Development and verification of meshless diffuse approximate method for simulation of compressible flow between parallel plates

K B Rana¹, R Zahoor¹, B Mavrič² and B Šarler¹,²

¹ Faculty of Mechanical Engineering, University of Ljubljana, Aškerčeva cesta 6, 1000 Ljubljana, Slovenia.
² Institute of Metals and Technology, Lepi pot 11, 1000 Ljubljana, Slovenia.

Email: bozidar.sarler@fs.uni-lj.si

Abstract. A meshless numerical model is developed to simulate single-phase, Newtonian, compressible flow in the Cartesian coordinate system. The coupled set of partial differential equations, i.e., mass conservation, momentum conservation, energy conservation, and equation of state is solved by using Diffuse Approximate Method (DAM) and Pressure Implicit with Splitting of Operators (PISO) pressure correction algorithm on an irregular node arrangement. DAM is structured by using the second-order polynomial basis functions and the Gaussian weight function, leading to the weighted least squares approximation on overlapping subdomains. Implicit time discretization is performed for the predictor step of PISO, while in the corrector steps the equations are discretized explicitly. The numerical model is validated for flow between parallel plates with helium obeying ideal gas law. The solver’s accuracy is assessed by investigating the shape of the Gaussian weight and the number of the nodes in the local subdomains. The calculated velocity, temperature and pressure fields are compared with the Finite Volume Method (FVM) results obtained by OpenFOAM software and show a reasonably good agreement.

1. Introduction
To overcome the limitations of classical mesh-based methods in terms of accuracy, efficiency, artificial diffusion and ease of numerical implementation, meshless numerical methods are being developed for the past few decades. Characteristics and applications of different meshless methods can be found in monographs [1–7]. Strong form meshless Diffuse Approximate Method (DAM) is employed in this work. The method was initially developed in [8] and popularized by the group of Prof. Sadat for heat transfer and fluid flow problems [9,10]. The method gained popularity also in industrial problems, such as direct chill casting applications [11–14]. Recent applications of the method for two-phase incompressible flow are given in [15,16]. The present paper represents the first successful implementation of DAM for compressible flow problems.

2. Problem formulation

2.1. Governing equations
The Newtonian, compressible flow of an ideal gas is governed by the following Navier-Stokes equations
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0,
\]
\[
\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla P + \nabla \cdot \mathbf{\tau},
\]
\[
P = \rho RT,
\]
\[
\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{v}) = -\nabla \cdot \mathbf{q}
\]

where \( \mathbf{v}, \rho, P, R \) and \( t \) denote the velocity, density, pressure, specific gas constant and time, respectively. \( \mathbf{\tau} \) is the viscous stress tensor, defined as \( \mathbf{\tau} = \mu \left\{ \nabla \mathbf{v} + (\nabla \mathbf{v})^T \right\} + \lambda (\nabla \cdot \mathbf{v}) \mathbf{I} \) where \( \mu, \lambda = (-2/3)\mu \) and \( \mathbf{I} \) are the dynamic viscosity, bulk viscosity and identity tensor, respectively.

\( e = c_v T + 0.5 |\mathbf{v}|^2 \) is the total energy per unit volume, composed of internal and kinetic energy per unit volume with \( c_v \) and \( T \) standing for specific heat capacity at constant volume and temperature, respectively. \( \mathbf{q} = -k \nabla T \) is the conduction heat flux with thermal conductivity \( k \). The gravitational effects, heat generation and other types of body forces are assumed to be negligible.

Consider solving a 2D flow problem, governed by equations (1) - (4) in a channel, bounded between two parallel plates, as shown in Figure 1. Let \( y \) be the coordinate in the primary flow direction and \( x \) the transverse coordinate. The nominal dimensions are \( D = 0.3 \text{mm}, \ d = 0.15 \text{mm}, \) and \( L = 5 \text{mm} \).

**Figure 1.** Schematic representation of the problem

**Figure 2.** Concept of overlapping subdomains

At the inlet, a parabolic velocity profile \( v_y = 6.0 v_{avg} \left( x/D \right) \left( 1- x/D \right) \), constant temperature \( T = 300 \text{ K} \) and Neumann pressure boundary condition \( \frac{\partial P}{\partial n} = \left\{ \nabla \cdot \mathbf{\tau} - \frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) \right\} n \) are enforced. At the outlet, Neumann boundary conditions for velocity \( (\partial v_y/\partial n = 0) \) and temperature \( (\partial T/\partial n = 0) \) and Dirichlet pressure boundary condition \( (P = 1 \times 10^5 \text{ Pa}) \) are applied. No-slip velocity and Neumann pressure boundary conditions are used at the walls of the parallel plates. The walls are being heated by applying a constant heat flux boundary condition \( \frac{\partial T}{\partial n} = \frac{1}{k} q \cdot n = \frac{1}{k} q_s \), with \( q_s = 10 \text{ kW/m}^2 \). Helium
gas with \( \rho = P / (RT) \), \( R = 2077 \text{J/(kg K)} \), \( \mu = 1.96 \times 10^{-5} \text{Pa s} \), \( c_p = 5192 \text{J/(kg K)} \), \( c_v = 3096 \text{J/(kg K)} \), and \( k = 0.142 \text{W/(m·K)} \) flows between the parallel plates with an average inlet velocity of \( v_{avg} = 412 \text{m/s} \). Initially, the fluid is at rest with \( P = 1 \times 10^5 \text{Pa} \) and \( T = 300 \text{ K} \).

2.2. Numerical procedure

The computational nodes are placed in the interior and at the boundary of the domain. The local subdomain of its closest neighbouring nodes is assigned to each computational node, as shown in Figure 2. DAM uses the weighted least squares approach to determine a locally smooth approximation of the discrete data. The weight function \( \theta = \exp[-\| p - p_l \| (a/h^2)] \) is used to determine the relative impact of any node \( p \) to the central node \( p_l \) of the subdomain \( l \). \( h \) is the scaling factor, while the free parameter \( a \) determines the width of the weight function. The partial derivatives (up to 2\textsuperscript{nd} order) are applied to the approximation to obtain the vector of unknown coefficients in the present case. The number of the nodes in one local subdomain \( n_{loc} \) is 13. Pressure Implicit with Splitting of Operators (PISO) [17] algorithm is implemented for pressure and velocity coupling. PISO flow chart is shown in Figure 3.

![Figure 3. Flow chart of the PISO algorithm](image)

3. Results and discussion

The code is based on the already established meshless library written in modern object-oriented Fortran and compiled in Intel Fortran Composer Version 19. The numerical code is parallelized using OpenMP, and six threads are used for each computation. The steady-state is supposed to be achieved when the relative change in velocity, pressure, temperature and density fields between the two consecutive time steps is less than \( 6 \times 10^{-6} \). The results are compared with Finite Volume Method (FVM) solution, calculated with OpenFOAM software on a uniform grid of \( 500 \times 30 \) cells and convergence criterion set to \( 1 \times 10^{-6} \). The convergence of the meshless method is investigated for three irregular node arrangements with the number of nodes: 34425, 44217, 51425 with their required time steps \( 2.6 \times 10^{-2} \text{s}, 2.2 \times 10^{-2} \text{s} \) and \( 1.7 \times 10^{-2} \text{s} \) respectively. The node arrangements are generated automatically, keeping the density function constant all over the domain. The details of the node generation procedure are described in [12]. For all node arrangements, a comparison of velocity along the horizontal axis and pressure drop along the length are presented in Figure 4. The increasingly denser node arrangements provide 96\%, 98.47\% and 98.5\% match with the reference pressure results and 97.2\%, 98.86\% and 98.9\% match with the reference velocity results, respectively. It is evident that both denser
node arrangements provide a better match with the reference FVM result. The percentage error is calculated as \( \frac{(E_{\text{cal}} - E_{\text{ref}})}{E_{\text{ref}}} \times 100 \), where \( E_{\text{cal}} \) and \( E_{\text{ref}} \) represent the maximum values of DAM and FVM, respectively. The stability of the solver is further investigated with 44217 nodes for three values of the weight function’s free parameter: \( a_0 = 5.0, 10.0, 15.0 \). Each value of the free parameter is imposed to three different number of the nodes in a local subdomain: \( n_{\text{loc}} = 9, 13, 17 \).

Figure 4. Comparison of different node arrangements with OpenFOAM solutions a) velocity profile along the x-axis at the centre, b) pressure drop downstream.

In Figure 5, velocity profiles at the outlet, obtained from different combinations of \( a_0 \) and \( n_{\text{loc}} \), are compared with FVM solution. The solutions are reasonably accurate throughout the domain, with almost 97.5% match at the outlet. As the flow is fully developed, the parabolic shape of the velocity profile does not change in the direction of flow. Density is correlated with pressure and temperature through the equation of state. The pressure is decreased due to viscous forces acting on the fluid as it flows between the plates. The temperature of the fluid is increased due to it being heated through the walls. This pressure drop and temperature rise cause the density drop downstream. To balance this, the velocity rises downstream, conserving the mass flow. Pressure, temperature and density change along the symmetry line achieve 97-98% match for all the presented combinations of \( a_0 \) and \( n_{\text{loc}} \) and are presented in Figure 5.

Figure 5. DAM with different combinations of \( n_{\text{loc}} \) and \( a_0 \) are compared with OpenFOAM solutions in terms of a) velocity at the outlet, b) pressure at the centreline between the plates, c) density at the centreline between the plates, d) temperature at the centreline between the plates.

The results are not sensitive (similar up to three decimal places) to all presented combinations of \( a_0 \) and \( n_{\text{loc}} \) in terms of accuracy. However, it was observed that the free parameter \( a_0 = 10.0 \) achieves the steady state faster for each \( n_{\text{loc}} \). The solutions with \( n_{\text{loc}} = 13 \) and 17 and the highest \( a_0 \) did not converge.
Furthermore, the attempts to use $a_0$ less than 5.0 are unsuccessful, and the solution diverges every time. In Table 1 a comparison of the required number of iterations and time per iteration for attempted combinations of $n_{loc}$ and $a_0$ are presented. The combination of $a_0 = 10.0$ and $n_{loc} = 9$ is computationally least expensive, which converges the solution in 10680 iterations and completes one iteration in 1.92 s. Oppositely, the combination of $a_0 = 15.0$ and $n_{loc} = 9$ reaches the steady-state in 13104 iterations consuming 3.02 s per iteration.

| $n_{loc}$ | $a_0 = 5.0$ | $a_0 = 10.0$ | $a_0 = 15.0$ | $a_0 = 5.0$ | $a_0 = 10.0$ | $a_0 = 15.0$ |
|-----------|-------------|-------------|-------------|-------------|-------------|-------------|
| 9         | 12483       | 10680       | 13104       | 2.01        | 1.92        | 3.02        |
| 13        | 11840       | 11260       | --          | 2.87        | 2.10        | --          |
| 17        | 10560       | 12152       | --          | 2.47        | 2.97        | --          |

4. Conclusions
The first numerical model to simulate the Newtonian, compressible flow between parallel plates by using meshless DAM has been developed. The presented results agree very well with the OpenFOAM FVM simulations. DAM provides easy and efficient implementation on irregular node arrangements. Out of all the tested combinations $a_0 = 10.0$ and $n_{loc} = 9$ is recommended for future applications. The study provides a strong foundation for further implementing the numerical method for phase-field formulated, two-phase, compressible flow problems, including the gas-focused micro-jets. The upgrade of the work includes implementing DAM for more complex test cases with shocks, axisymmetric coordinate system and in combination with the phase-field formulation to two-phase flow problems.

Acknowledgement
The first author would like to thank the Slovenian Grant Agency ARRS for support in the framework of the Young Researcher project. The work was also funded by the ARRS projects J2-1718 and core funding P2-0162.

References
[1] Atluri S N 2004 The Meshless Methods for Domain and BIE Discretization. (USA).
[2] Atluri S N and Shen S 2002 The Meshless Local Petrov-Galerkin (MLPG) Method. (USA).
[3] Chen W, Fu Z J and Chen C S 2013 Recent Advances in Radial Basis Function Collocation Methods. (USA).
[4] Šarler B and Atluri S N (editors) 2010 Recent Studies in Meshless and Other Novel Computational Methods. (USA).
[5] Liu G R and Gu Y T 2005 An Introduction to Meshfree Methods and their Programming (The Netherlands).
[6] Liu G R 2003 Meshfree Methods: Moving Beyond the Finite Element Method (USA).
[7] Li H and Mulay S S 2013 Meshless Methods and Their Numerical Properties. (USA).
[8] Nayroles B, Touzat G and Villon P 1992 Comput. Mech 10, 307-318.
[9] Sadat H and Prax C 1996 Int. J. Heat Mass Transf. 39 214–18.
[10] Prax C, Sadat H and Salagnac P 1996 Transp. Porous Media 22, 215–23.
[11] Šarler B, Vertnik R and Perko J 2005 Comput. Mater. Contin. 2 77–83.
[12] Hatič V, Mavrič B, Košnik N and Šarler B 2018 Appl. Math. Model 54 170–88.
[13] Hatič V, Mavrič B and Sarler B 2018 Int. J. Numer. Methods Heat Fluid Flow 28 361–380.
[14] Hatič V, Mavrič B and Sarler B 2020 Eng. Anal. Bound. Elem. 113 191–203.
[15] Talat N, Mavrič B, Hatič V, Bajt S and Šarler B 2018 Eng. Analysis Bound. Elem. 87 78–89.
[16] Talat N, Mavrič B, Belšak G, Hatič V, Bajt S and Šarler B 2018 Int. J. Multiph. Flow 108 169–180.
[17] Issa R I 1985 J. Comput. Phys. 62 40–65.