

MESHLESS METHOD FOR SIMULATION OF COMPRESSIBLE FLOW

By

Ebrahim Nabizadeh Shahrebabak

Bachelor of Science- Chemical Engineering
Isfahan University of Technology, Iran
2014

A thesis submitted in partial fulfillment
of the requirements for the

Master of Science - Mechanical Engineering

Department of Mechanical Engineering
Howard R. Hughes College of Engineering
The Graduate College

University of Nevada, Las Vegas
August 2017

ProQuest Number: 10616580

All rights reserved

INFORMATION TO ALL USERS

The quality of this reproduction is dependent upon the quality of the copy submitted.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if material had to be removed, a note will indicate the deletion.



ProQuest 10616580

Published by ProQuest LLC (2017). Copyright of the Dissertation is held by the Author.

All rights reserved.

This work is protected against unauthorized copying under Title 17, United States Code
Microform Edition © ProQuest LLC.

ProQuest LLC.
789 East Eisenhower Parkway
P.O. Box 1346
Ann Arbor, MI 48106 – 1346

Copyright 2017 by Ebrahim Nabizadeh Shahrebabak

All Rights Reserved

Thesis Approval

The Graduate College
The University of Nevada, Las Vegas

June 28, 2017

This thesis prepared by

Ebrahim Nabizadeh Shahrebabak

entitled

Meshless Method for Simulation of Compressible Flow

is approved in partial fulfillment of the requirements for the degree of

Master of Science - Mechanical Engineering
Department of Mechanical Engineering

Darrell W. Pepper, Ph.D.
Examination Committee Chair

Kathryn Hausbeck Korgan, Ph.D.
Graduate College Interim Dean

Dr. Hui Zhao, Ph.D.
Examination Committee Member

William Culbreth, Ph.D.
Examination Committee Member

Laxmi Gewali, Ph.D.
Graduate College Faculty Representative

ABSTRACT

In the present age, rapid development in computing technology and high speed supercomputers has made numerical analysis and computational simulation more practical than ever before for large and complex cases. Numerical simulations have also become an essential means for analyzing the engineering problems and the cases that experimental analysis is not practical. There are so many sophisticated and accurate numerical schemes, which do these simulations. The finite difference method (FDM) has been used to solve differential equation systems for decades. Additional numerical methods based on finite volume and finite element techniques are widely used in solving problems with complex geometry. All of these methods are mesh-based techniques. Mesh generation is an essential preprocessing part to discretize the computation domain for these conventional methods. However, when dealing with mesh-based complex geometries these conventional mesh-based techniques can become troublesome, difficult to implement, and prone to inaccuracies. In this study, a more robust, yet simple numerical approach is used to simulate problems in an easier manner for even complex problem.

The meshless, or meshfree, method is one such development that is becoming the focus of much research in the recent years. The biggest advantage of meshfree methods is to circumvent mesh generation. Many algorithms have now been developed to help make this method more popular and understandable for everyone. These algorithms have been employed over a wide range of problems in computational analysis with various levels of success. Since there is no connectivity between the nodes in this method, the challenge was considerable. The most fundamental issue is lack of conservation, which can be a source of unpredictable errors in the solution process. This problem is particularly evident in the presence of steep gradient

regions and discontinuities, such as shocks that frequently occur in high speed compressible flow problems.

To solve this discontinuity problem, this research study deals with the implementation of a conservative meshless method and its applications in computational fluid dynamics (CFD). One of the most common types of collocating meshless method the RBF-DQ, is used to approximate the spatial derivatives. The issue with meshless methods when dealing with highly convective cases is that they cannot distinguish the influence of fluid flow from upstream or downstream and some methodology is needed to make the scheme stable. Therefore, an upwinding scheme similar to one used in the finite volume method is added to capture steep gradient or shocks. This scheme creates a flexible algorithm within which a wide range of numerical flux schemes, such as those commonly used in the finite volume method, can be employed. In addition, a blended RBF is used to decrease the dissipation ensuing from the use of a low shape parameter. All of these steps are formulated for the Euler equation and a series of test problems used to confirm convergence of the algorithm.

The present scheme was first employed on several incompressible benchmarks to validate the framework. The application of this algorithm is illustrated by solving a set of incompressible Navier-Stokes problems.

Results from the compressible problem are compared with the exact solution for the flow over a ramp and compared with solutions of finite volume discretization and the discontinuous Galerkin method, both requiring a mesh. The applicability of the algorithm and its robustness are shown to be applied to complex problems.

ACKNOWLEDGMENTS

I would like to express my profound gratitude to my academic advisor, Prof. Darrell Pepper, for his scholastic advice and technical guidance throughout this investigation. His devotion, patience, and focus on excellence allowed me to reach this important milestone in my life. I also wish to extend my acknowledgements to the examination committee members, Dr. William Culbreth, Dr. Hui Zhao, and Dr. Laxmi Gewali for their guidance and suggestions.

DEDICATION

This study is dedicated to my wife, Farideh, and my parents, Khadijeh and Azizollah. Without their continued and unconditional love and support, I would not be the person I am today.

TABLE OF CONTENTS

ABSTRACT	iii
ACKNOWLEDGMENTS.....	v
LIST OF TABLES	x
LIST OF FIGURES.....	xi
1. CHAPTER 1- INTRODUCTION	1
1.1. Previous research studies	1
1.2. Motivation	2
1.3. Objective	4
1.4. Thesis Outline	6
2. CHAPTER 2- RADIAL BASIS FUNCTIONS.....	7
2.1. Introduction	7
2.2. Shape parameter	10
2.3. Global RBF	11
2.4. Localized RBF.....	12
2.5. RBF-DQ	14
2.6. Summery	16
3. CHAPTER 3- APPLICATION OF RBF-DQ TO SOLVE INCOMPRESSIBLE FLOW	18
3.1. Introduction	18
3.2. Flow Solution.....	18
3.2.1. Governing Equations	18
3.2.2. Projection Method.....	19

3.3.	Benchmarks Examination	21
3.3.1.	Moving Wall Cavity	22
3.3.1.1.	Problem setup	22
3.3.1.2.	Localized RBF-DQ.....	22
3.3.2.	Natural Convection inside a close square	23
3.3.2.1.	Problem setup	23
3.3.2.2.	Localized RBF-DQ.....	23
3.3.3.	Flow with Forced Convection over a Backward-Facing Step	24
3.3.3.1.	Problem setup	24
3.3.3.2.	Localized RBF-DQ.....	25
3.4.	Summery	25
4.	CHAPTER 4- RBF-DQ FOR COMPRESSIBLE FLOW	28
4.1.	Introduction	28
4.2.	The Euler Equations.....	28
4.2.1.	Conservation-Law Form	30
4.3.	RBF-DQ Implementation.....	31
4.4.	Upwinding Scheme	33
4.4.1.	Roe's approximate Riemann solver	34
4.5.	Summery	39
5.	CHAPTER 5- COMPERISSIBLE FLOW BENCHMARK.....	41
5.1.	Benchmark Problem.....	41
5.1.1.	Problem domain	41
5.1.2.	Boundary conditions	43

5.2. Effect of parameters	47
5.2.1. Shape parameter	47
5.2.2. Size of subdomain.....	47
5.2.3. Slope limiter.....	47
5.2.4. Time step.....	48
5.3. Postprocessing.....	49
6. CHAPTER 6- CONCLUSION AND FUTURE WORK	53
6.1. Summary of work.....	53
6.2. Future work	54
APPENDIX: NOMENCLATURE.....	55
REFERENCES	56
CURRICULUM VITAE.....	60

LIST OF TABLES

Table 3.1 Constant parameter of each case.....	19
Table 5.1 Inlet condition.....	46

LIST OF FIGURES

Figure 2.1 Size of subdomain a) 9 node stencil, b) 15 node stencil, c) 20 node stencil	14
Figure 3.1 Cavity problem: Velocity vector plots of two scheme for $Re=100$. a) Present Scheme, b) Reference scheme (Waters & Pepper, 2015).....	23
Figure 3.2 Natural convection: Velocity Vector plots of two scheme for $Pr=0.71$, and $Ra=103$. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)	26
Figure 3.3 Natural convection: Temperature contours of two schemes for distribution with $Pr=0.71$, $Ra=103$. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)	26
Figure 3.4 Backward facing: Streamline for fluid flow. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)	27
Figure 3.5 Backward facing: isothermal for fluid flow. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)	27
Figure 4.1 Configuration in subdomain	32
Figure 4.2 Position of conservative variables	37
Figure 4.3 Present algorithm flowchart	40
Figure 5.1 Benchmark domain.....	42
Figure 5.2 Node distribution	42
Figure 5.3 Configuration in subdomain	42
Figure 5.4 Wall boundary condition	44
Figure 5.5 Mach number contour.....	50
Figure 5.6 Pressure contour	50
Figure 5.7 Pressure contour using Finite Volume Method	51
Figure 5.8 Pressure contour using DG Method	51

Figure 5.9 Exact solution (Wang & Widhopf, 1989).....	52
Figure 5.10 Pressure along the bottom wall.....	52

CHAPTER 1- INTRODUCTION

Natural phenomena, whether electrical, biological, mechanical, chemical, environmental, geological or electronic, can be described by means of mathematical models. Since most of these problems are complex, it is hard to find exact solutions for these models. The way to find solutions for these problems is to solve them numerically or statistically. Nowadays, researchers have to be familiar with numerical or statistical techniques for a wide variety of problems. By the advent of supercomputer technology, computational simulation techniques have increasingly become an essential way for simulating complex and practical problems in engineering and science where experimental analysis is highly expensive.

The main purpose of numerical simulation is to discretize the continuum physical domain to a discretized domain which is solvable on computers. The discretization process is applied to both equations and the domain of the problem. Researchers can find an approximate solution for a complex problem efficiently, as long as a proper and reliable numerical method is implemented.

1.1. Previous research studies

Many studies have focused on numerical or approximation methods to develop an efficient technique. Many numerical methods have been proposed and developed, utilizing the finite difference method (FDM), the finite volume method (FVM), the finite element method (FEM), the boundary element method (BEM), and more recently the meshless method to be discussed here.

Studies on meshless methods can be traced back to 1977, but only a few studies had been done in this area until the past two decades. Lucy (1977), using one of the oldest forms of the meshless method, smooth particle hydrodynamics (SPH), modeled astrophysical phenomenon. More recently, a wide range of meshless methods have been developed and studied. Such improved methods include the smooth particle hydrodynamics (SPH) (Gingold & Monaghan, 1977; Lucy, 1977; Monaghan, 1988; Randles & Libersky, 1996), the diffuse element method (DEM) (Nayroles, Touzot, & Villon, 1992), the element free Galerkin (EFG) method (Belytschko, Lu, & Gu, 1994; Lu, Belytschko, & Gu, 1994; Noguchi, Kawashima, & Miyamura, 2000), the reproducing kernel particle method (RKPM), the moving least-squares reproducing kernel (MLSRK) method (Liu, Jun, Li, Adee, & Belytschko, 1995), the hp-clouds method (Duarte & Oden, 1996; Liszka, Duarte, & Tworzydło, 1996), the finite point method (Oñate, Idelsohn, Zienkiewicz, Taylor, & Sacco, 1996), the meshless local Petrov-Galerkin (MLPG) method (Atluri & Zhu, 1998), boundary node method (BNM) (Mukherjee & Mukherjee, 1997), the meshless local boundary integral equation (MLBIE) method (Atluri & Zhu, 2000), and the gridless Euler/Navier–Stokes solution (Batina, 1993; Morinishi, 1995). Another group of meshless methods, are based on radial basis functions (RBFs). More recently, RBFs have become attractive for solving partial differential equations. The RBF methods for multivariate approximation have wide applications in modern approximation theory when the task is to approximate scattered data in several dimensions.

1.2. Motivation

For decades, the finite element method (FEM) and the finite volume method (FVM) have been the standards tool for numerically solving a wide variety of engineering problems

especially fluid flow and thermal problem simulation. However, as problems become more complex, these methods become inadequate and inefficient. A good mesh is very important in CFD. This can be an expensive issue in the sense of storage needed and CPU time, especially when dealing with complex geometries and/or complex physics such as crack propagation, shock propagation, astrophysics phenomena, metal cutting and extrusion. Conventional methods always have some difficulties when they are used to solve these kinds of problems. Therefore, to get accurate results, a highly dense mesh near the discontinuity is usually required; otherwise, the computational results are not reliable.

Using conventional methods can cause some degradation of accuracy in complex physics, since adaptive meshing in conventional methods is difficult to implement. On the other hand, it is impractical to solve system containing billions of unknowns. The time and cost of mesh generation and mesh refinement in conventional methods is high.

To reduce the cost of the meshing process, different methods have been proposed over the past three decades and a significant progress has been achieved in this area. Like all the previous techniques, the governing equations of essential parameters like mass, momentum, and energy must be conserved by each these new techniques. One of the promising numerical techniques to satisfy all these limitations is the meshless, or meshfree method.

Engineering research has began focusing the use of meshless method, over the past decade. Since meshless methods eliminate the mesh generation required for discretization of problem domains, the approximation process only needs to collocate a functional value on distributed set of nodes. The connection between nodes is not required, which helps reduce

storage. Time can also be saved by using a fully automated procedure to generate nodes (G. R. Liu, 2010).

The use of meshless methods can lead to computational advantages with less programming efforts; especially a hybrid meshless method in combination with a conventional method can improve the result for complicated “multiphysics” problems. Some of the advantages of meshless are :

- 1) Computational cost is reduced and storage saved significantly since a mesh and book keeping are not required.
- 2) For cases where more refinement is needed, one can easily increase the accuracy by using r-adaptation or adding nodes to the computational domain. Providing high-order shape functions are constructed.
- 3) The Meshless routine can be used many time during the solving process.

1.3. Objective

Compared to conventional FDM, FVM, and FEM, the meshless method can be used to track strong discontinuities or large deformations of strongly nonlinear problems. To increase the resolution near geometric complexities, one can add nodes and refine the simulation. This makes the programing and simulation more convenient to solve complex system of equations over arbitrary domains.

There are different ways to distribute the nodes throughout the domain. One can use any type of uniform, nonuniform or hybrid distribution to collocate the data. The dependency of conventional methods on the mesh causes some problems in refinement processes near a

discontinuity. Thus, they are not very suitable and applicable for tracking discontinuities such as shocks or strong deformation, especially when they are not aligned with the original mesh edges (Belytschko, Krongauz, Organ, Fleming, & Krysl, 1996). The common procedure for solving a moving or evolving discontinuity in conventional techniques is to remesh the simulation field in each iteration. The remeshing process can be a source of numerous difficulties such as reduction of accuracy and cumbersome programming. Moreover, successive remeshing processes can also be a significant waste in terms of the computational time and cost. On the other hand, the meshless methods does not significantly require mesh dependency processes. The goal of meshless methods is to remove the mesh related problems by performing approximations over all the nodes. Therefore, moving deformation or discontinuity propagation can be tracked without remeshing, with little compromise in accuracy. Since the refinement process is easier in meshless, this degradation in accuracy can be compensated by performing refinement around the discontinuities.

In the present study a blended localized Radial Basis Function Differential Quadrature (RBF-DQ) for solving the Euler equation is introduced. The algorithm is blended to three different regions and include a steep gradient limitation for the shape parameter, which changes to high and low values. The derivatives are approximated using RBF-DQ, and will be explained later. The other essential part of the study is dealing with the discontinuities and capturing shock propagation. An upwinding scheme is applied to compute the fluxes at the mid-point between the reference point and the support point. Roe's solver which is an approximate Riemann solver is used in this study. The conservative values at the each side of the mid-point approximated by using blended RBF-DQ and then the fluxes are computed.

1.4. Thesis Outline

The scheme is examined for several incompressible and compressible cases with discontinuities. In the chapter 2, an introduction to RBF and more details about the aspects of RBF are provided. In chapter 3, we test the RBF-DQ code on several incompressible flow problems and compare with flow benchmarks obtained from other studies. Some details about the setup process are also provided. Chapter 4 introduce the Euler equation and its hyperbolic characteristics. The details of Roe's scheme, which is used here for the purpose of upwinding, is described. The blended RBF-DQ is introduced and the idea of blending explained. In chapter 5, we obtain results for supersonic flow with an oblique shock throughout the domain. The results are in a good agreement with the exact solution. The method is also compared qualitatively and quantitatively with several other numerical schemes and in some cases, the results are more accurate. In addition, the dependency of different parameters on the accuracy of the solution such as uniform or nonuniform distribution of nodes and value of shape parameter and timestep are also discussed. Chapter 6 provides the conclusions drawn from this study and suggestions for future research in the area of RBF approximation.

CHAPTER 2- RADIAL BASIS FUNCTIONS

2.1. Introduction

Radial basis function methods are the means to approximate the multivariate functions we wish to study in this section. This type of truly meshfree interpolation begins with the idea that any arbitrary domain, especially irregular domains can be approximated by collocating about a number of nodes distributed in the domain with some set of basis functions. There are two types of RBF approximation, global meshless method and local meshless method. The former method solves the domain by using a large sparse matrix that is calculated from all the nodes inside the domain. This type of approximation has some well-known drawbacks, which will be discussed later. For the second method, one needs to divide the overall domain into a number of smaller subdomains, which leads to more effective and precise results when compared to global approximation. The size of the subdomain includes a predetermined number of the nearest nodes surrounding the reference point. The accuracy of the solution relies on many parameters such as the number of supporting nodes and their distribution and the value of the shape parameter. Formulation and details of these methods follow.

RBFs were initially developed for multivariate scatter data and function interpolation. The meshfree feature of RBFs on higher dimensional problems motivated researchers to employ them in solving PDEs. After some research, they found that this type of meshfree approximation has high-order accuracy than conventional finite difference schemes on a scattered distribution of nodes (Tota, 2006). Due to the simplicity of programming and small amount storage required, researchers began to use them in all area of numerical modeling.

There are different types of RBFs, that have been developed and employed in approximating different numerical algorithms. The most frequently used RBFs are

Multiquadrics (MQs): $\varphi(r) = \sqrt{r^2 + c^2}$

Gaussians: $\varphi(r) = e^{-ar^2}$, $a > 0$

Thin-Plate Splines (TPS): $\varphi(r) = r^2 \log(r)$

Inverse MQs: $\varphi(r) = \frac{1}{\sqrt{r^2 + c^2}}$, $c > 0$

The most popular type of RBF is the Multiquadrics basis function. It was first proposed by Hardy (1971). Results of a study by Franke (1982) showed that Multiquadrics generally do better with scattered data compared to the other basis functions. In fact, the exponential convergence of MQ makes it preferable to other basis functions. MQ RBFs are used in the present work. Kansa's method (Kansa, 1990) was developed by directly collocating radial basis functions, especially MQ approximations. He initially employed RBF methods to solve problems in fluid flow and CFD. He discretized the domain and equations by using RBFs over a random distribution of nodes. The approach was similar to finite difference methods applied on random distribution of nodes. Other researchers began to show interest in RBFs for a variety of applications such as electromagnetic, fluid mechanics, heat transfer and solid mechanics problems. Other research in the area of RBFs comes from Larsson and Fornberg (2003), and Zhou et al. (2003). Some of the well-known drawbacks of RBFs are poor conditioning of large matrices resulting from the discretization of governing equations, and selection of an appropriate shape parameter especially, when dealing with steep gradient regions and discontinuities. To

overcome the first issue, some preconditioning and domain decomposition technique is required (Ling & Kansa, 2005; Mai-Duy & Tran-Cong, 2002). This is the reason for better performance of localized meshless method over global meshless.

The solution for these problems will be discussed later. Wu and Shu (2002) developed a new branch of differential quadrature, which is fully mesh-free. This type of DQ uses radial basis functions as the approximation functions and the nodes inside each sub domain are used to approximate partial derivatives at a reference node. Shu et al. (2003) proposed a local RBF-based differential quadrature (RBF-DQ) method. They applied RBF-DQ for simulation of incompressible flow problems. There are also some pioneer studies on using RBFs for capturing shocks and discontinuities. Shu et al. (2005) used RBF-DQ with an upwinding scheme to capture the shock waves and compressible flow simulation. The method is fully mesh-free but due to the small value of shape parameter, shocks are smeared. Harris et al. (2017) implemented a blended RBFs scheme where the shape parameter switches to high and low value according to the gradient region. Low value was used for steep gradient regions and higher values in smooth regions.

The framework of this current meshfree approach is formed of three parts: the first part is the derivative approximation by the means of localized RBF-DQ method; then blend the approximation depending on the position of the node by changing the shape parameter; the last step is the meshfree upwinding scheme for flux calculation.

In the following sections, we will introduce the global and localized RBF method. In addition, details about localized DQ-RBF will be provided later.

2.2. Shape parameter

The shape parameter is a key factor in RBFs approximation when using MQ and inverse MQ. Choosing an irrelevant number can create an inaccurate solution. It has a positive real value less than one. There is no theory and proven analysis of how to select the shape parameter to obtain the most accurate result. Wang and Liu (2002) investigated the effect of the shape parameter for MQ and Gaussian basis function. They found that for MQ basis, the condition number of the matrix is stable when the shape parameter is less than 1. They also showed that a high shape parameter increases the condition number. Another study, has carried out by Frank and Schaback (1998), examined the RBF method to solve partial derivatives equations and derived a formula for choosing the shape parameter as

$$c = \frac{1.25 \times R}{\sqrt{N_I}}$$

where N_I is the size of the subdomain and R is the radius of the smallest subdomain. Hardy (1971) introduced another formula for estimating the shape parameter:

$$c = 0.815 \times d$$

where d is calculated by $d = \frac{1}{N_I} \sum_{i=1} d_i$ with d_i being the distance between the reference node and the other nodes in the subdomain. Afiatdoust and Esmailbeigi (2015) also examined an algorithm to find the optimal value of the shape parameter, and showed some improvement in the accuracy of the simulations. Their proposed algorithm make a balance between accuracy and ill-condition and obtain desirable accuracy only for solving ODEs. The algorithm is time-consuming and its cost of calculation is high.

2.3. Global RBF

Consider the general differential equation

$$Lu = f(x) \quad \text{in } \Omega, \quad Bu = g(x) \quad \text{on } \Gamma \quad 2.1$$

where L can be any arbitrary differential operator, B is an operator imposed on the boundaries and can be any kind of boundary condition, such as Dirichlet, Neumann, or Robin condition. Let

$\{P_i = (x_i)\}_{i=1}^N$ be N collocation points in the analyzed domain, where $\{(x_i)\}_{i=1}^{N_I}$ are interior nodes and $\{(x_i)\}_{i=N_I+1}^N$ are boundary points. Kansa (1990) suggested the following approximation for Eq.

(2.1)

$$u(x) = \sum_{j=1}^N u_j \phi_j(x) \quad 2.2$$

where $u_{j=1}^N$ are the summation of unknowns and boundary values and $\phi_j(x) = \phi(\|P - P_j\|)$ is the radial basis function. $u_{j=1}^{N_I}$ are the unknown coefficients to be determined, $u_{j=N_I+1}^N$ are the boundary values which needed to be fixed after each iteration. For MQ RBFs, $r = \|P - P_j\|$ is the Euclidean distance between nodes $P=(x)$ and $P_j=(x_j)$. For each point (x_j, y_j) , ϕ_j is calculated by the following expression

$$\phi_j(x) = \sqrt{(x - x_j)^2 + (y - y_j)^2 + c^2} \quad 2.3$$

Substituting Eq. (2.1) into Eq. (2.2), leads to an $N \times N$ linear system of equations,

$$\sum_{j=1}^N (L\phi_j)(x_j)u_j = f(x_j) , j=0,1,2,\dots,N_I \quad 2.4$$

$$\sum_{j=1}^N (B\phi_j)(x_j)u_j = g(x_j) , j= N_{N+1}, N_{I+2}, \dots, N \quad 2.5$$

in which we just solve for the interior nodes. Therefore, a matrix of $N_I \times N_I$ Eq. (2.1), for the unknown $\{(x_i)\}_{i=1}^{N_I}$ needs to be solved. In order to approximate each governing equation, there is a $N_I \times N_I$ linear system to be solved in each iteration, as N_I denotes all the interior nodes. Random distribution of nodes can increase the condition number of the matrices and consequently leading to ill-conditioning and a source of instability. The idea of using the meshless method is to preform simulations on PC-Level creating huge matrices is not applicable in this situation. The solution can be highly expensive when large matrices need to be inverted. In addition, the global RBF meshless method is applicable for small and regular domains but is not practical for large complex geometries or/and physics involving many nodes.

2.4. Localized RBF

The second type of RBF approximation is the localized meshless method. The drawbacks arising from the global meshless method can be largely overcome by using the localized meshless method. In fact, the idea of localized RBF interpolation starts with the concept that any irregular domain can be interpolated by doing collocation about a small number of nodes in each subdomains. The basis functions are calculated utilizing local points in each subdomain. Using subdomains to approximate a set of unknowns result in a more efficient and accurate solution method when compared to global interpolation techniques. To create the subdomains, for each interior node P_j , we assign a subdomain including m nearest nodes in the subdomain of

influence $\Omega_j = P_{j,k=1}^m$. Here the reference node is marked P_j , while the index k changes between 1 to m consisting of the reference point itself. Fig. (2.1) shows different subdomain stencils for 9, 15 and 20 nodes. The reference point and its supporting points x_j^j , $j = 1, 2 \dots m$ are identified by the red color.

Once again, consider Eq. (2.1). To approximate the function or its derivatives, we can use support domains instead of the whole domain. Here, the function value, u , can be interpolated as:

$$u(x) = \sum_{j=i}^m u_j \phi_j(x) \text{ in } \Omega_j \quad 2.6$$

and the differential operator L , can be calculated as

$$\sum_{j=1}^m (L\phi_j)(x_j) u_j = f(x_j) \text{ in } \Omega_j \quad 2.7$$

where the subdomain Ω_j is a small domain surrounding the reference point. Comparing Eq. (2.6) with Eq. (2.2) we see that the only difference in formulation is the size of the matrix, i.e., $N \times N$ versus a small matrix of $m \times m$. This feature of the localized meshless method has some attractive advantages such as parallel processing capability (because of the presence of subdomains), easy calculation of the matrix inversion, and the ability for a fully independent approach at the problem setup stage.

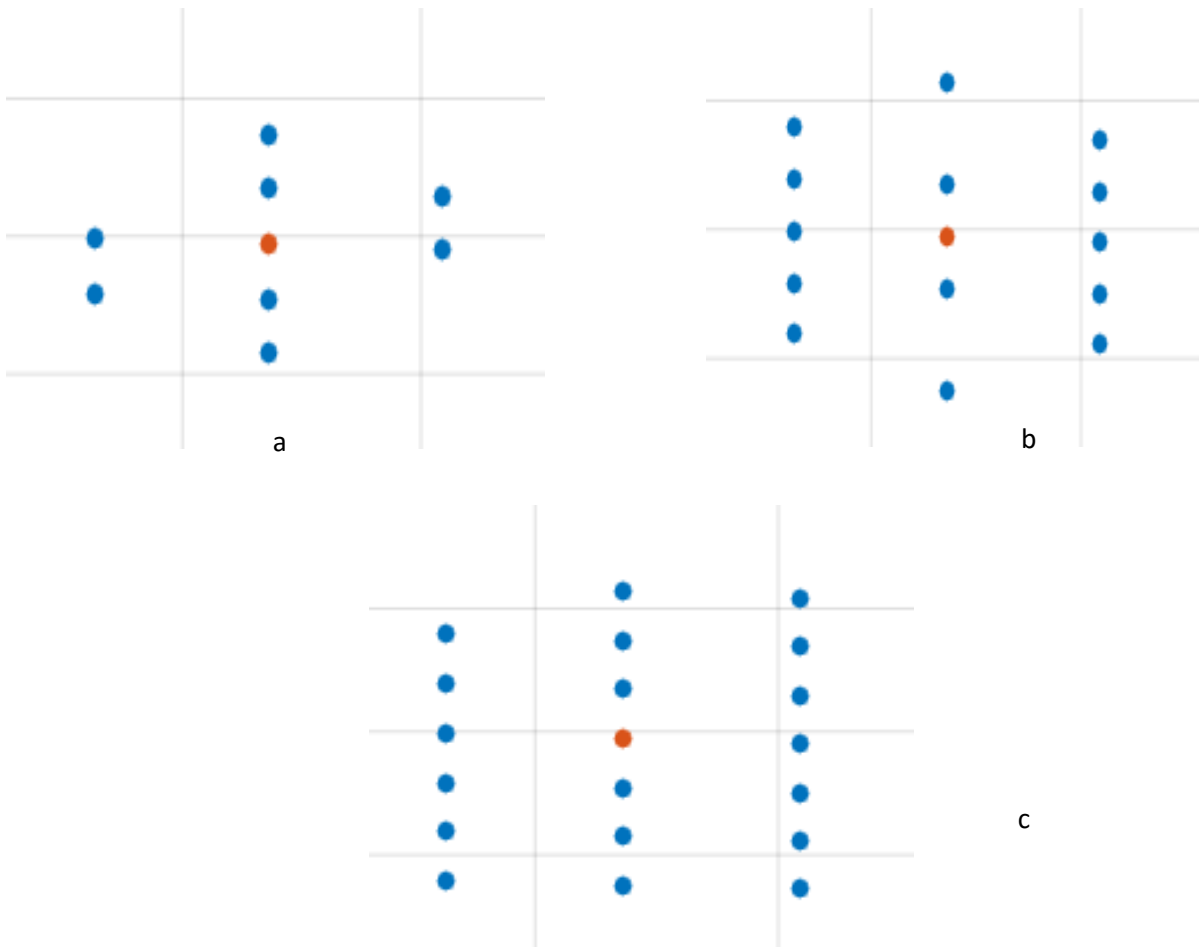


Figure 2.1 Size of subdomain a) 9 node stencil, b) 15 node stencil, c) 20 node stencil

2.5. RBF-DQ

In this section, the formulation of the Localized RBF differential quadrature, LRBFDQ, method is given. In this method, the function is approximated by RBFs and all the derivatives are approximated by differential quadrature (DQ). Therefore, the derivative of the function at the reference point is interpolated by a weighted linear sum of function values at a number of discrete nodes inside its subdomain. It should be noted that these weight coefficients are only a function of the space between the distributed nodes. Shu et al. (2003) showed that the weighting

coefficients can be easily computed utilizing a linear vector space and a function approximation.

In this study the unknown function $f(x)$ is approximated by the linear combination of the multiquadrics (MQs). As mentioned before, they are the most accurate basis function among various RBF-based interpolation methods. It should be noted that in the localized RBF-DQ method, the subdomain at other nodes are different. The size of the subdomain can also be changed, making the method more flexible. For example, the approximation of the m^{th} order derivative of a function $f(x)$ at the node x_I by RBF-DQ can be expressed as

$$\frac{\partial^m f}{\partial x^m} = \sum_{j=0}^{N_I} w_{I,j}^m f(x_j^j), \quad j=0,1,2,\dots,N_I \quad 2.6$$

$f(x_j^j)$ = function values at the distributed points

$w_{I,j}^m$ = RBF-DQ weight coefficients at the points

where x_j^j are the positions of supporting point x_j , and $x_j^0 = x_j$. The symbol N_I shows the number of supporting nodes within the subdomain of the reference point x_I . For simplicity of notation

$w_k(x)$ is used to replace $w_k(\|x - x_k\|)$, where $\|x - x_k\|$ is the Euclidean distance.

According to the principle of superposition, all the basis functions should satisfy the relation described in Eq. (2.6), i.e. expressed in matrix form as

$$\frac{\partial^{(m)}}{\partial x^m} \begin{bmatrix} \phi_1(x_I) \\ \phi_2(x_I) \\ \vdots \\ \phi_N(x_I) \end{bmatrix} = \begin{bmatrix} \phi_1(x_1) & \phi_1(x_2) & \cdots & \phi_1(x_N) \\ \phi_2(x_1) & \phi_2(x_2) & \cdots & \phi_2(x_N) \\ \vdots & \vdots & \ddots & \vdots \\ \phi_N(x_1) & \phi_N(x_2) & \cdots & \phi_N(x_N) \end{bmatrix} \begin{bmatrix} w_{i1}^m \\ w_{i2}^m \\ \vdots \\ w_{iN}^m \end{bmatrix} \quad 2.7$$

From Eq. (2.3), one can easily obtain the first order derivative of $\frac{\partial^1 \phi_i}{\partial x^1}$ as:

$$\frac{\partial^1 \phi_k(x_i, y_i)}{\partial x^1} = \frac{x_i - x_j}{\sqrt{(x_i - x_j)^2 + (y_i - y_j)^2 + c^2}} \quad 2.8$$

In a similar manner, the weighting coefficients of the y-derivatives can also be computed. Also, one can obtain the second and higher order derivatives of $\phi(x)$ by differentiating Eq. (2.3) successively. The second derivative can be expressed as

$$\frac{\partial^2 \phi_k(x_i, y_i)}{\partial x^2} = \frac{(x_i - x_j)^2 + c^2}{(\sqrt{(x_i - x_j)^2 + (y_i - y_j)^2 + c^2})^3} \quad 2.9$$

Since the weighting coefficients are based on the local position of supporting points in the subdomains, this approach is very suitable in dealing with nonlinear problems. Since the derivatives are also evaluated directly from the function values at the nodes distributed in the domain, the method can be systematically employed to solve the both linear and nonlinear equations. Another interesting property of RBF-based DQ method is that it is truly meshfree, i.e., all the information required about the nodes in the domain only depends on their positions.

2.6. Summery

In this chapter, we examined the radial basis function. There are few studies on the comparison of global and localized meshless. For example, Islam et al. (Yao, Siraj-Ul-Islam, & Sarler, 2012) showed the benefits of the localized meshless method for the case of the diffusion-reaction problem in three dimensions. Waters and Pepper (2015) proved the advantages of the localized method over the global method by comparing results for different incompressible flow benchmarks. Sarler and (Šarler & Vertnik, 2006) showed that the drawbacks of global meshless method can be resolved using localized meshless.

To review the localized RBF meshless method, the main computational domain is divided to a number of smaller subdomains. For each interior node, a small matrix of $m \times m$ where m is the number of nodes in each subdomain is inverted. The size of the matrix in the global meshless method is $N \times N$, where N is the total nodes in the computation domain. Since the distance between nodes are different for the random distribution, the order of matrix entries can be very different. Thus, the matrix can become ill-conditioned. The matrices in the localized meshless method are small and the concern of ill-conditioned matrices are mitigated. Shape parameters and time steps are to the distribution of nodes and velocities in both methods.

3.1. Introduction

The advantages of using a localized meshless method have been discussed in the previous chapter. In this chapter, we implement the localized RBF-DQ to simulate several incompressible benchmark problems to verify the scheme. The results of this chapter are compared with the results of Waters and Pepper (2015). These benchmarks are coupled to the Navier-Stokes equations with convective heat transfer.

There are different classical benchmarks are examined here: (1) the moving wall cavity, (2) natural convection inside a closed square, and (3) convective flow over a backward-facing step. We examine the localized RBF-DQ method on each of these benchmarks, and discuss the consistency and accuracy of the scheme.

3.2. Flow Solution

3.2.1. Governing Equations

The nondimensional form of governing equation are used for incompressible laminar flow with convective heat transfer effects. The following scaling relations are used in the governing equations for momentum and energy:

$$X = \frac{\hat{X}}{L}, V = \frac{\hat{V}}{\alpha/L}, p = \frac{\hat{p}}{\rho\alpha^2/L^2}, T = \frac{(\hat{T} - T_c)}{(T_h - T_c)}, t = \frac{\hat{t}}{L^2/\alpha} \quad 3.1$$

The, hats denote the dimensional variables. The dimensionless numbers, i.e. Reynolds number, Rayleigh number, Prandtl number, and Peclet number are defined as

$$\text{Re} = \frac{\rho VL}{\mu}, \text{Ra} = \frac{g\beta(T_h - T_c)L^3}{\alpha\nu}, \text{Pr} = \frac{\nu}{\alpha}, \text{Pe} = \frac{VL}{\alpha} \quad 3.2$$

The nondimensional forms of the governing equations can become written as

$$\nabla \cdot V = 0 \quad 3.3$$

$$\frac{\partial V}{\partial t} + V \cdot \nabla V = -\nabla p + C_{\text{visc}} \nabla^2 V + B \quad 3.4$$

$$\frac{\partial T}{\partial t} + V \cdot \nabla T = C_T \nabla^2 T \quad 3.5$$

The body force is defined as $B = \text{Pr Ra T}$ in the y direction for natural-convection problems. For all the other cases, $B = 0$.

Table 3.1 Constant parameter of each case

Case	C_{visc}	C_T	B
2-D cavity	1/Re		0
Natural convection	Pr	1	Pr.Ra.T
Flow with forced convection over backward-facing step	1/Re	1/Pe	0

3.2.2. Projection Method

The projection method is an efficient method to simulate time dependent Navier-Stokes equations. Chorin (1968), Chorin and Marsden (1993), and Temam (2000) were the first people who introduced this effective method for solving the incompressible fluid flow cases. An intermediate velocity, V^* , is calculated explicitly without involving the pressure gradient in Chorin's method. Therefore the equation takes the form:

$$\frac{V^* - V^n}{\Delta t} = -(V^n \cdot \nabla)V^n + C_{visc} \nabla^2 V^n + B \quad 3.6$$

where B and C_{vis} are defined in the table and V^n is the velocity vector at the present time. The pressure gradient can be obtained from the expression

$$\frac{V^{n+1} - V^*}{\Delta t} = -\nabla p^{n+1} \quad 3.7$$

Rewriting the above equation for the velocity at the (n+1) level, we obtaine

$$V^{n+1} = V^* - \Delta t \nabla p^{n+1} \quad 3.8$$

The pressure at n+1 is required to compute the above equation. A divergence-free constraint is applied on the velocity field at this next time level, $\nabla \cdot V^{n+1} = 0$, to compute the pressure. The resulting equation is a Poisson equation for p^{n+1} in the following form:

$$\nabla^2 p^{n+1} = \frac{\nabla \cdot V^*}{\Delta t} \quad 3.9$$

rearranging the above equation,

$$V^{n+1} = V^* - \Delta t \nabla p^{n+1}$$

which is the standard Hodge decomposition if the boundary condition for p on the domain boundary is $\nabla p^{n+1} \cdot n = 0$. Thus, the boundary condition for p is

$$\frac{\partial p^{n+1}}{\partial n} = 0 \quad 3.10$$

It should be noticed that the continuity equation needs to be satisfied through the simulation domain. Therefore, the velocity field should satisfy the continuity equation in Chorin's method after each iteration. In this chapter we implement the following steps to simulate the projection method:

- First we solve Eq. (3.6) by using the velocity at the present time and calculate the intermediate velocity. To make sure that the boundaries are satisfied, we fixed them after each iteration. Waters and Pepper (2015) showed that for the lid driven cavity and natural-convection problems, solving the convection term explicitly gives better accuracy. On the other hand, for the flow over a backward step, treating the convection part implicitly, yielded better accuracy. Thus the equation to solve for the first two cases is:

$$\frac{V^* - V^n}{\Delta t} - C_{visc} \nabla^2 V^* = -(V^n \cdot \nabla) V^n + B \quad 3.11$$

and for the backward step,

$$\frac{V^* - V^n}{\Delta t} - C_{visc} \nabla^2 V^* + (V^n \cdot \nabla) V^* = B \quad 3.12$$

This is the predictor step.

- For the second step, the intermediate velocity calculated from the previous step is used to compute the pressure at the time $n+1$ by using Eq. (3.9). The divergence-free constrain is applied in this step. It should be noted that the boundary condition for pressure in Eq. (3.10) need to be applied.

- Finally, a new pressure is subjected to Eq. (3.8) to update the velocity. All the domain variables are now updated to the next time level.

3.3. Benchmarks Examination

In this section, the present RBF-DQ algorithm is examined for the different benchmark cases. These cases are used to validate the code. As in the previous studies, we also did the simulation on a nonuniform distribution of points.

3.3.1. Moving Wall Cavity

The cavity problem is one of the most popular benchmark problems employed to validate and verify a new scheme. Commercial CFD software primarily use this benchmark to evaluate accuracy and feasibility of their code. For $Re=100$, the solution is characterized by the presence of a main circulation zone and counter rotating vortices in the corners of the squares.

3.3.1.1. Problem setup

For the cavity problem, the computational domain is a square of $(0 \leq x \leq 1, 0 \leq y \leq 1)$. The top wall is moving at $u=1$ (dimensionless value) and the remaining walls are stationary. The boundary conditions are

Upper edge: $u=1, v=0$

Other edges: $u=v=0$

3.3.1.2. Localized RBF-DQ

To define the problem, the same number of total nodes as in reference study (Waters & Pepper, 2015) 240 points is used here. 200 of these nodes are interior nodes and 40 are boundaries. The size of the subdomains $\Omega_j = P_{j,k=1}^m$ are set to 9 including the reference point itself; the shape parameter is $c=0.2$. This value is higher than in a the regular localized RBF (Waters & Pepper, 2015).

The plot of the velocity vector for the localized RBF and the localized RBF-DQ are shown in Fig.(3.1) for $Re=100$. Results of the present method are in good qualitative agreement with results shown in reference paper (Waters & Pepper, 2015).

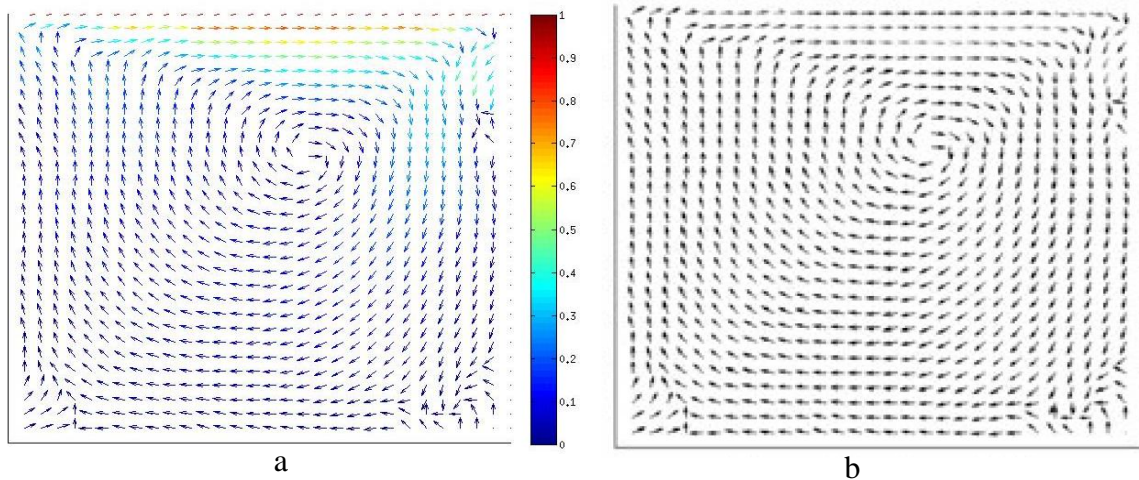


Figure 3.1 Cavity problem: Velocity vector plots of two scheme for $Re=100$. a) Present Scheme, b) Reference scheme (Waters & Pepper, 2015)

3.3.2. Natural Convection inside a close square

3.3.2.1. Problem setup

As in the cavity problem, the simulation domain is $(0 \leq x \leq 1, 0 \leq y \leq 1)$ for the natural convection problem with a Rayleigh number of 10^3 . The velocity of the walls are equal to zero, the left wall is heated, and the right wall is cold. For the top and bottom walls, an adiabatic condition is set. The boundary conditions are

Left side: $u=v=0, T=1$

Right side: $u=v=0, T=0$

Upper and Lower sides: $u=v=0, \partial T/\partial y=0$

3.3.2.2. Localized RBF-DQ

The implementation is the same as the cavity problem except that the size of the subdomain changes to 20 and the shape parameter changes to 0.05. The reference paper (Waters

& Pepper, 2015) used 50 nodes for each subdomain. They also fixed the shape parameter to 0.02, which is relatively small.

Velocity vector plots, and temperature contours are shown in Fig. (3.2) and Fig. (3.3), respectively. The velocity vectors are similar in both methods but the temperature isothermal plot is a bit different. In the localized RBF the isothermal lines are not normal to the adiabatic walls. However, for the case of RBF-DQ, they are normal to the top and bottom walls. The reason is due to the different size of the subdomains and different value for shape the parameter.

3.3.3. Flow with Forced Convection over a Backward-Facing Step

3.3.3.1. Problem setup

The simulation domain for this case is highly elongated and the scale is 30×1 . The velocity and temperature at the inlet is define as:

$$u(y) = \begin{cases} 0 & 0 < y < \frac{1}{2} \\ y(2y-1) & \frac{1}{2} \leq y \leq 1 \end{cases}$$

$$v(y) = 0$$

$$T(y) = \begin{cases} \frac{\partial T(y)}{\partial x} & 0 < y < \frac{1}{2} \\ (1-(4y-3)^2) \left[1 - \frac{1}{5}(4y-3)^2 \right] & \frac{1}{2} \leq y \leq 1 \end{cases}$$

and for top and bottom wall:

$$u(x)=v(x)=0$$

$$\nabla T.n = 32/5$$

where n is the inward unit vector normal to the domain boundary.

3.3.3.2. Localized RBF-DQ

The total number of nodes for this case is 7680 and the size of the subdomain is 11. The shape parameter is fixed to $c=0.05$.

The Dirichlet boundary points were updated explicitly after each time step. For $Re=100$, both of methods, produced comparable results, as shown in Fig. (3.4) for the streamlines and Fig. (3.4) for the isotherms. The present method captures the reattachment length as $2h$ which is the same as the result from Armaly et.al. experiment (Armaly, Durst, Pereira, & Schonung, 1983).

3.4. Summery

The purpose of this chapter was to compare the results of the present method utilizing well-known incompressible benchmarks. The results from the present scheme for the three cases of incompressible flow indicate that it produces accurate solution. We also distributed the nodes randomly to examine the sensitivity of the code to the location of the nodes. The results show that the position of the nodes does not affect the results. Good agreement was also achieved when compared with the results obtained by Waters and Pepper (2015). For complex cases, some preprocessing is required to obtain good results. Also, care must be exercised as the shape parameter changes case by case. The method can be easily combined with other conventional methods to enhance solution accuracy, e.g., in cases involving conjugate heat transfer.

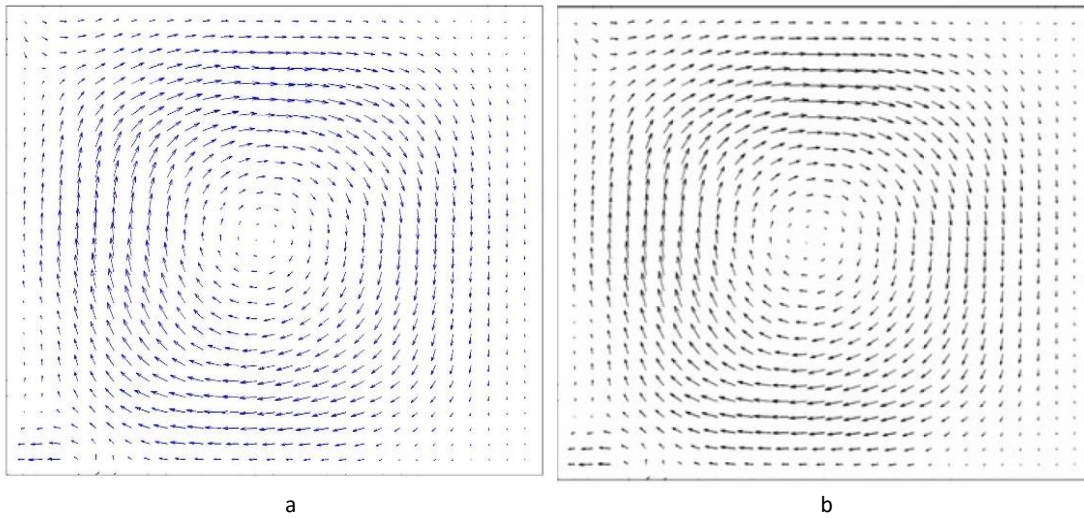


Figure 3.2 Natural convection: Velocity Vector plots of two scheme for $Pr=0.71$, and $Ra=10^3$. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)

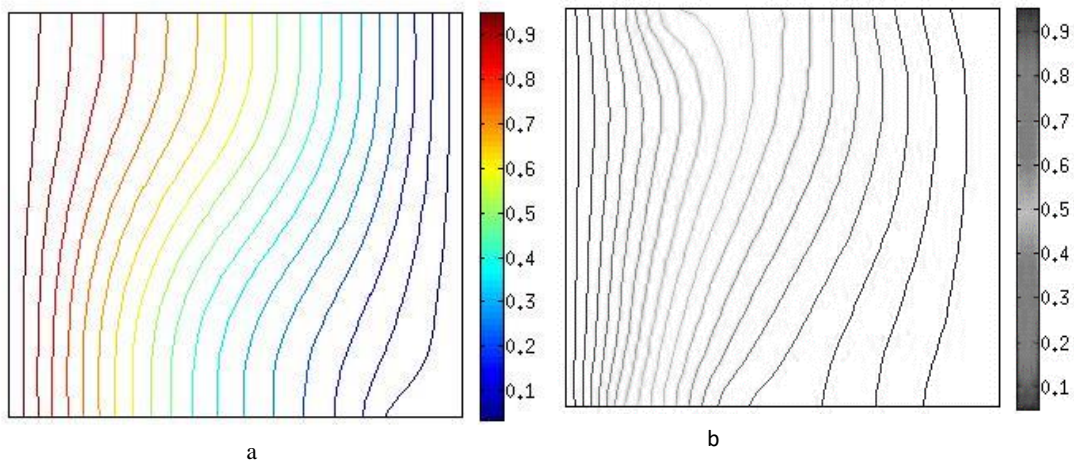


Figure 3.3 Natural convection: Temperature contours of two schemes for distribution with $Pr=0.71$, $Ra=10^3$. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)

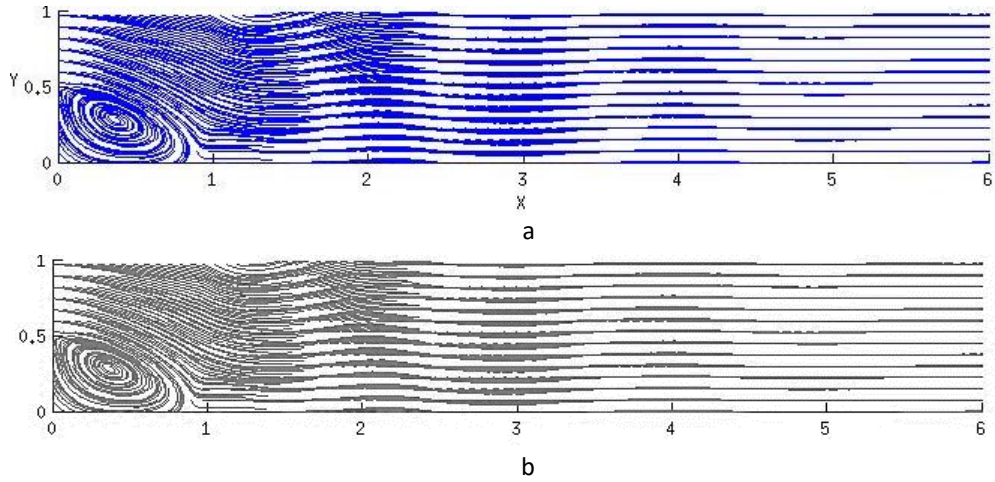


Figure 3.4 Backward facing: Streamline for fluid flow. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)

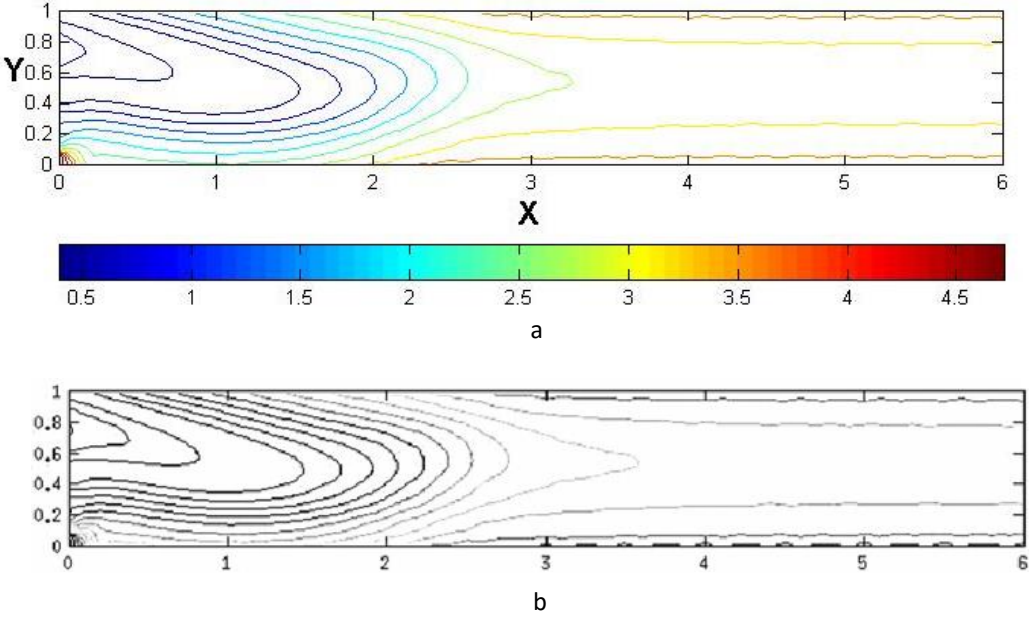


Figure 3.5 Backward facing: isothermal for fluid flow. a) Present scheme, b) Reference scheme (Waters & Pepper, 2015)

4.1. Introduction

The Navier-Stokes equations are generally displayed in one of two forms. The first form is introduced as the incompressible flow equation, as mentioned in the previous chapter. The second form is for compressible flow. Unlike the first form, the compressible flow equations allow the density of the fluid to change with the flow. In this chapter, we will discuss the Euler equation, which is one of the governing equations for the dynamics of a compressible flow without viscosity. The system originated from the general fluid flow equation i.e. Navier-Stokes equation in combination with equation of state. The principal goal of this chapter is to provide the detailed methodology of the blended RBF-DQ and the Riemann solvers and to apply the algorithm to a benchmark for validation and verification.

4.2. The Euler Equations

Our focus in this section is deals with a hyperbolic system of PDEs that is subject to hyperbolic conservation laws. These types of equations need more requirements on the discretization techniques and are more complicated to solve than general parabolic or elliptic equations. Many studies have been carried out in this area to develop reliable and accurate schemes with high-resolution. Since many of the problems in this category are expensive to investigate experimentally a reliable simulation can be helpful.

For the case of high speed compressible fluid flow, the effect of the boundary layer and viscosity is neglected, when assuming inviscid flow. The Navier-Stokes equations become hyperbolic, resulting in the Euler equations. The Euler equations are a system of non-linear

equations that can produce discontinuities throughout the domain even in the case of smooth initial conditions. In this section, we consider the unsteady Euler equations and the characteristics of this equation.

There is some freedom in choosing the form of the governing equation describing the flow under consideration. A possible option is the primitive variables or physical variables, namely, mass density, pressure, and the x and y components of velocity for 2D domains. An alternative choice is the conservative variables, containing of mass density, the x-momentum component, the y-momentum component, and the total energy per unit mass. Physically, these conserved quantities result naturally from the application of the fundamental laws of conservation of mass, Newton's Second Law and the law of conservation of energy. Computationally, there are some advantages in expressing the governing equations in terms of the conserved variables. In fact, the primitive form of the Euler equations fails at shock waves. It gives the wrong jump conditions; consequently, they give the wrong shock strength, the wrong shock speed and thus the wrong shock position. Shock waves in air are small transition layers of very rapid changes of physical quantities such as pressure, density and temperature. The transition layer for a strong shock is of the same order of magnitude as the mean-free path of the molecules, that is about 10^{-7} m. Therefore replacing these waves as mathematical discontinuities is a reasonable approximation. A work by Hou and Floch (1994) show that non-conservative schemes do not converge to the correct solution if a shock wave is present in the solution. The classical result of Lax and Wendroff (1960), on the other hand, says these problems can be solved by using the conservative form of the equations. Therefore, it is prudent to work with conservative methods if shock waves occur in the solution.

In the following section, we review the Euler equation and its properties.

4.2.1. Conservation-Law Form

The Euler equations in two dimension can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad 4.1$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = 0 \quad 4.2$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2 + p)}{\partial y} = 0 \quad 4.3$$

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho u E + up)}{\partial x} + \frac{\partial(\rho v E + vp)}{\partial y} = 0 \quad 4.4$$

These equation represent conservation of mass, momentum in x and y, and energy, respectively. Here ρ is the density, u is the x-velocity component, v is the y-velocity component, p is pressure, and E is the total energy / mass, and can be expressed in terms of the specific internal energy and kinetic energy as:

$$E = e + \frac{1}{2}(u^2 + v^2) \quad 4.5$$

The equations are closed with the addition of an equation of state. A common choice is the gamma-law equation of state:

$$p = \rho e(\gamma - 1) \quad 4.6$$

where γ is the ratio of specific heats for the gas/fluid (for an ideal, monatomic gas, $\gamma = 5/3$), but any relation of the form $p = p(\rho, e)$ will work. In the case of non-linear systems of conservation

laws, the character of the flux function is determined by the Equation of State. One thing we notice immediately is there is no need for temperature in this equation set. However, when source terms are present, we need to obtain temperature from the equation of state.

4.3. RBF-DQ Implementation

We write the conservative form of the Euler equations in the form of flux vectors. In this form, the equations can be written as:

$$U_t + [F(U)]_x + [G(U)]_y = 0 \quad 4.7$$

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, F(U) = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uE + up \end{bmatrix}, G(U) = \begin{bmatrix} \rho v \\ \rho vu \\ \rho v^2 + p \\ \rho vE + vp \end{bmatrix}$$

For discretization of the spatial derivatives in Eq. (4.7), the localized RBF-DQ is applied.

The meshless approximation of the Euler equation can be expressed as:

$$\frac{\partial U_i}{\partial t} = - \sum_{j=0}^{N_i} [w_{i,j}^{1x} F(U_{i,j}) + w_{i,j}^{1y} G(U_{i,j})] \quad 4.8$$

where $w_{i,j}^{1x}$ and $w_{i,j}^{1y}$ are the coefficients for the first order derivatives in the x and y directions, respectively. The points used for discretization are not located at the supporting nodes, and $U_{i,j}$ are the conservative values at the mid-points between the support point j and the reference point i, as shown in Fig. (4.1).

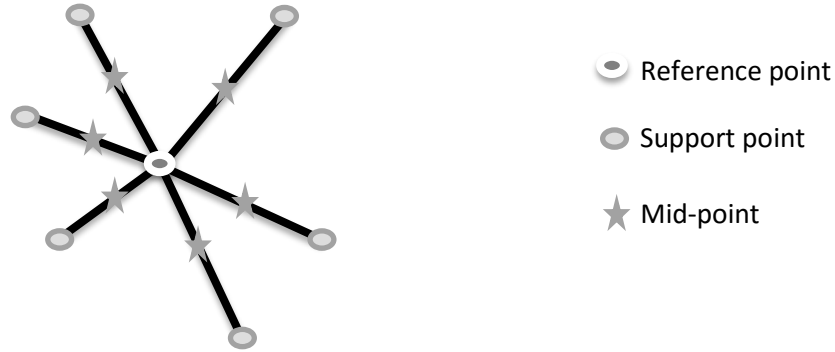


Figure 4.1 Configuration in subdomain

Shu et al. (2005) defined new flux and approximation functions for the mid-points. The new flux defined as

$$E = n_x F(U_{i,j}) + n_y G(U_{i,j}) \quad 4.9$$

in which n_x and n_y for each support point is expressed as

$$n_x = \frac{w_{i,j}^{1x}}{\sqrt{(w_{i,j}^{1x})^2 + (w_{i,j}^{1y})^2}}, \quad n_y = \frac{w_{i,j}^{1y}}{\sqrt{(w_{i,j}^{1x})^2 + (w_{i,j}^{1y})^2}} \quad 4.10$$

Then, by defining a new approximate function, $W_{i,j}$

$$W_{i,j} = \sqrt{(w_{i,j}^{1x})^2 + (w_{i,j}^{1y})^2} \quad 4.11$$

the discretization form turns to a new form which is

$$\frac{\partial U_i}{\partial t} = - \sum_{j=0}^{N_I} W_{i,j} E(U_{i,j}) \quad 4.12$$

N_I denotes the size of the subdomain for the reference node i and $E(U_{i,0}) = E(U_i)$. The new flux E , can be measured by the weighted linear sum of the new fluxes at the reference point and the mid-

points in the i^{th} subdomain. Therefore, for highly convective problems, calculation of the new fluxes at the mid-point is important and needs to be efficient as well as accurate.

4.4. Upwinding Scheme

There are different types of stabilization techniques and shock capturing schemes. The common feature of all these techniques is that they add a specific amount of numerical diffusion to insure stability while preserving accuracy to a desired degree.

For high order accurate schemes, the fluxes at the mid-points between the related points are denoted by the follow general equation:

$$\bar{E} = \frac{1}{2}(E_R + E_L) - d \quad 4.13$$

where an interface flux \bar{E} consists of the central average of normal fluxes and a diffusive flux, d , between the reference node (which is the left side and the support node (which the right side node). To eliminate nonphysical oscillations caused by discontinuities and steep gradients, we must determine the flow direction and the influence of upstream or downstream. Since the RBF-DQ is not a mesh based scheme, it cannot identify the direction of this influence. A solution to this problem is to distinguish the directions of wave propagations of the considered hyperbolic system to calculate the flux at the mid-point. An upwind scheme can be employed to evaluate the new fluxes at the mid-point. Godunov's method (Godunov, 1959) is one of the most popular upwind schemes. This method has been used to solve Riemann problems, and can provide the exact value of numerical flux at the mid-point. The scheme is convenient for the computation of the flux in Eq. (4.12). This is achieved by substituting the function values at the reference point i , and the specific supporting point k to set up a local 1D Riemann solution. It should be noted that

this flux evaluation still holds for the meshfree feature. For nonlinear equations, the Riemann problem requires trial an error, which can increase solution time. One way to reduce this burden is to employ an approximate Riemann solver to evaluate the fluxes at the mid-points in Eq. (4.12). In this study we use Roe's approximate Riemann solver (Roe, 1981), which is one of the more common schemes.

4.4.1. Roe's approximate Riemann solver

Evaluating numerical diffusion requires some pre-analysis. Roe's scheme is one of those solvers that perform these calculation to evaluate artificial diffusion. This method decomposes the conservative variables into characteristic waves (LeVeque, 1992). The main principle behind such schemes is that, given the characteristic decomposition of the waves, one can diagonalize an approximate Jacobian, A_{ij} satisfying

$$A(U_R - U_L) = E_R - E_L \quad 4.14$$

i.e.,

$$A = M\Lambda M^{-1} \quad 4.15$$

where Λ includes the eigenvalues denoting the speeds of individual waves, and M is the matrix indicating the transformation of conservative variables to characteristic variables. The columns of M are eigenvectors of A . With this diagonalization, the diffusion part can be calculated as:

$$d = |A|(U_R - U_L) \quad 4.16$$

where $A = M\Lambda M^{-1}$, is Roe's averaging matrix and produces an upwind scheme with different dissipation added to each characteristic wave to eliminate oscillations. The averaging denoted a specific construction to calculate the Roe's matrix.

For given states of the conservative variables on the both sides of the mid-point, the interface Jacobian, A , is evaluated using specially averaged variables. For a 2D domain,

$$\begin{aligned}\rho &= \sqrt{\rho_L \rho_R} \\ u &= \frac{\sqrt{\rho_L} u_L + \sqrt{\rho_R} u_R}{\sqrt{\rho_L} + \sqrt{\rho_R}} \\ v &= \frac{\sqrt{\rho_L} v_L + \sqrt{\rho_R} v_R}{\sqrt{\rho_L} + \sqrt{\rho_R}} \\ H &= \frac{\sqrt{\rho_L} H_L + \sqrt{\rho_R} H_R}{\sqrt{\rho_L} + \sqrt{\rho_R}}\end{aligned}\tag{4.17}$$

The result of the Jacobian for a 2D domain can be computed as:

$$\frac{\partial E}{\partial U} = \begin{bmatrix} 0 & n_x & n_y & 0 \\ n_x(\gamma-1) * \frac{(u^2 + v^2)}{2} - u * u_n & u_n - (\gamma-2) * n_x * u & n_y * v - (\gamma-1) * n_x * v & n_x * (\gamma-1) \\ n_y(\gamma-1) * \frac{(u^2 + v^2)}{2} - v * u_n & n_x * v - (\gamma-1) * n_y * u & u_n - (\gamma-2) * n_y * u & n_y * (\gamma-1) \\ u_n \left((\gamma-1) * \frac{(u^2 + v^2)}{2} - H \right) & n_x * H - (\gamma-1) * u * u_n & n_y * H - (\gamma-1) * v * u_n & \gamma * u_n \end{bmatrix}\tag{4.18}$$

for the direction normal to the interface with

$$M = \begin{bmatrix} 1 & 0 & 1 & 1 \\ u - c * n_x & -n_y * c & u & u + c * n_x \\ v - c * n_y & n_x * c & v & v + c * n_y \\ H - u_n * c & u_t & \frac{1}{2}(u^2 + v^2) & H + u_n * c \end{bmatrix}\tag{4.19}$$

$$\Lambda = \begin{bmatrix} u_n - c & 0 & 0 & 0 \\ 0 & u_n & 0 & 0 \\ 0 & 0 & u_n & 0 \\ 0 & 0 & 0 & u_n + c \end{bmatrix} \quad 4.20$$

with where n_x , n_y are the components of the unit interface normal, $u_n = u.n_x + v.n_y$ is the velocity component normal to the interface, and H is the enthalpy evaluated using interface quantities.

Utilizing the approximate Roe's solver, the fluxes at the mid-point can be computed by

$$E(U_L, U_R) = \frac{1}{2} [(E(U_L) + E(U_R)) - |A|(U_L - U_R)] \quad 4.21$$

where $E(U^L)$, $E(U^R)$ and $E(U_L, U_R)$ indicate the flux at the reference node, supporting node and the mid-point respectively. The L and R subscript is just chosen for simplicity. The subscript L indicates the domain variable at the reference node and the subscript R denotes the supporting node. The symbol A is the constant Jacobian matrix, which approximates the Jacobian matrix defined by $\frac{\partial E}{\partial U}$.

Recalling Eq. (4.21), Roe's scheme presented here only has first-order accuracy. In fact, the flux between the mid-point and the points on both sides, is constant, and represents a first order spatial approximation. To increase the order of Roe's approximation, we need to construct a high order spatial approximation. For the conventional methods, changing the order of the polynomial will increase the order of accuracy. For the meshless method, we can extrapolate the conservative values to both sides of the mid-point and approximate a higher order flux. Therefore, the equation can be expressed as

$$E(U_L, U_R) = \frac{1}{2} [(E(U^L) + E(U^R)) - |A|(U^L - U^R)] \quad 4.22$$

in which the U^L and U^R indicate the conservative values at the mid-point, shown in Fig. (4.2), computed from the reference node and supporting node, respectively.



Figure 4.2 Position of conservative variables

The new Jacobian matrix, A^* , is computed at the mid-point by using the new conservative values. In mesh-based conventional methods, domain variables at the mid-point can be obtained by upwind approximation using functional values at specific mesh points. This approach is difficult when using the local RBF-DQ method, since the nodes are nonuniformly distributed. Since the derivatives can be calculated using RBF-DQ, the values at the mid-point can be calculated easily. Here, we just need to use the first two terms of the Taylor series expansion, i.e., only the function value and its derivatives at the reference point or the supporting point are needed to compute the mid-point variables.

In this study, the equations to calculate the conservative values on the left and right side of the mid-point (U^L and U^R , respectively) are defined

$$U^L = U_L + \Delta U_L \quad 4.23$$

$$U^R = U_R + \Delta U_R$$

where ΔU_L and ΔU_R are obtained from:

$$\Delta U_L = \frac{\partial U_R}{\partial x} \cdot \left(\frac{x_R - x_L}{2} \right) + \frac{\partial U_R}{\partial y} \cdot \left(\frac{y_R - y_L}{2} \right) \quad 4.24$$

$$\Delta U_R = \frac{\partial U_R}{\partial x} \cdot \left(\frac{x_L - x_R}{2} \right) + \frac{\partial U_R}{\partial y} \cdot \left(\frac{y_L - y_R}{2} \right)$$

To decrease the dissipation of using low shape parameters in the smooth regions, we combine the blended method with the above formula to calculate ΔU_L and ΔU_R . This means that similar formula as Eq. (4.24) are used with a high shape parameter for completely smooth regions and two different values are saved for each of ΔU_L and ΔU_R . Since the higher scheme can cause spurious numerical oscillation around discontinuities and high steep gradient regions, a monotonic solution can not be obtained unless special treatment is enforced. Therefore, to eliminate the presence of these oscillations, an essential principle was then suggested in the reconstruction procedure, i.e., a ‘limiter’. In this study, after the employment of a limiter, the variables are modified as

$$U^L = \begin{cases} U_L + s^{High} \Delta U_L^{High} & \min(U_k - U_L) \leq s \Delta U_L \leq \max(U_k - U_L) \\ U_L + s^{Low} \Delta U_L^{Low} & \min(U_k - U_L) \leq s \Delta U_L \leq \max(U_k - U_L) \\ U_L & else \end{cases} \quad 4.25$$

in which the superscript High and Low refer to high and low shape parameters and subscript k denotes all the support nodes of reference point i, and s denotes a Van Albada limiter (Sweby, 1984), which for the high shape parameter can be expressed as

$$s^{High} = \left| \frac{2 * \Delta U_L^{High} * \left(\frac{U_R - U_L}{2} \right) + \varepsilon}{(\Delta U_L^{High})^2 + \left(\frac{U_R - U_L}{2} \right)^2 + \varepsilon} \right| \quad 4.26$$

where ε is a very small number (for example, $\varepsilon = 10^{-6}$), to prevent division by zero in a uniform flow region, where the flux difference is very small. For the case of a low shape parameter, we just need to substitute the low value into the equation.

4.5. Summery

To summarize this chapter we first discussed the Euler equation and then the methodology for capturing a shock wave. The blended localized RBF-DQ includes two approximation steps in the spatial discretization. These steps have a huge impact on the accuracy and consistency of the algorithm. The first approximation is to calculate the conservative values and consequently evaluate fluxes at the mid-point by using a good approximate upwinding scheme. We then approximate the divergence of the flux field by applying a weighted linear sum of function values at a number of discrete nodes inside the subdomain. The domain variable at the mid-point blends into three categories based on the position of the node. The following flowchart Fig. (4.3) illustrates the algorithm.

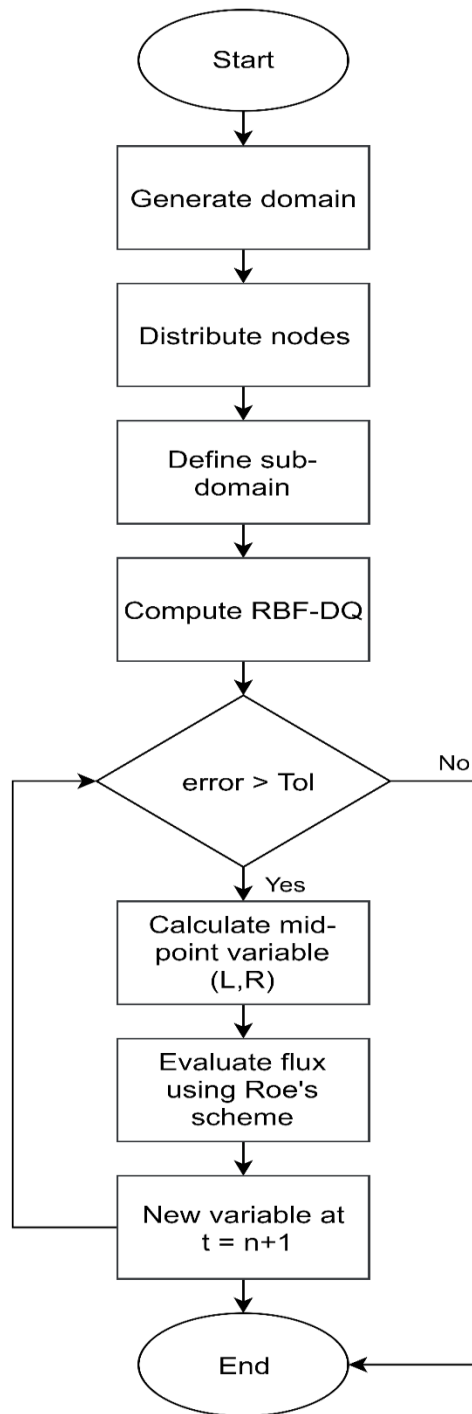


Figure 4.3 Present algorithm flowchart

5.1. Benchmark Problem

For the case of compressible flow with the presence of a discontinuity, a supersonic scramjet engine is simulated. Supersonic flow passes through the system, and the result is the presence of successive oblique shocks. The computational domain is displayed in Fig. (5.1). The sharp wedge on the top wall creates the first oblique shock, and the subsequent reflections from the bottom wall and the wedge surface generate the reflected waves. The exact solution was provided by Wang and Widhopf (1989) and we can clearly see in Fig. (5.7) the accurate position of the reflections. The objective of this section is to verify the accuracy of the present meshless method using a uniform distribution of nodes with a specific set of initial and boundary conditions.

5.1.1. Problem domain

As shown in Fig. (5.1), the top wall rotates -10.94° , generating a shock structure. Some similar, but simpler problems have been widely used as a benchmark for numerical schemes dealing with shocks (Tota, 2006). The total number of node is 1296, distributed uniformly in the domain as shown in Fig.(5.2) with 1177 interior nodes and 119 defining boundaries. The number of points within a subdomain is limited to 9, including the reference node itself for all the nodal distributions. Choosing nodes is an important issue in shock capturing problems. To more accurately capture the discontinuities the support nodes in different direction and nodes with the same direction along the joint line between the reference and support node were removed. A schematic of the nodal configuration for the subdomains is shown in Fig. (5.3).

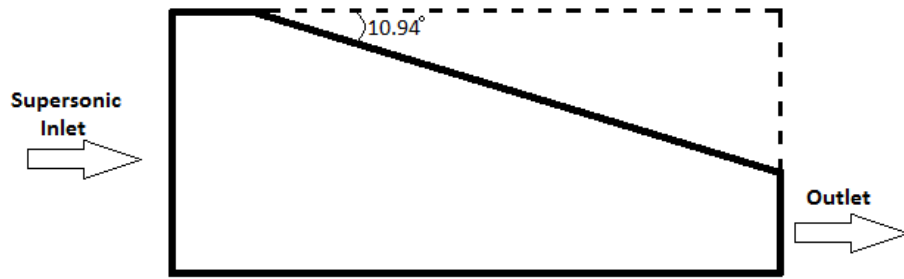


Figure 5.1 Benchmark domain

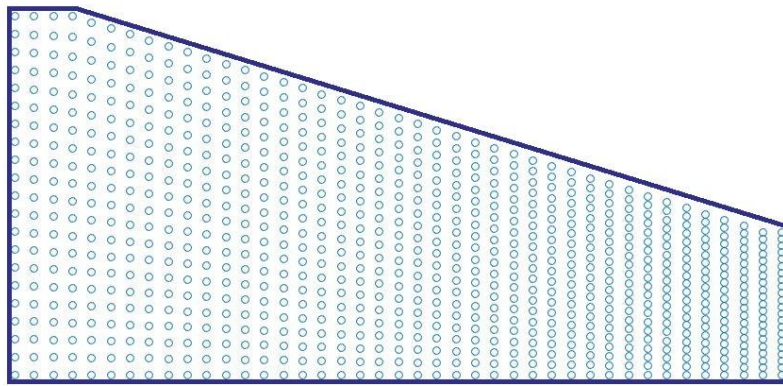


Figure 5.2 Node distribution

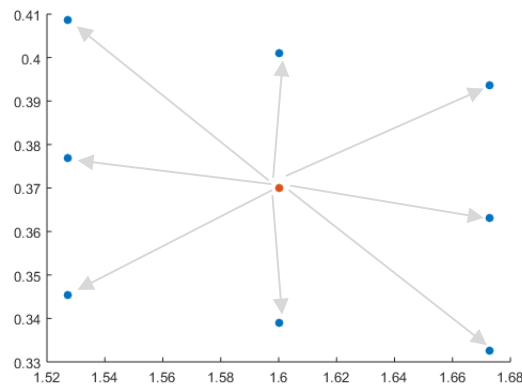


Figure 5.3 Configuration in subdomain

The flow variables at interior nodes are updated by the solution of Eq. (4.12) as time marches. The treatment is different for the case of boundary nodes. It is clear that appropriate boundary conditions are needed for the well-posed hyperbolic partial differential equation. Here, two classes of boundary conditions are considered: solid boundary conditions; and inflow and outflow boundary conditions.

5.1.2. Boundary conditions

For the top and bottom walls, no-flow was utilized. Since the flow is inviscid, a no-slip reflection cannot be applied. A no-flow boundary is imposed by maintaining the velocity normal to the solid boundary at 0 (the tangential velocity can be nonzero). To make the solution more accurate and enforce no-flow boundary conditions on the solid wall, additional nodes are introduced on the outside of the solution domain, shown in Fig. (5.4). Therefore, the flow variables are defined to ensure vanishing normal velocities at the wall.

Combined with the other boundary conditions, the flow variables can be defined at the reflected point R by the corresponding interior node I as

$$u_{nR} = -u_{nI} \tag{5.1}$$

$$u_{tR} = u_{tI}$$

$$\rho_R = \rho_I$$

$$e_R = e_I$$

If the solid boundary is at an angle, θ , below the horizontal, then the normal and tangential components of velocity can be written as

$$u_{nR} = -(u_I * \sin \theta + v_I * \cos \theta)$$

5.2

$$u_{tR} = u_I * \cos \theta - v_I * \sin \theta$$

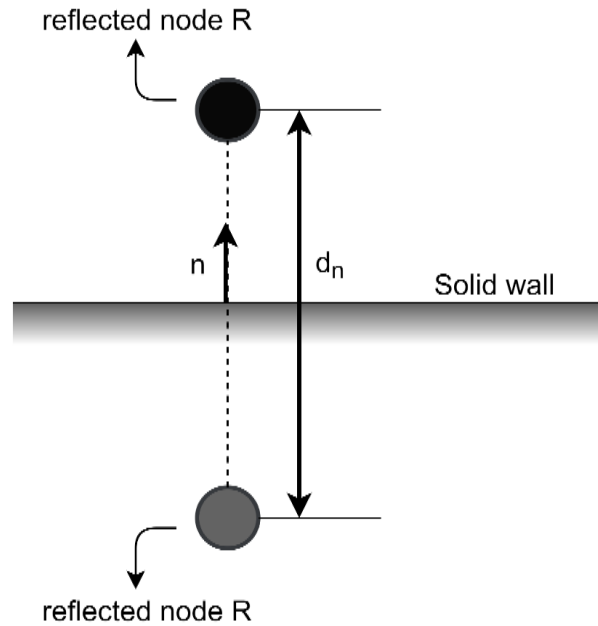


Figure 5.4 Wall boundary condition

Depending on the flow condition, careful consideration must be taken for the inlet and outlet boundary conditions. The method of characteristics used to identify the boundary condition on the inlet and outlet assuming a 1D Riemann relation. Note that the method considers the direction of the characteristic waves and the Riemann invariants throughout the system in an effort to ensure accuracy.

For the case of 1D Euler equations, the Riemann invariants are calculated as:

$$w_1 = \frac{P}{\rho^r},$$

5.3

$$w_2 = u_t$$

$$w_3 = u_n + \frac{2c}{(\gamma-1)},$$

$$w_3 = u_n - \frac{2c}{(\gamma-1)}$$

where u_n and u_t are the normal and tangential velocity components at the inlet or outlet.

Rearranging, we can obtain the primitive variables as:

$$u_n = \left(\frac{w_3 + w_4}{2} \right),$$

5.4

$$u_t = w_2$$

$$\rho = \left(\frac{c^2}{\gamma \cdot w_1} \right)^{\frac{1}{\gamma-1}},$$

$$p = w_1 \cdot \rho^r$$

Following the above equations and theory of characteristic waves for supersonic flow, no additional treatment is required. For a supersonic upstream, the domain variables are fixed from the free stream value. For the exit condition, the values are calculated from the interior nodes.

For the current benchmark, the inlet state is supersonic and its condition specified by prescribing the pressure, density and velocity vector as fixed. The inlet condition is the same as the reference value (Wang & Widhopf, 1989) for the nondimensional form of governing equation, and they are shown in Table (5.1).

Since the flow at the outlet is still supersonic, the following first-order extrapolation is implemented,

$$U_{outlet} = 2 * U_{outlet-1} - U_{outlet-2} \quad 5.5$$

Table 5.1 Inlet condition

Pressure	0.714
Density	1.0
Mach Number	2.9
Y component of velocity	0

The computations are performed with different shape parameters and local timesteps. For the temporal discretization in Eq. (4.12), a local timestep based of the CFL condition is used for the explicit method. For each subdomain, a specific timestep is calculated. The size of the subdomains was set to 9 nodes.

The results were compared with the exact solution and two conventional numerical methods the finite volume and discontinuous Galerkin method. The results actually appear to be better than the conventional methods in some cases.

The Mach number and pressure contours are shown in Fig (5.5) and Fig (5.6), respectively. The method successfully captures the shock and agrees with the numerical methods.

The finite volume model using a Steger Warming scheme Fig. (5.8) and a second order discontinuous galerkin, Fig. (5.9), were run with a fine grid to obtain accurate results. The number of nodes for the FV case was 9456 and for DG was 4544. Fig (5.10) shows the pressure

value along the bottom wall. The location of the shock agrees with the exact solution, along with the location of the reflected wave.

5.2. Effect of parameters

5.2.1. Shape parameter

We also studied the effect of the shape parameter on the accuracy of the solution. A high shape parameter increases the condition number and consequently leads to ill-conditioning of the problem. Here we used two shape parameters to calculate the domain variables at the midpoint to evaluate the fluxes. For incompressible flow, the values are generally high. Using a blended scheme, we tried to increase the accuracy in the smooth region and eliminate the dissipation caused by low value shape parameter. Using just one small shape parameter results in a low value domain variable.

5.2.2. Size of subdomain

The size of the subdomain has a great impact on the results. We compared results for different subdomains and tried to use the optimum value. The results for a subdomain of 5 was quite inaccurate. On the other hand, a subdomain of 15 was quite good but the computation time and cost were high. Therefore, we compromised with a subdomain of 9 nodes which gave good result with low computation time and cost.

5.2.3. Slope limiter

In this study we used the Van Albada limiter (Sweby, 1984) to eliminate spurious oscillations near shocks. By using the limiter, we could increase the order of accuracy in the upwinding scheme and obtain higher resolution. The rate of convergence for a first order scheme

is faster than using a second order scheme. However, by increasing the order of the scheme with the limiter, the residual does not drop the value like the first order and it oscillated around a much higher value than schemes without the limiter. Therefore, if we want to avoid oscillations near discontinuities we need to pay this unavoidable cost.

5.2.4. Time step

As mentioned in Chapter 4, a local time step was employed. The term local implies that the time were determined by the means of the subdomain, i.e., within each subdomain, a specific time step is used to calculate new domain variables. Chiu (2011) emphasizes that when one uses a uniform time step for the whole domain in a localized meshless method, the constraint on the time step usually depends upon the spectral radii of the inviscid flux Jacobian associated with the solution nodes with the smallest subdomain. Here for the each subdomain the time step was calculated by

$$\Delta t_i = CFL \frac{r_i}{\lambda_i} \quad 5.5$$

where r_i is the subdomain radius and λ_i is the maximum wave speed. This is the time step for the i^{th} reference node. Therefore, different time step, based on the size of subdomain and maximum wave speed result in saving the computation time. This method works when the limit of time accuracy is not important anymore. For the case of final time restriction, we need to use the global time step.

5.3. Postprocessing

One of the great features of meshless methods is that one can do postprocessing separately. To show results with high resolution, we can define a new system of subdomains with a high number of nodes inside the whole domain and/or subdomains. The procedure is exactly the same as the processing method while in the post processing step we introduce more nodes with just performing one iteration. Therefore the result can be interpolated on more nodes with higher resolution.

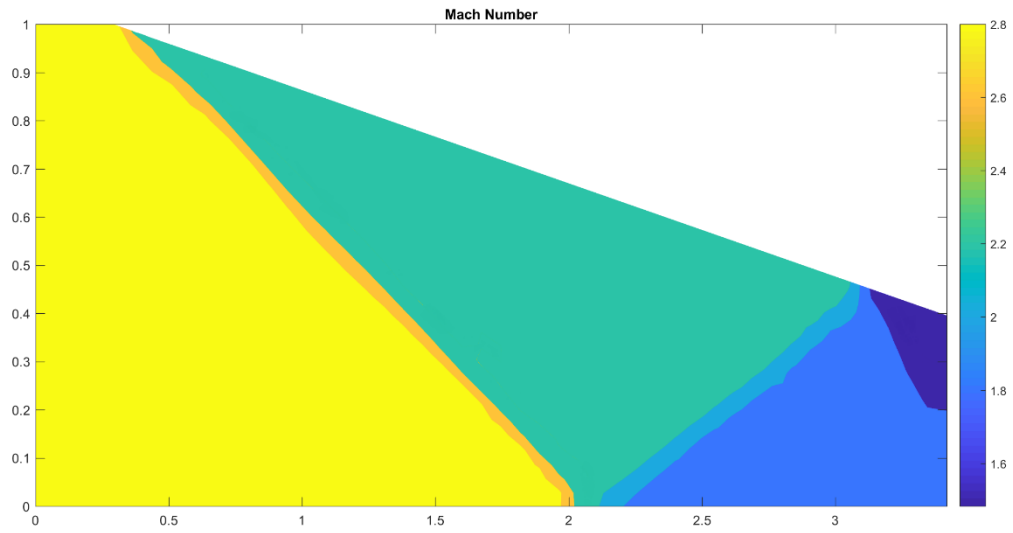


Figure 5.5 Mach number contour

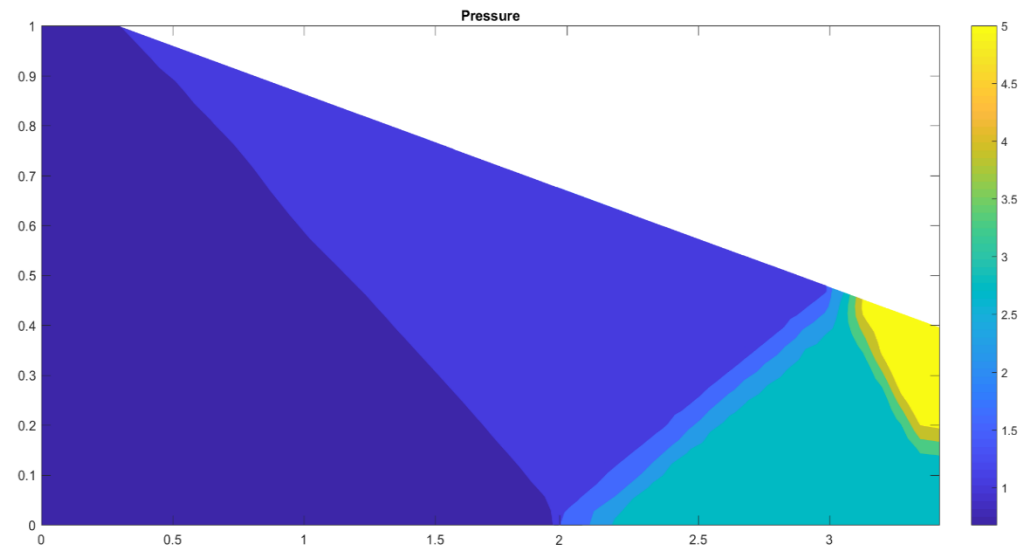


Figure 5.6 Pressure contour

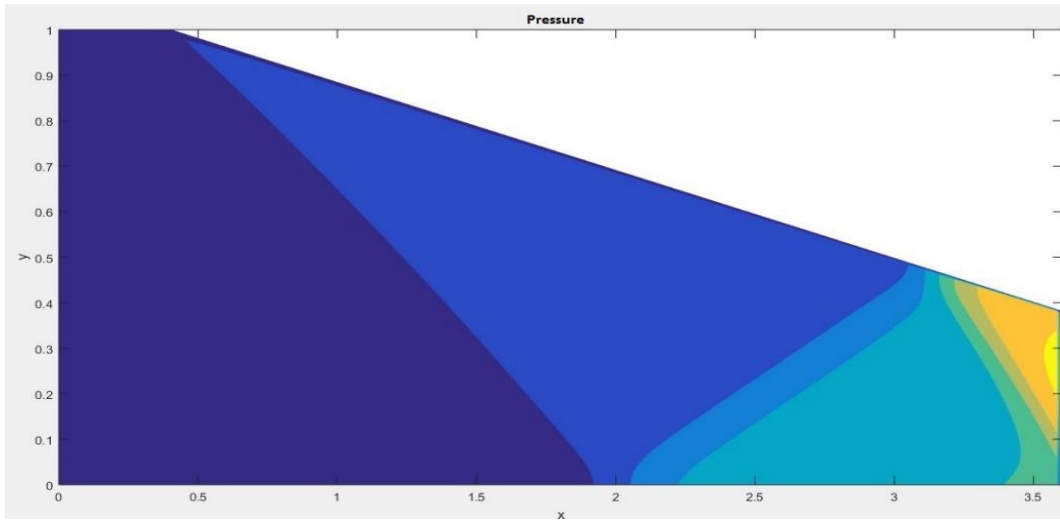


Figure 5.7 Pressure contour using Finite Volume Method

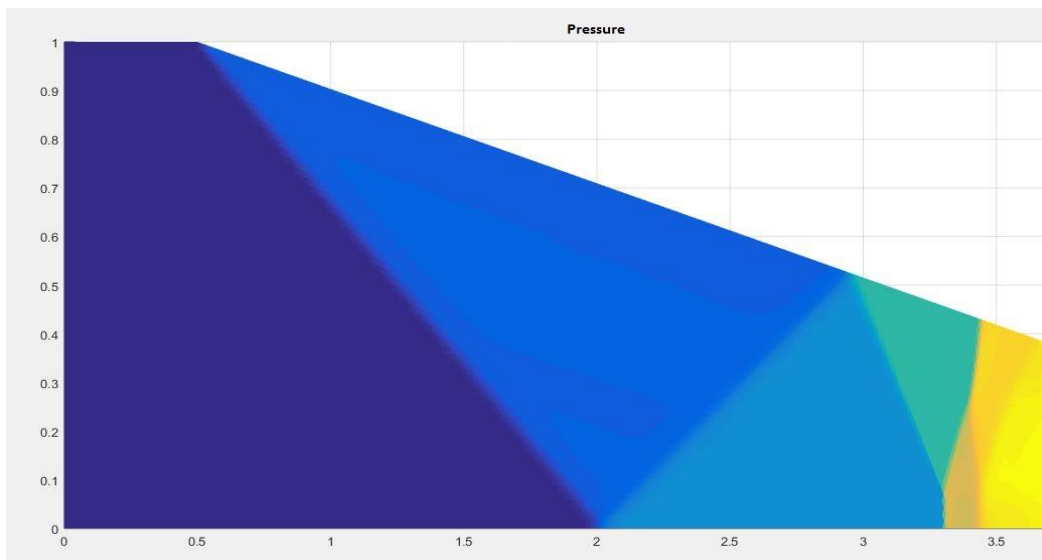


Figure 5.8 Pressure contour using DG Method

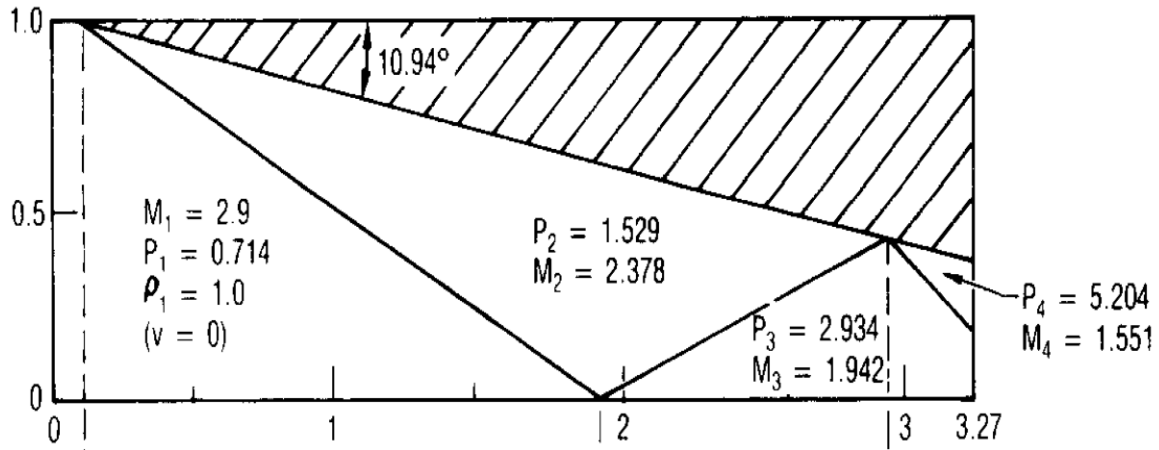


Figure 5.9 Exact solution (Wang & Widhopf, 1989)

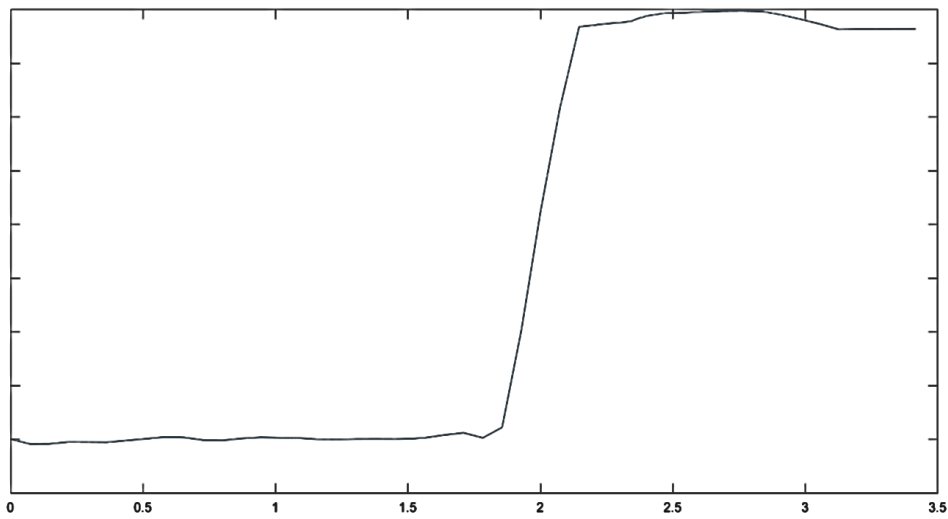


Figure 5.10 Pressure along the bottom wall

CHAPTER 6- CONCLUSION AND FUTURE WORK

6.1. Summary of work

In this study, we have presented a meshless based solver to capture shocks and discontinuities. Nonlinear convection dominant problems can produce discontinuities, even with a smooth initial condition. Numerical schemes need to be able to distinguish the direction of wave propagation to capture shocks. Since there is no connectivity among the nodes in a meshless method, the scheme must be able to identify upstream and downstream flows.

The numerical scheme used here was based on RBF-DQ to approximate the function derivatives and an upwinding scheme added to the system to capture the shock wave propagation. The localized RBF-DQ method is very sensitive to the shape parameter, size of the subdomain and the number of nodes distributed within the domain.

The model was first tested using three incompressible fluid flow benchmarks i.e., the Cavity flow problem, Natural convection, and forced convection over a backward facing step. The result were in a good agreement with previous studies. After assuring that the scheme and based algorithm were effective and accurate, we employed it on an Euler equation problem with the presence of oblique shocks within the domain.

In order to increase the resolution, a second order Roe's scheme with a Van Albada limiter was applied to calculate the fluxes at the mid-points. The scheme was then used to simulate the supersonic flow within a scramjet configuration and the effect of involved parameters on the solution were examined. Results of the present scheme were compared with

finite volume and discontinuous Galerkin techniques, and was found to be robust and accurate in capturing shocks location.

6.2. Future work

Meshless methods and especially RBFs, are particularly promising an alternative for solving fluid flow. Recently many research studies have focused on the meshless method. Since this method is more recent than other methods, there are still issues that need more investigation. Studying the effect of the shape parameter is an important issue. This constant varies case by case and there is no theoretical approach yet to identify the optimum value for each problem. A bad value for the shape parameter can lead to inappropriate results. Parallel processing is the other potential feature of the meshless method that is attractive for the future. Since the domain is typically divided into a number of smaller subdomains, we can distribute operation among the processors to speed up the algorithm and save the computation time. Nodes distribution techniques is another important parameter. Here one might implement several statistical methods to find the best way to establish node distribution based on the problem difficulties.

Adaptation techniques can also be added to the meshless method to improve accuracy. Since mesh connectivity is no longer existent here, the process of adding or removing a node is very comfortable and easy.

APPENDIX: NOMENCLATURE

<p>A Jacobian matrix</p> <p>B 0; PrRaT</p> <p>c Shape parameter</p> <p>C_{vis} Pr; 1/Re</p> <p>C_t 1; 1/Pr</p> <p>d_i The distance between the ith data point and its nearest neighbor</p> <p>E_i Fluxes at the mid-point</p> <p>E Total energy</p> <p>e Internal energy</p> <p>F Fluxes in x direction</p> <p>g Gravity</p> <p>G Fluxes in y direction</p> <p>H Enthalpy</p> <p>L Reference length</p> <p>N_I Internal node</p> <p>N all collocation points in the domain</p> <p>p dimensional pressure</p> <p>Pe Peclet number $Pe = \frac{VL}{\alpha}$</p> <p>Pr Prandtl number $Pr = \frac{\nu}{\alpha}$</p> <p>P_j point</p> <p>r_i Radial dimension</p> <p>Re Reynolds number $Re = \frac{\rho VL}{\mu}$</p> <p>Ra Rayleigh number $Ra = \frac{g\beta(T_h - T_c)L^3}{\alpha\nu}$</p> <p>s Van Albada limiter</p> <p>t Time</p> <p>\hat{t} Dimensional time</p>	<p>T Temperature</p> <p>\hat{T} Dimensional temperature</p> <p>T_c Cold (or reference) temperature</p> <p>T_h Hot temperature (heated wall)</p> <p>U Conservative variables</p> <p>u x component of velocity</p> <p>V^n Velocity at n^{th} time level</p> <p>V^* Intermediate velocity</p> <p>\hat{V} Dimensional velocity vector</p> <p>v y component of velocity</p> <p>$w^{1,x}$ RBF-DQ for first x derivative</p> <p>$w^{1,y}$ RBF-DQ for first y derivative</p> <p>γ Gas constant</p> <p>λ_i Maximum wave speed</p> <p>α Thermal diffusivity</p> <p>B Coefficient of thermal expansion</p> <p>ρ Density</p> <p>μ Dynamic viscosity</p> <p>ν Kinematic viscosity</p> <p>ϕ Radial basis function</p> <p>Ω Entire domain</p> <p>Ω_j Local domain of influence</p> <p>∇ Del operator</p> <p>∇p Pressure gradient</p> <p>Δt Time step</p> <p>Δt_i Local time step</p>
---	---

REFERENCES

- Afiatdoust, F., & Esmailbeigi, M. (2015). Optimal variable shape parameters using genetic algorithm for radial basis function approximation. *Ain Shams Engineering Journal*, 6(2), 639–647.
- Armaly, B., Durst, F., Pereira, J., & Schonung, B. (1983). Experimental and theoretical investigation of backward-facing step flow. *Journal of Fluid Mechanics*, 127, 473–496. <https://doi.org/10.1017/S0022112083002839>
- Atluri, S. N., & Zhu, T. (1998). A new Meshless Local Petrov-Galerkin (MLPG) approach in computational mechanics. *Computational Mechanics*, 22(2), 117–127. <https://doi.org/10.1007/s004660050346>
- Atluri, S. N., & Zhu, T. (2000). New concepts in meshless methods. *International Journal for Numerical Methods in Engineering*. [https://doi.org/10.1002/\(SICI\)1097-0207\(20000110/30\)47:1/3<537::AID-NME783>3.0.CO;2-E](https://doi.org/10.1002/(SICI)1097-0207(20000110/30)47:1/3<537::AID-NME783>3.0.CO;2-E)
- Batina, J. T. (1993). A gridless Euler/Navier–Stokes solution algorithm for complex aircraft applications. In *31st Aerospace Sciences Meeting*. AIAA.
- Belytschko, T., Krongauz, Y., Organ, D., Fleming, M., & Krysl, P. (1996). Meshless methods: An overview and recent developments. *Computer Methods in Applied Mechanics and Engineering*, 139(1–4), 3–47. [https://doi.org/10.1016/S0045-7825\(96\)01078-X](https://doi.org/10.1016/S0045-7825(96)01078-X)
- Belytschko, T., Lu, Y. Y., & Gu, L. (1994). Element-free Galerkin methods. *International Journal for Numerical Methods in Engineering*, 37(2), 229–256. <https://doi.org/10.1002/nme.1620370205>
- Chorin, A. J. (1968). Numerical Solution of the Navier-Stokes Equations. *Mathematics of Computation*, 22(104), 745. <https://doi.org/10.2307/2004575>
- Chorin, A. J., & Marsden, J. E. (1993). *A Mathematical Introduction to Fluid Mechanics. Performance Evaluation* (Vol. 4). <https://doi.org/10.1007/978-1-4612-0883-9>
- Duarte, C. A., & Oden, J. T. (1996). An h-p adaptive method using clouds. *Computer Methods in Applied Mechanics and Engineering*, 139(1–4), 237–262. [https://doi.org/10.1016/S0045-7825\(96\)01085-7](https://doi.org/10.1016/S0045-7825(96)01085-7)
- Franke, C., & Schaback, R. (1998). Solving partial differential equations by collocation using radial basis functions. *Applied Mathematics and Computation*, 93, 73–82. [https://doi.org/10.1016/S0096-3003\(97\)10104-7](https://doi.org/10.1016/S0096-3003(97)10104-7)
- Franke, R. (1982). Scattered data interpolation: tests of some methods. *Mathematics of Computation*, 38(157), 181–181. <https://doi.org/10.1090/S0025-5718-1982-0637296-4>

- Gingold, R. a., & Monaghan, J. J. (1977). Smoothed particle hydrodynamics-theory and application to non-spherical stars. *Monthly Notices of the Royal Astronomical Society*, 181, 375–389. <https://doi.org/10.1093/mnras/181.3.375>
- Godunov, S. K. (1959). A finite-difference method for the numerical computation of discontinuous solutions of the equations of fluid dynamics. *Matematicheskii Sbornik*, 47, 271–306.
- Hardy, R. L. (1971). Multiquadric equations of topography and other irregular surfaces. *Journal of Geophysical Research*, 76(8), 1905–1915. <https://doi.org/10.1029/JB076i008p01905>
- Harris, M., Kassab, A., & Divo, E. (2017). An Rbf Interpolation Blending Scheme For Effective Shock-capturing. *International Journal of Computational Methods and Experimental Measurements*, 5(3), 281–292.
- Hou, T. Y., & Floch, P. G. Le. (1994). Why Nonconservative Schemes Converge to Wrong Solutions: Error Analysis. *Mathematics of Computation*, 62(206), 497. <https://doi.org/10.2307/2153520>
- Kansa, E. J. (1990). Multiquadrics-A scattered data approximation scheme with applications to computational fluid-dynamics-II solutions to parabolic, hyperbolic and elliptic partial differential equations. *Computers and Mathematics with Applications*, 19(8–9), 147–161. [https://doi.org/10.1016/0898-1221\(90\)90271-K](https://doi.org/10.1016/0898-1221(90)90271-K)
- Kwan Yu Chiu. (2011). *A conservative meshless framework for conservation laws with applications in computational fluid dynamics*.
- Larsson, E., & Fornberg, B. (2003). A numerical study of some radial basis function based solution methods for elliptic PDEs. *Computers & Mathematics with Applications*, 46(5–6), 891–902. [https://doi.org/10.1016/S0898-1221\(03\)90151-9](https://doi.org/10.1016/S0898-1221(03)90151-9)
- Lax, P., & Wendroff, B. (1960). Systems of conservation laws. *Communications on Pure and Applied Mathematics*, 13(2), 217–237. <https://doi.org/10.1002/cpa.3160130205>
- LeVeque, R. J. (1992). *Numerical Methods for Conservation Laws*. *Mathematics of Computation* (Vol. 57). <https://doi.org/10.2307/2938728>
- Ling, L., & Kansa, E. J. (2005). A least-squares preconditioner for radial basis functions collocation methods. *Advances in Computational Mathematics*, 23(1–2), 31–54. <https://doi.org/10.1007/s10444-004-1809-5>
- Liszka, T. J., Duarte, C. A. M., & Tworzydło, W. W. (1996). hp-Meshless cloud method. *Computer Methods in Applied Mechanics and Engineering*, 139(1–4), 263–288. [https://doi.org/10.1016/S0045-7825\(96\)01086-9](https://doi.org/10.1016/S0045-7825(96)01086-9)
- Liu, G. R. (2010). *Meshfree Methods: Moving Beyond the Finite Element Method*. *Meshfree Methods*. <https://doi.org/10.1115/1.1553432>

- Liu, W. K., Jun, S., Li, S., Adee, J., & Belytschko, T. (1995). Reproducing kernel particle methods for structural dynamics. *International Journal for Numerical Methods in Engineering*, 38(10), 1655–1679. <https://doi.org/10.1002/nme.1620381005>
- Lu, Y. Y., Belytschko, T., & Gu, L. (1994). A new implementation of the element free Galerkin method. *Computer Methods in Applied Mechanics and Engineering*, 113(3–4), 397–414. [https://doi.org/10.1016/0045-7825\(94\)90056-6](https://doi.org/10.1016/0045-7825(94)90056-6)
- Lucy, L. B. (1977). A numerical approach to the testing of the fission hypothesis. *The Astronomical Journal*, 82, 1013. <https://doi.org/10.1086/112164>
- Mai-Duy, N., & Tran-Cong, T. (2002). Mesh-free radial basis function network methods with domain decomposition for approximation of functions and numerical solution of Poisson's equations. *Engineering Analysis with Boundary Elements*, 26(2), 133–156. [https://doi.org/10.1016/S0955-7997\(01\)00092-3](https://doi.org/10.1016/S0955-7997(01)00092-3)
- Monaghan, J. J. (1988). An introduction to SPH. *Computer Physics Communications*, 48(1), 89–96. [https://doi.org/10.1016/0010-4655\(88\)90026-4](https://doi.org/10.1016/0010-4655(88)90026-4)
- Morinishi, K. (1995). A gridless type solution for high Reynolds number multielement flow fields. In *AIAA Paper*.
- Mukherjee, Y. X., & Mukherjee, S. (1997). The boundary node method for potential problems. *International Journal for Numerical Methods in Engineering*, 40(June 1996), 797–815. [https://doi.org/10.1002/\(SICI\)1097-0207\(19970315\)40:5<797::AID-NME89>3.0.CO;2-#](https://doi.org/10.1002/(SICI)1097-0207(19970315)40:5<797::AID-NME89>3.0.CO;2-#)
- Nayroles, B., Touzot, G., & Villon, P. (1992). Generalizing the finite element method: Diffuse approximation and diffuse elements. *Computational Mechanics*, 10(5), 307–318. <https://doi.org/10.1007/BF00364252>
- Noguchi, H., Kawashima, T., & Miyamura, T. (2000). Element free analyses of shell and spatial structures. *International Journal for Numerical Methods in Engineering*, 47(6), 1215–1240. [https://doi.org/10.1002/\(SICI\)1097-0207\(20000228\)47:6<1215::AID-NME834>3.0.CO;2-M](https://doi.org/10.1002/(SICI)1097-0207(20000228)47:6<1215::AID-NME834>3.0.CO;2-M)
- Onate, E., Idelsohn, S., Zienkiewicz, O. C., Taylor, R. L., & Sacco, C. (1996). A stabilized finite point method for analysis of fluid mechanics problems. *Computer Methods in Applied Mechanics and Engineering*, 139(1–4), 315–346. [https://doi.org/10.1016/S0045-7825\(96\)01088-2](https://doi.org/10.1016/S0045-7825(96)01088-2)
- Randles, P. W., & Libersky, L. D. (1996). Smoothed Particle Hydrodynamics: Some recent improvements and applications. *Computer Methods in Applied Mechanics and Engineering*, 139(1–4), 375–408. [https://doi.org/10.1016/S0045-7825\(96\)01090-0](https://doi.org/10.1016/S0045-7825(96)01090-0)
- Roe, P. L. (1981). Approximate Riemann solvers, parameter vectors, and difference schemes. *Journal of computational physics. Journal of Computational Physics*, 43(2), 357–372.

- Sarler, B., & Vertnik, R. (2006). Meshfree explicit local radial basis function collocation method for diffusion problems. *Computers and Mathematics with Applications*, 51(8 SPEC. ISS.), 1269–1282. <https://doi.org/10.1016/j.camwa.2006.04.013>
- Shu, C., Ding, H., Chen, H. Q., & Wang, T. G. (2005). An upwind local RBF-DQ method for simulation of inviscid compressible flows. *Computer Methods in Applied Mechanics and Engineering*, 194(18–20), 2001–2017. <https://doi.org/10.1016/j.cma.2004.07.008>
- Shu, C., Ding, H., & Yeo, K. . (2003). Local radial basis function-based differential quadrature method and its application to solve two-dimensional incompressible Navier–Stokes equations. *Computer Methods in Applied Mechanics and Engineering*, 192(7–8), 941–954. [https://doi.org/10.1016/S0045-7825\(02\)00618-7](https://doi.org/10.1016/S0045-7825(02)00618-7)
- Sweby, P. K. (1984). High Resolution Schemes Using Flux Limiters for Hyperbolic Conservation Laws. *SIAM Journal on Numerical Analysis*, 21(5), 995–1011. <https://doi.org/10.1137/0721062>
- Temam, R. (2000). *Navier-Stokes Equations: Theory and Numerical Analysis*.
- Tota, P. V. (2006). *Meshless Euler solver using radial basis functions for solving inviscid compressible flows*.
- Wang, J. C. T., & Widhopf, G. F. (1989). A high-resolution tvd finite volume scheme for the euler equations in conservation form. *Journal of Computational Physics*, 84(1), 145–173. [https://doi.org/10.1016/0021-9991\(89\)90186-1](https://doi.org/10.1016/0021-9991(89)90186-1)
- Wang, J. G., & Liu, G. R. (2002). On the optimal shape parameters of radial basis functions used for 2-D meshless methods. *Computer Methods in Applied Mechanics and Engineering*, 191(23–24), 2611–2630. [https://doi.org/10.1016/S0045-7825\(01\)00419-4](https://doi.org/10.1016/S0045-7825(01)00419-4)
- Waters, J., & Pepper, D. W. (2015). Global versus localized RBF meshless methods for solving incompressible fluid flow with heat transfer. *Numerical Heat Transfer, Part B: Fundamentals*, 68(3), 185–203.
- Wu, Y. L. & Shu, C. (2002). Development of RBF-DQ method for derivative approximation and its application to simulate natural convection in concentric annuli. *Computational Mechanics*, 29(6), 477–485.
- Yao, G., Siraj-Ul-Islam, & Sarler, B. (2012). Assessment of global and local meshless methods based on collocation with radial basis functions for parabolic partial differential equations in three dimensions. *Engineering Analysis with Boundary Elements*, 36(11). <https://doi.org/10.1016/j.enganabound.2012.04.012>
- Zhou, X., Hon, Y. C., & Li, J. (2003). Overlapping domain decomposition method by radial basis functions. *Applied Numerical Mathematics*, 44(1–2), 241–255. [https://doi.org/10.1016/S0168-9274\(02\)00107-1](https://doi.org/10.1016/S0168-9274(02)00107-1)

CURRICULUM VITAE

Graduate College
University of Nevada, Las Vegas

Ebrahim Nabizadeh Shahrebabak

Degrees:

Bachelor of Science in Chemical Engineering 2014

Isfahan University of Technology

Isfahan, Iran

Thesis topic: CFD Simulation of Flow Boiling With OpenFOAM

Master of Science, Mechanical Engineering 2016

University of Nevada, Las Vegas

Las Vegas, Nevada

Thesis topic: Meshless Method for Simulate of Compressible Flow

Honor Membership:

Tau Beta Pi, The Engineering Society, 2016- Present

Publications:

E. Nabizadeh, D. W. Pepper, "A Localized RBF-MQ Meshless Method for Simulating Compressible Flows", American Society of Thermal and Fluids Engineers (ASTFE) Conference, April 2017

E. Nabizadeh, D. W. Pepper, and J. Waters, "Simulating Hydraulic Fracturing and Contaminant Transport with MATLAB and COMSOL Multiphysics Software". COMSOL conference Boston, Oct. 2016

E Nabizadeh Shahrebabak, S Golafshan, MR Chamani, "Simulation of Air Flow through Skylight in Historical House, Case Study: Historic House of Mosavar-al-Molki". International Journal of Architectural Engineering and Urban Planning, Jun. 2016

E Nabizadeh Shahrebabak, MR Chamani, S Golafshan, "Thermal Behavior of Skylight in Historical House, Case Study: Historic House of Mosavar-al-Molki". (under review)

Thesis Examination Committee:

Chairperson, Dr. Darrell W. Pepper, Ph.D.

Committee Member, Dr. Hui Zhao, Ph.D.

Committee Member, Dr. William Culbreth, Ph.D.

Graduate College Faculty Representative, Dr. Laxmi Gewali, Ph.D.