts have been done using finite element and other mesh-based methods and authors believe that any adaptive process (including node enrichment) can be performed more simply in a meshfree technique. Adding nodes is simpler and more flexible than adding elements or grids to the computational domain of the problem. To assess the accuracy, the presented procedure is tested for two problems. In the first problem, an Oldroyd-B fluid creeping flow around a confined cylinder with a

* Corresponding author.
E-mail addresses: mlbolok@iust.ac.ir (M. Lashkarbolok), jabbari@iust.ac.ir (E. Jabbari), J.westerweel@tudelft.nl (J. Westerweel).

http://dx.doi.org/10.1016/j.enganabound.2014.07.011
0955-7997/

blockage ratio equal to 0.5 which is considered as a benchmark in computational rheology. In order to show how the method works at high Weissenberg numbers, in the second problem, the viscoelastic lid driven cavity flow is solved. In the proposed scheme, at each Weissenberg number, first a rather sparse nodal distribution is used for the solution, then using an error indicator, the positions of new nodes are obtained and finally the problem is solved using this enriched nodal distribution.

2. Governing equations

The governing equations for a two dimensional (x–y plane), steady and incompressible creeping flow of an Oldroyd-B fluid [9] in Cartesian coordinates is given by

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$-\frac{\partial p}{\partial x}+\eta_s\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}=0 \tag{2}$$

$$-\frac{\partial p}{\partial y}+\eta_s\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}=0 \tag{3}$$

$$\tau_{xx}+\lambda\left(u\frac{\partial \tau_{xx}}{\partial x}+v\frac{\partial \tau_{xx}}{\partial y}-2\tau_{xx}\frac{\partial u}{\partial x}-2\tau_{xy}\frac{\partial u}{\partial y}\right)=2\eta_p\frac{\partial u}{\partial x} \tag{4}$$

$$\tau_{xy}+\lambda\left(u\frac{\partial \tau_{xy}}{\partial x}+v\frac{\partial \tau_{xy}}{\partial y}-\tau_{xx}\frac{\partial v}{\partial x}-2\tau_{yy}\frac{\partial u}{\partial y}\right)=\eta_p\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) \tag{5}$$

$$\tau_{yy}+\lambda\left(u\frac{\partial \tau_{yy}}{\partial x}+v\frac{\partial \tau_{yy}}{\partial y}-2\tau_{yy}\frac{\partial v}{\partial y}-2\tau_{xy}\frac{\partial v}{\partial x}\right)=2\eta_p\frac{\partial v}{\partial y} \tag{6}$$

where u, v, p and η_s are velocity components in the x and y directions, pressure and solvent viscosity respectively. $\tau_{xx}, \tau_{xy}, \tau_{xy}$ are stress tensor components. η_p and λ are the polymer viscosity and relaxation time, respectively. Usually an additional parameter is defined to show the magnitude of solvent viscosity with respect to the total viscosity. It is defined by

$$\beta=\frac{\eta_s}{\eta_s+\eta_p}$$

The relevant non-dimensional quantity is the Weissenberg number, defined as

$$Wi=\frac{\lambda U}{R}$$

where U and R are characteristic velocity and characteristic length, respectively.

3. CDLS method

Consider the following differential equation:

$$L(u)=f \quad \text{in } \Omega \tag{7}$$

$$B(u)=g \quad \text{on } \Gamma_t, \tag{8}$$

$$u=\overline{u} \quad \text{on } \Gamma_u, \tag{9}$$

where u is the unknown function. L and B are differential operators defined on the problem domain Ω and its Neumann boundary Γ_t. Γ_u represents the Dirichlet boundaries with a prescribed value of $\overline{u}$, and f is the source term in the domain of the problem. The idea behind the least squares method is to find a solution that minimizes the residuals arising from the approximation. In CDLS, the domain of the problem is discretized using two sets of nodes named field nodes and collocation points. As shown

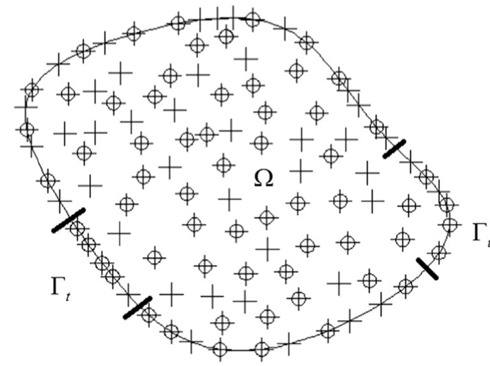

Fig. 1. The domain discretized by field nodes (O) and collocation points (+).

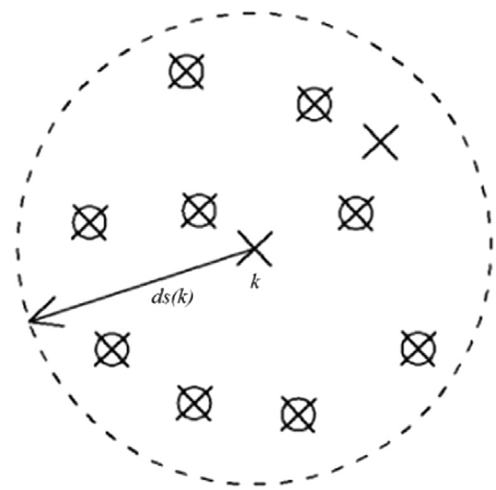

Fig. 2. Compact support of the kth collocation.

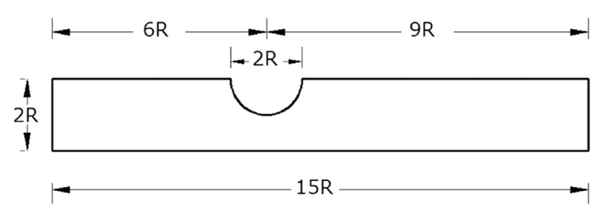

Fig. 3. Computational domain of the problem.

in Fig. 1, the problem domain and its boundaries are discretized by field nodes and collocation points. Assume n_p is the number of the field nodes in the domain and on the boundaries. Besides the field nodes, the collocation points are used in the problem domain and on its boundaries. In this methodology, one collocation point has to be placed in each field node, as shown in Fig. 1. The approximated value of the function u at a collocation point k with coordinate x_k, can be obtained through the following interpolation:

$$u(x_k)=\sum_{i=1\overline{n}} N_i(X_k)u_i, \tag{10}$$

where u_i is the value of the unknown function at the ith field node. $\overline{n}$ is the number of field nodes that the kth collocation point with coordinate X_k, has in its domain. This idea of compact support is shown in Fig. 2. To set up such a domain for each collocation point, a radius ds is defined so that a specific number of field nodes are placed into its support domain. In Eq. (10), $N_i(X_k)$ is the value of the shape function of the ith node at the kth collocation point that will be defined later. In this paper, first the number of nodes to the support collocation points is defined as $\overline{n}$. Then, for each