Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 55(3) (2023) 63-66
DOI: 10.22060/mej.2023.21728.7497

Meshless Method for Numerical Solution of Internal Flows with Axial Symmetry

M. Hadidoolabi, M. Hashemabadi*, R. Jamshidi

Faculty of Aerospace, Malek Ashtar University of Technology, Tehran, Iran

ABSTRACT: In this research, a meshless numerical method has been developed to solve internal and axisymmetric flows. In this method, the least squares of the Taylor series are used for spatial discretization and explicit multi-step Runge-Kutta method is used for temporal discretization. Governing equations are based on two-dimensional and symmetric Euler equations. The second and forth order artificial dissipation are used to solve the flows. In order to model boundary condition, subsonic and supersonic inlet and outlet boundary conditions as well as the wall boundary have been used according to the problem. To validate the results of the code, the inviscid flow inside a two-dimensional nozzle and the supersonic flow inside the channel along with bump have been simulated and the results have been compared with valid data. The simulation of the steady flow inside a axi-symmetric convergent-divergent supersonic nozzle with Mach 5 in outlet has been done to measure the accuracy of solving the numerical code at the hypersonic speed. The results show that the developed code can simulate steady internal and axi-symmetric flows with very good accuracy. The process of code convergence is also presented, which shows the appropriate convergence of the developed code. The analysis time for shock capturing in the axi-symmetric nozzle is about 64% faster than the Fluent software.

Review History:

Received: Aug. 26, 2022
Revised: Mar. 06, 2023
Accepted: Mar. 07, 2023
Available Online: Mar. 24, 2023

Keywords:

Meshless method
internal flow
axi-symmetric flow
numerical method
inviscid flow

1- Introduction

The numerical meshless methods recently are used to overcome some deficiency of mesh based numerical methods. In this methods only nodes are defined without any relation between nodes and nodes cloud are used for estimation of derivatives at each point. Easiness of node generation and improvement of node distribution are some advantages of meshless methods. At the present years some researchers are done on node generations for meshless solvers [1]. Liu and Gu [2] introduced flow solution technics in meshless methods. Batina [3] used constant weight function and Deshpande [4] used upwind methods to estimate functions based on least squares methods. Katz and Jameson [5], proposed multicloud method to increase convergence rate of meshless operators. Hashemabadi and Hadidoolabi used high order discretization to increase meshless methods accuracy [6-8]. Shahane [9] developed a high order meshless method for incompressible flow solution. Couturier [10] used a meshless method base on approximate diffusion for solving 2D and 3D flows. In the present research an efficient meshless method is used for solving the internal 2D and axisymmetric nozzle flow.

2- Numerical Methodology

Inviscid Euler equations in 2D and axisymmetric form are used for internal flow solution. The nodes cloud as shown in Figure 1 are used with first order Taylor series for spacial discretization.

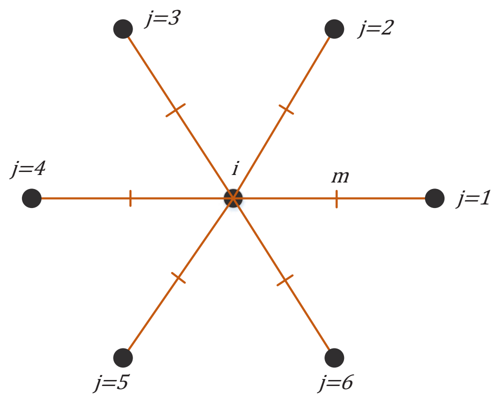

Fig. 1. nodes cloud for point i

2^{nd} and 4^{th} order artificial dissipation are used for eliminate flow oscillation. An explicit multi-step Runge-Kutta method is used for time discretization.

Two standard models are used for meshless based flow solver validation. First model is the Mason [11] two-

*Corresponding author's email: hashemabadi@mut.ac.ir

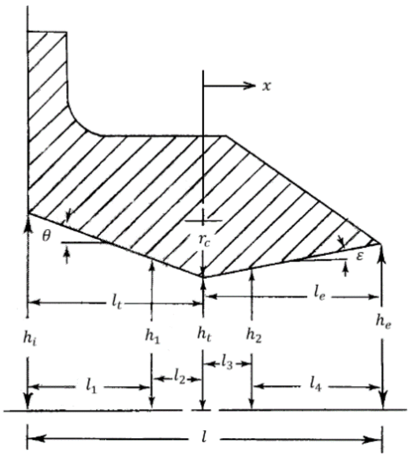

Fig. 2. Mason nozzle B1 geometry

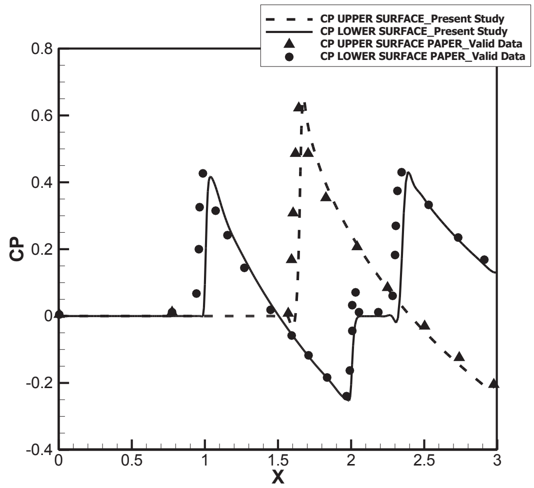

Fig. 5.Pressure coefficient on the two-dimensional channel walls with 4% bump (M=1.4)

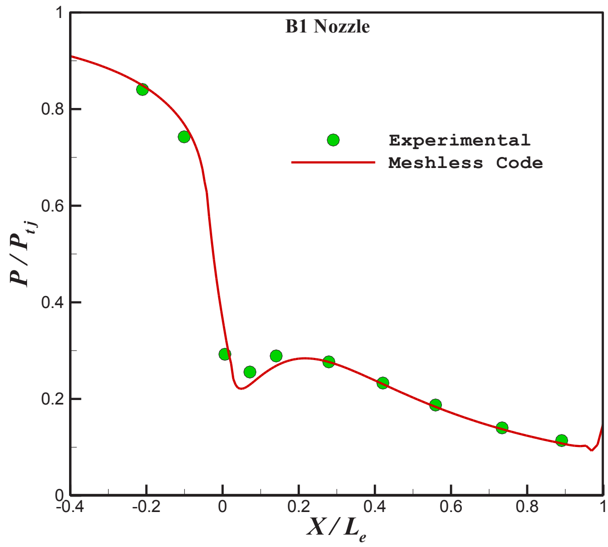

Fig. 3. Nondimension pressure distribution over nozzle B1 wall

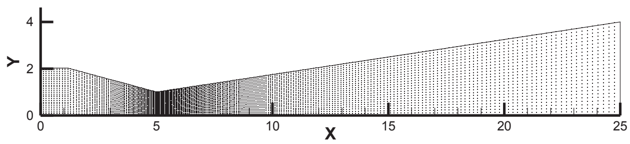

Fig. 6. Schematic of geometry and points distribution for 2-D nozzle

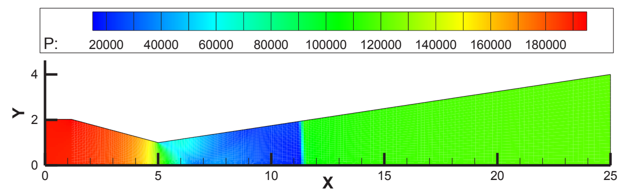

Fig. 7. Pressure contour in the 2-D nozzle

dimensional converging-diverging nozzles B1. The model geometry is shown in Figure 2.

Figure 3 shows the comparison of present meshless solver results with Mason experimental data. The good agreement is shown in this figure.

Second validation model is a 2D bumped channel. A supersonic 1.4 Mach number flow is solved in channel with a 4% bump on the lower surface. The Mach contour for this flow simulation is shown in Figure 4. A good estimation of shock waves and their reflection are obtained in the flow field. In Figure 5 the pressure distribution on the upper and lower surfaces of channel are compared with results of reference [12]. Good agreement between present results and that reference is seen.

3- Results and Discussion

Two problem are simulated with present meshless based solver. First problem is the shock capturing in a 2D converging-diverging nozzle. The nozzle area ratio is $A_{exit} / A_{throat} = 4$. Inlet total pressure is 2bar and static pressure at outlet set as 1.21 bar. The node distribution is shown in Figure 6 and the pressure contours in figure7. Based on Figure 7 a normal shock is captured at position 11.4m that is with good agreement with analytical result with position 1.66.

The second problem is flow in an axisymmetric nozzle. Nozzle inlet and throat areas are 0.1963 and 0.00875 respectively. Inlet total pressure is 21.56 bar and outlet static pressure set as 0.98 bar. The Mach contour in the nozzle is shown in Figure 8. As seen the output Mach number is 5 that is compatible with theory result.

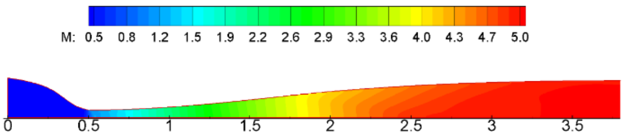

Fig. 8. Mach contour in axisymmetric nozzle

4- Conclusions

A numerical solver is developed based on meshless method. The solver is used for simulation of 2D and axisymmetric internal flows. The solver results are with good agreement with validated data.

References

[1] J.S. Rhee, Development of Point Generation Technique for a Meshless Method, Master Thesis, Seoul National University, 2014.

[2] G.R. Liu, Y.T. Gu, An introduction to meshfree methods and their programming, Springer Science & Business Media, 2005.

[3] J. BATINA, A gridless Euler/Navier-Stokes solution algorithm for complex-aircraft applications, in: 31st Aerospace Sciences Meeting, 1993, pp. 333.

[4] S.M. Deshpande, V. Ramesh, K. Malagi, K. Arora, Least squares kinetic upwind mesh-free method, Defence Science Journal, 60(6) (2010).

[5] A. Katz, A. Jameson, Multicloud: Multigrid convergence with a meshless operator, Journal of Computational Physics, 228(14) (2009) 5237-5250.

[6] M. Hashemabadi, M. Hadidoolabi, An accurate and efficient gridless method based on implicit, fast, and constrained weights optimization schemes for compressible flows, Journal of the Brazilian Society of Mechanical Sciences and Engineering, 40(2) (2018) 1-17.

[7] M. Hashemabadi, M. Hadidoolabi, Efficient Gridless Method Using Constrained Weights Optimization for Two-Dimensional Unsteady Inviscid Flows at Low Angles of Attack, Journal of Aerospace Engineering, 30(5) (2017) 04017052.

[8] M. Hashemabadi, M. Hadidoolabi, Implicit second-order CUSP gridless method for unsteady moving boundary simulations, Computers & Mathematics with Applications, 74(4) (2017) 842-858.

[9] S. Shahane, A. Radhakrishnan, S.P. Vanka, A high-order accurate meshless method for solution of incompressible fluid flow problems, Journal of Computational Physics, 445 (2021) 110623.

[10] S. Couturier, H. Sadat, A meshless method for the solution of incompressible flow equations, arXiv preprint arXiv:2103.02247, (2021).

[11] M.L. Mason, L.E. Putnam, R.J. Re, The effect of throat contouring on two-dimensional converging-diverging nozzles at static conditions, (1980).

[12] Z. Ma, N. Emad, H. Chen, A Local Meshless Method for Solving Compressible Euler Equations, Space Research Journal, 1(1) (2008) 1-16.

HOW TO CITE THIS ARTICLE

M. Hadidoolabi, M. Hashemabadi, R. Jamshidi, Meshless Method for Numerical Solution of Internal Flows with Axial Symmetry, Amirkabir J. Mech Eng., 55(3) (2023) 63-66.

DOI: 10.22060/mej.2023.21728.7497