

Proceedings of the 12th International Conference on
Computational Fluid Dynamics in the Oil & Gas,
Metallurgical and Process Industries

Progress in Applied CFD – CFD2017



SINTEF Proceedings

Editors:

Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Proceedings of the 12th International Conference on Computational Fluid Dynamics
in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 2

Editors: Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Selected papers from 10th International Conference on Computational Fluid
Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:

CFD, Flow, Modelling

Cover, illustration: Arun Kamath

ISSN 2387-4295 (online)

ISBN 978-82-536-1544-8 (pdf)

© Copyright SINTEF Academic Press 2017

The material in this publication is covered by the provisions of the Norwegian Copyright Act. Without any special agreement with SINTEF Academic Press, any copying and making available of the material is only allowed to the extent that this is permitted by law or allowed through an agreement with Kopinor, the Reproduction Rights Organisation for Norway. Any use contrary to legislation or an agreement may lead to a liability for damages and confiscation, and may be punished by fines or imprisonment

SINTEF Academic Press

Address: Forskningsveien 3 B
 PO Box 124 Blindern
 N-0314 OSLO

Tel: +47 73 59 30 00

Fax: +47 22 96 55 08

www.sintef.no/byggforsk

www.sintefbok.no

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains all manuscripts approved by the reviewers and the organizing committee of the 12th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in May/June 2017 and is also known as CFD2017 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focuses on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. In addition pragmatic modelling concepts and bio-mechanical applications have become an important part of the conference. The papers in this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are included in the proceedings. 108 contributions were presented at the conference together with six keynote presentations. A majority of these contributions are presented by their manuscript in this collection (a few were granted to present without an accompanying manuscript).

The organizing committee would like to thank everyone who has helped with review of manuscripts, all those who helped to promote the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: ANSYS, SFI Metal Production and NanoSim.

Stein Tore Johansen & Jan Erik Olsen



Organizing committee:

Conference chairman: Prof. Stein Tore Johansen

Conference coordinator: Dr. Jan Erik Olsen

Dr. Bernhard Müller

Dr. Sigrid Karstad Dahl

Dr. Shahriar Amini

Dr. Ernst Meese

Dr. Josip Zoric

Dr. Jannike Solsvik

Dr. Peter Witt

Scientific committee:

Stein Tore Johansen, SINTEF/NTNU

Bernhard Müller, NTNU

Phil Schwarz, CSIRO

Akio Tomiyama, Kobe University

Hans Kuipers, Eindhoven University of Technology

Jinghai Li, Chinese Academy of Science

Markus Braun, Ansys

Simon Lo, CD-adapco

Patrick Segers, Universiteit Gent

Jiyuan Tu, RMIT

Jos Derksen, University of Aberdeen

Dmitry Eskin, Schlumberger-Doll Research

Pär Jönsson, KTH

Stefan Pirker, Johannes Kepler University

Josip Zoric, SINTEF

CONTENTS

PRAGMATIC MODELLING	9
On pragmatism in industrial modeling. Part III: Application to operational drilling	11
CFD modeling of dynamic emulsion stability	23
Modelling of interaction between turbines and terrain wakes using pragmatic approach	29
FLUIDIZED BED	37
Simulation of chemical looping combustion process in a double looping fluidized bed reactor with cu-based oxygen carriers.....	39
Extremely fast simulations of heat transfer in fluidized beds.....	47
Mass transfer phenomena in fluidized beds with horizontally immersed membranes	53
A Two-Fluid model study of hydrogen production via water gas shift in fluidized bed membrane reactors	63
Effect of lift force on dense gas-fluidized beds of non-spherical particles	71
Experimental and numerical investigation of a bubbling dense gas-solid fluidized bed	81
Direct numerical simulation of the effective drag in gas-liquid-solid systems	89
A Lagrangian-Eulerian hybrid model for the simulation of direct reduction of iron ore in fluidized beds.....	97
High temperature fluidization - influence of inter-particle forces on fluidization behavior	107
Verification of filtered two fluid models for reactive gas-solid flows	115
BIOMECHANICS.....	123
A computational framework involving CFD and data mining tools for analyzing disease in carotid artery	125
Investigating the numerical parameter space for a stenosed patient-specific internal carotid artery model.....	133
Velocity profiles in a 2D model of the left ventricular outflow tract, pathological case study using PIV and CFD modeling.....	139
Oscillatory flow and mass transport in a coronary artery.....	147
Patient specific numerical simulation of flow in the human upper airways for assessing the effect of nasal surgery.....	153
CFD simulations of turbulent flow in the human upper airways	163
OIL & GAS APPLICATIONS	169
Estimation of flow rates and parameters in two-phase stratified and slug flow by an ensemble Kalman filter	171
Direct numerical simulation of proppant transport in a narrow channel for hydraulic fracturing application	179
Multiphase direct numerical simulations (DNS) of oil-water flows through homogeneous porous rocks	185
CFD erosion modelling of blind tees	191
Shape factors inclusion in a one-dimensional, transient two-fluid model for stratified and slug flow simulations in pipes	201
Gas-liquid two-phase flow behavior in terrain-inclined pipelines for wet natural gas transportation	207

NUMERICS, METHODS & CODE DEVELOPMENT	213
Innovative computing for industrially-relevant multiphase flows	215
Development of GPU parallel multiphase flow solver for turbulent slurry flows in cyclone.....	223
Immersed boundary method for the compressible Navier–Stokes equations using high order summation-by-parts difference operators	233
Direct numerical simulation of coupled heat and mass transfer in fluid-solid systems	243
A simulation concept for generic simulation of multi-material flow, using staggered Cartesian grids.....	253
A cartesian cut-cell method, based on formal volume averaging of mass, momentum equations.....	265
SOFT: a framework for semantic interoperability of scientific software	273
 POPULATION BALANCE	 279
Combined multifluid-population balance method for polydisperse multiphase flows	281
A multifluid-PBE model for a slurry bubble column with bubble size dependent velocity, weight fractions and temperature.....	285
CFD simulation of the droplet size distribution of liquid-liquid emulsions in stirred tank reactors	295
Towards a CFD model for boiling flows: validation of QMOM predictions with TOPFLOW experiments	301
Numerical simulations of turbulent liquid-liquid dispersions with quadrature-based moment methods.....	309
Simulation of dispersion of immiscible fluids in a turbulent couette flow	317
Simulation of gas-liquid flows in separators - a Lagrangian approach.....	325
CFD modelling to predict mass transfer in pulsed sieve plate extraction columns	335
 BREAKUP & COALESCENCE	 343
Experimental and numerical study on single droplet breakage in turbulent flow	345
Improved collision modelling for liquid metal droplets in a copper slag cleaning process	355
Modelling of bubble dynamics in slag during its hot stage engineering.....	365
Controlled coalescence with local front reconstruction method	373
 BUBBLY FLOWS	 381
Modelling of fluid dynamics, mass transfer and chemical reaction in bubbly flows	383
Stochastic DSMC model for large scale dense bubbly flows.....	391
On the surfacing mechanism of bubble plumes from subsea gas release.....	399
Bubble generated turbulence in two fluid simulation of bubbly flow	405
 HEAT TRANSFER	 413
CFD-simulation of boiling in a heated pipe including flow pattern transitions using a multi-field concept	415
The pear-shaped fate of an ice melting front	423
Flow dynamics studies for flexible operation of continuous casters (flow flex cc).....	431
An Euler-Euler model for gas-liquid flows in a coil wound heat exchanger.....	441
 NON-NEWTONIAN FLOWS.....	 449
Viscoelastic flow simulations in disordered porous media	451
Tire rubber extrudate swell simulation and verification with experiments	459
Front-tracking simulations of bubbles rising in non-Newtonian fluids.....	469
A 2D sediment bed morphodynamics model for turbulent, non-Newtonian, particle-loaded flows.....	479

METALLURGICAL APPLICATIONS.....	491
Experimental modelling of metallurgical processes	493
State of the art: macroscopic modelling approaches for the description of multiphysics phenomena within the electroslag remelting process	499
LES-VOF simulation of turbulent interfacial flow in the continuous casting mold	507
CFD-DEM modelling of blast furnace tapping	515
Multiphase flow modelling of furnace tapholes	521
Numerical predictions of the shape and size of the raceway zone in a blast furnace.....	531
Modelling and measurements in the aluminium industry - Where are the obstacles?	541
Modelling of chemical reactions in metallurgical processes.....	549
Using CFD analysis to optimise top submerged lance furnace geometries	555
Numerical analysis of the temperature distribution in a martensic stainless steel strip during hardening.....	565
Validation of a rapid slag viscosity measurement by CFD.....	575
Solidification modeling with user defined function in ANSYS Fluent.....	583
Cleaning of polycyclic aromatic hydrocarbons (PAH) obtained from ferroalloys plant.....	587
Granular flow described by fictitious fluids: a suitable methodology for process simulations	593
A multiscale numerical approach of the dripping slag in the coke bed zone of a pilot scale Si-Mn furnace.....	599
 INDUSTRIAL APPLICATIONS	 605
Use of CFD as a design tool for a phosphoric acid plant cooling pond	607
Numerical evaluation of co-firing solid recovered fuel with petroleum coke in a cement rotary kiln: Influence of fuel moisture	613
Experimental and CFD investigation of fractal distributor on a novel plate and frame ion-exchanger	621
 COMBUSTION	 631
CFD modeling of a commercial-size circle-draft biomass gasifier.....	633
Numerical study of coal particle gasification up to Reynolds numbers of 1000.....	641
Modelling combustion of pulverized coal and alternative carbon materials in the blast furnace raceway	647
Combustion chamber scaling for energy recovery from furnace process gas: waste to value	657
 PACKED BED.....	 665
Comparison of particle-resolved direct numerical simulation and 1D modelling of catalytic reactions in a packed bed	667
Numerical investigation of particle types influence on packed bed adsorber behaviour	675
CFD based study of dense medium drum separation processes	683
A multi-domain 1D particle-reactor model for packed bed reactor applications.....	689
 SPECIES TRANSPORT & INTERFACES	 699
Modelling and numerical simulation of surface active species transport - reaction in welding processes	701
Multiscale approach to fully resolved boundary layers using adaptive grids.....	709
Implementation, demonstration and validation of a user-defined wall function for direct precipitation fouling in Ansys Fluent.....	717

FREE SURFACE FLOW & WAVES	727
Unresolved CFD-DEM in environmental engineering: submarine slope stability and other applications.....	729
Influence of the upstream cylinder and wave breaking point on the breaking wave forces on the downstream cylinder	735
Recent developments for the computation of the necessary submergence of pump intakes with free surfaces	743
Parallel multiphase flow software for solving the Navier-Stokes equations	752
 PARTICLE METHODS	 759
A numerical approach to model aggregate restructuring in shear flow using DEM in Lattice-Boltzmann simulations	761
Adaptive coarse-graining for large-scale DEM simulations.....	773
Novel efficient hybrid-DEM collision integration scheme.....	779
Implementing the kinetic theory of granular flows into the Lagrangian dense discrete phase model.....	785
Importance of the different fluid forces on particle dispersion in fluid phase resonance mixers	791
Large scale modelling of bubble formation and growth in a supersaturated liquid.....	798
 FUNDAMENTAL FLUID DYNAMICS	 807
Flow past a yawed cylinder of finite length using a fictitious domain method	809
A numerical evaluation of the effect of the electro-magnetic force on bubble flow in aluminium smelting process.....	819
A DNS study of droplet spreading and penetration on a porous medium.....	825
From linear to nonlinear: Transient growth in confined magnetohydrodynamic flows.....	831

IMMERSED BOUNDARY METHOD FOR THE COMPRESSIBLE NAVIER–STOKES EQUATIONS USING HIGH ORDER SUMMATION-BY-PARTS DIFFERENCE OPERATORS

M. EHSAN KHALILI^{1*}, MARTIN LARSSON^{2†}, BERNHARD MÜLLER^{1‡}

¹Department of Energy and Process Engineering, Norwegian University of Science and Technology (NTNU), Kolbjørn Hejes vei 2, NO-7491 Trondheim, Norway

²Sportradar AS, Ferjemannsveien 10, NO-7014 Trondheim, Norway

* E-mail: mohammadtaghi.khalili@ntnu.no

† E-mail: m.larsson@sportradar.com

‡ E-mail: bernhard.muller@ntnu.no

ABSTRACT

A ghost-point immersed boundary method is devised for the compressible Navier–Stokes equations by employing high order summation-by-parts (SBP) difference operators. The immersed boundaries are treated as sharp interfaces by enforcing the solid wall boundary conditions via flow variables at ghost points using bilinearly interpolated flow variables at mirror points. The approach is verified and validated for compressible flow past a circular cylinder at moderate Reynolds numbers.

Keywords: High order finite difference method, Immersed boundary method, Compressible viscous flow .

NOMENCLATURE

Greek Symbols

ρ	Mass density
μ	Dynamic viscosity
τ	Viscous stress tensor
γ	Ratio of specific heats
κ	Heat conduction coefficient
ϕ	Generic variable
\mathcal{G}	Set of body intercept points that are part of the interpolation stencil
Λ	Parameter
σ	Source term in steady state heat equation
ξ, η	Transformed coordinates in computational domain

Latin Symbols

t	Physical time
E	Specific total energy
H	Total enthalpy
p	Pressure
S_c	Sutherland constant
T	Temperature
\mathbf{U}	Vector of conservative variables
\mathbf{V}	Vandermonde matrix
x, y	Cartesian coordinates in physical domain
J^{-1}	Jacobian determinant

Sub/superscripts

GP	Ghost point
IP	Image point
BI	Body intercept
c'	Conservative perturbation
v'	Viscous perturbation

INTRODUCTION

Many applications in engineering, biology and medicine involve low and moderate Reynolds number flow problems with complex boundaries between fluid and structure. Simulating these problems with conventional methods requires the process of generating high quality and body-conforming grids which is challenging and time-consuming. Recently, there has been a growing interest in the development of non-boundary conforming methodologies for the solution of the Navier–Stokes equations (Mittal and Iaccarino, 2005). In such methods, the requirement that the grid should conform to a solid boundary is dropped, and the effect of the immersed boundary of the solid body on the flow is introduced through the proper treatment of the solution variables near the boundary. The basic advantage of these formulations is the simplicity compared with conventional body-conforming grid generation, especially in cases of complex stationary or moving boundaries where the demand for regeneration or deformation of the grid is eliminated. Therefore, efficient non boundary conforming strategies with robust Cartesian coordinate solvers can directly be applied to a wide range of flow problems.

Over the past decades a variety of non-body conforming approaches with various degrees of accuracy and complexity have been proposed. The so-called immersed-boundary method (IBM) was introduced by (Peskin, 1972). IB methods are categorized into continuous forcing and discrete (direct) forcing approaches. In the first category, a continuous forcing term is added to the governing equations to represent the interaction between the immersed boundary and the fluid, and a discrete Dirac–delta function is used to smooth this singular force on the Euler grid (Peskin, 1972; Goldstein *et al.*, 1993; Saiki and Biringen, 1996). Numerous modifications and improvements have been implemented in this category (Haeri and Shrimpton, 2012; Sotiropoulos and Yang, 2014). The second category, including the sharp interface method, mimics the presence of a surface force exerted by the boundary on the fluid by adjusting the discretization in the vicinity of the immersed boundary in order to directly take into account the boundary conditions at the IB (Ye *et al.*, 1999; Fadlun *et al.*, 2000; Mohd-Yusof, 1997; Balaras, 2004). The ghost cell immersed boundary (GCIB) method as sharp interface method is proposed in the studies by (Tseng and Ferziger, 2003; Ghias *et al.*, 2007; Mittal *et al.*, 2008). Ghost cells are defined as a layer of cells within the solid body having at least one nearby point in the fluid domain

i.e., adjoining to the immersed boundary. The flow variables at the ghost points are calculated with the boundary conditions at the immersed boundary and the flow variables at grid points near the IB in the fluid domain. The presence of the immersed boundary is introduced by the flow variables at the ghost points. The idea of image points inside the fluid domain is adopted to ensure suitable weighting coefficients in the reconstruction formula in order to avoid numerical instability caused by the large, negative weighting coefficients in the extrapolation formulation (Tseng and Ferziger, 2003). The ghost point method has shown large potential to deal with different fluid-solid interaction problems, including those involving highly complex geometries and moving or deforming bodies (Mittal *et al.*, 2008).

In the IBM, all the equations can be solved on a body non-conformal, Cartesian grid which does not require to be updated for moving or deforming bodies. Due to the flexibility of the method, many different types of IBM have been developed in incompressible and compressible flow solvers. However, most of the attention on IBM is devoted to incompressible flows (Mittal and Iaccarino, 2005). Works on viscous compressible flows are still scarce and a few IBM for viscous compressible flows has been developed (De Palma *et al.*, 2006; Ghias *et al.*, 2007; de Tullio *et al.*, 2007; Brehm *et al.*, 2015). Due to the different nature of the Navier–Stokes equations for compressible and incompressible flows, i.e. the requirement of equation of state for compressible flows, there are differences in implementation of the boundary conditions between these two types of equations as well as in the spatial discretisation schemes employed.

In this study, the ghost point IB approach has been adopted for a high order finite difference method based on summation-by-parts operators (SBP) to provide an accurate and efficient approach for studying low Mach number compressible viscous flows. The major ambition of the present work is to extend this approach for fluid structure interaction (FSI) in the upper airways to study the obstructive sleep apnea syndrome. The main focus in our study is subsonic flow which permits us to characterize the acoustic wave propagation induced by the structure oscillation in FSI to obtain a better understanding of snoring. The proposed approach is verified and validated for two dimensional flows over a circular cylinder. In the following sections, a brief review of the governing equations and their numerical solution is given. Then, the IB approach is described in detail. Finally, results are provided and compared with numerical and experimental ones available in the literature.

MODEL DESCRIPTION

Governing equations

The 2D compressible Navier–Stokes equations in perturbation form are solved. To minimize cancellation errors when discretizing the Navier–Stokes equations for compressible low Mach number flow, the perturbation formulation is employed (Sesterhenn *et al.*, 1999; Müller, 2008). The conservative form of the 2D compressible Navier–Stokes equations in perturbation formulation can be written as

$$\mathbf{U}'_t + \mathbf{F}'_x + \mathbf{G}'_y = \mathbf{F}''_x + \mathbf{G}''_y \quad (1)$$

where $\mathbf{U}' = \mathbf{U} - \mathbf{U}_0$ is the vector of conservative perturbation variables with $\mathbf{U} = (\rho, \rho u, \rho v, \rho E)^T$ the vector of the conservative variables and $\mathbf{U}_0 = (\rho_0, 0, 0, (\rho E)_0)^T$ the stagnation values.

The conservative perturbation variables \mathbf{U}' and the inviscid ($\mathbf{F}^{c'}$, $\mathbf{G}^{c'}$) and viscous perturbation flux vectors ($\mathbf{F}^{v'}$, $\mathbf{G}^{v'}$) are defined by $\mathbf{F}^{c'} = \mathbf{F}^c(\mathbf{U}) - \mathbf{F}^c(\mathbf{U}_0)$, etc.

$$\mathbf{U}' = \begin{pmatrix} \rho' \\ (\rho u)' \\ (\rho v)' \\ (\rho E)' \end{pmatrix},$$

$$\mathbf{F}^{c'} = \begin{pmatrix} (\rho u)' \\ (\rho u)'u' + p' \\ (\rho v)'u' \\ ((\rho H)_0 + (\rho H)')u' \end{pmatrix}, \mathbf{G}^{c'} = \begin{pmatrix} (\rho v)' \\ (\rho u)'v' \\ (\rho v)'v' + p' \\ ((\rho H)_0 + (\rho H)')v' \end{pmatrix},$$

$$\mathbf{F}^{v'} = \begin{pmatrix} 0 \\ \tau'_{xx} \\ \tau'_{xy} \\ u'\tau'_{xx} + v'\tau'_{xy} + \kappa T'_x \end{pmatrix}, \mathbf{G}^{v'} = \begin{pmatrix} 0 \\ \tau'_{yx} \\ \tau'_{yy} \\ u'\tau'_{yx} + v'\tau'_{yy} + \kappa T'_y \end{pmatrix},$$

where t is physical time and x and y are the Cartesian coordinates. ρ denotes density, u and v the x - and y -direction velocity components, E the specific total energy, T the temperature and κ the heat conduction coefficient calculated from the constant Prandtl number $\text{Pr} = 0.72$. ρ_0 , $(\rho E)_0$ and $(\rho H)_0$ denote the stagnation values of density, total energy density and total enthalpy density. The perturbation variables are defined as:

$$\rho' = \rho - \rho_0, \quad (\rho \mathbf{u})' = (\rho \mathbf{u}),$$

$$(\rho E)' = \rho E - (\rho E)_0, \quad (\rho H)' = (\rho E)' + p', \quad \mathbf{u}' = \frac{(\rho \mathbf{u})'}{\rho_0 + \rho'},$$

$$\tau' = \mu(\nabla \mathbf{u}' + (\nabla \mathbf{u}')^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u}')\mathbf{I}, \quad T' = \frac{p'/R - \rho'T_0}{\rho_0 + \rho'}$$

Here, R is the specific gas constant and μ is the viscosity which is determined from the Sutherland law $\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{1.5} \left[\frac{1 + S_c}{T/T_0 + S_c} \right]$ with the non-dimensional Sutherland constant $S_c = \frac{110}{301.75}$.

Since perfect gas is considered, the pressure perturbation can be related to the conservative perturbation variables by $p' = (\gamma - 1)[(\rho E)' - \frac{1}{2}((\rho \mathbf{u}') \cdot \mathbf{u}')]'$, where the ratio of specific heats $\gamma = c_p/c_v = 1.4$ for air.

The viscous flux vectors $\mathbf{F}^{v'}$ and $\mathbf{G}^{v'}$ are the same as for the standard conservative form, except for using the temperature perturbation T' instead of temperature T for the heat flux terms. The momentum density and velocity perturbations are taken as the same as their unperturbed counterparts, i.e. $(\rho \mathbf{u})' = \rho \mathbf{u}$ (Larsson and Müller, 2009). For convenience the variables are non-dimensionalized with ρ_0 , stagnation speed of sound c_0 and $\rho_0 c_0^2$ as reference values. In order to generalize the geometry for non-uniform Cartesian grids, the equations of motions are transformed from the physical domain (x, y) to the computational domain (ξ, η) by the following relations,

$$\begin{aligned} x &= x(\xi, \eta) \\ y &= y(\xi, \eta) \end{aligned} \quad (2)$$

Thus, the transformed 2D compressible Navier–Stokes equations in perturbation form are expressed as:

$$\hat{\mathbf{U}}'_t + \hat{\mathbf{F}}'_\xi + \hat{\mathbf{G}}'_\eta = 0 \quad (3)$$

where $\hat{\mathbf{U}}' = J^{-1}\mathbf{U}'$, $\hat{\mathbf{F}}' = J^{-1}(\xi_x(\mathbf{F}^{c'} - \mathbf{F}^{v'}) + \xi_y(\mathbf{G}^{c'} - \mathbf{G}^{v'}))$ and $\hat{\mathbf{G}}' = J^{-1}(\eta_x(\mathbf{F}^{c'} - \mathbf{F}^{v'}) + \eta_y(\mathbf{G}^{c'} - \mathbf{G}^{v'}))$. The chain rule for partial differentiation provides the expressions for Cartesian derivatives in the viscous flux vectors $\mathbf{F}^{v'}$ and $\mathbf{G}^{v'}$, e.g.

$u'_x = u'_\xi \xi_x + u'_\eta \eta_x$ and $u'_y = u'_\xi \xi_y + u'_\eta \eta_y$. The Jacobian determinant of the transformation is $J^{-1} = x_\xi y_\eta - x_\eta y_\xi$ and metric terms are

$$\begin{aligned} J^{-1} \xi_x &= y_\eta, & J^{-1} \xi_y &= -x_\eta, \\ J^{-1} \eta_x &= -y_\xi, & J^{-1} \eta_y &= x_\xi. \end{aligned} \quad (4)$$

Numerical methodology

The summation-by-parts (SBP) operator Q is an approximation to the first ξ - and η - derivatives in (4) and (3). In the interior, it corresponds to the standard sixth order central operator, while being third order accurate near the boundaries. Through a special boundary treatment, SBP operators permit energy estimates for discrete problems similar to those for the continuous ones that are approximated. Therefore, SBP operators can yield strictly stable schemes for general boundary conditions (Strand, 1994; Gustafsson *et al.*, 1995; Gustafsson, 2008). The global order of accuracy of the present SBP operator Q is fourth order (Müller, 2008). The energy method and the summation-by-parts operators are discussed in the Appendix A and B, respectively.

Second derivatives of viscous parts of $\hat{\mathbf{F}}'_\xi$ and $\hat{\mathbf{G}}'_\eta$ are approximated by applying the SBP operator for first derivatives twice. However, successively applying the first derivative operator makes the scheme wider, which requires special treatment for the immersed boundary method, and will be discussed in section boundary conditions below. Spurious high wave number oscillations are suppressed by a sixth order explicit filter (Visbal and Gaitonde, 2002; Müller, 2008). The classical fourth order explicit Runge–Kutta method is employed for time integration.

Immersed boundary formulation

The sharp interface method is well suited for compressible viscous flow, due to imposing the boundary conditions at immersed boundaries, without computing any forcing term and introducing any force distribution function. The ghost point immersed boundary method employed in this study is based on the ghost cell immersed boundary approach for second order methods (Ghias *et al.*, 2007; Mittal *et al.*, 2008).

The basic idea in this method is to compute the value of the flow variables at each of the ghost points (referring to the layer of points inside the solid body adjoining the immersed boundary) such that the boundary conditions at the immersed boundary are satisfied. As illustrated in Fig. 1, the procedure begins by determining the immersed boundary and then distinguishing the solid points, i.e. the nodes lying inside the solid body, and the fluid points, i.e. the nodes lying outside the body in the fluid domain. The ghost points (denoted by GP) are identified by those nodes that lie inside the body and adjacent to the immersed boundary which have at least one neighbour node in the fluid domain with the difference stencil centered at the ghost point. The image point (denoted by IP) can be found by extending a normal probe, i.e. a line normal to the immersed boundary, from the ghost point to intersect with the immersed boundary at the body intercept point (denoted by BI) such that the body intercept point lies at the midpoint of the line connecting the ghost point and the image point. Once the flow variables at the image point are computed, the ghost point variables can be determined by imposing the boundary conditions. In other words, the general strategy is to compute the flow variables at the image point by taking into account the nodal values at the surrounding fluid points and then use the boundary conditions to obtain the values at the ghost point.

Among the available options for determining the flow variables at the image points, the computationally most efficient scheme will be the bilinear interpolation scheme in 2D (Ghias *et al.*, 2007; Mittal *et al.*, 2008) where the flow variables are linearly interpolated from four nodal points surrounding the image points. This interpolation scheme leads to a nominally second order accuracy of the immersed boundary condition. The high order SBP operator used in this study for spatial discretization, corresponding to the sixth order central finite difference method at interior grids, requires three layers of ghost points inside the immersed boundary in order to maintain the overall high order of accuracy, as shown in Fig. 2.

In the case of bilinear interpolation, the interpolating polynomial involves four nodes and hence four nodal values need to be specified. The bilinear interpolation for a generic variable ϕ can be expressed as

$$\phi(x, y) = C_1 + C_2x + C_3y + C_4xy. \quad (5)$$

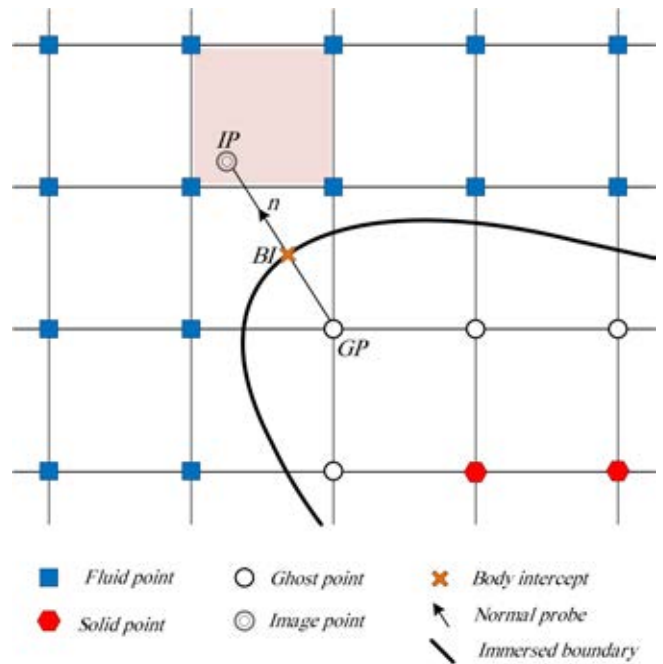


Figure 1: Schematic of points used to interpolate the variable located at a ghost point.

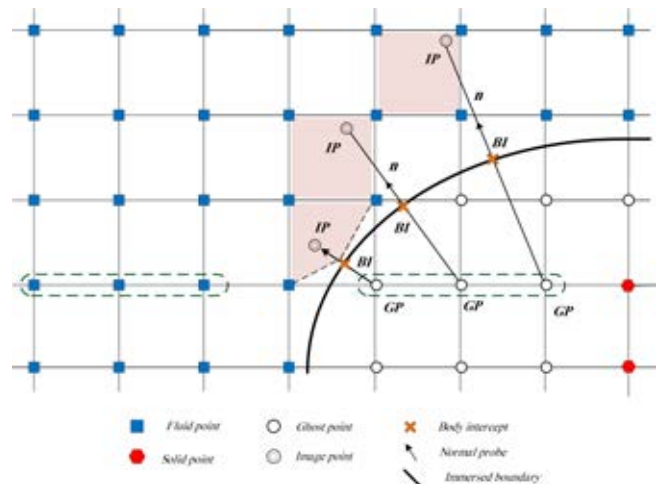


Figure 2: Schematic of 3 layers of ghost points inside immersed body on a Cartesian mesh.

The four unknown coefficients $C_i, i = 1, \dots, 4$, can be determined using values at the four nodes surrounding the image point. Thus, the variable at the image point is reconstructed through bilinear interpolation using unknown coefficients and known flow variables at surrounding fluid nodes. The four weighting coefficients are evaluated as the solution of the linear system

$$\mathbf{VC} = \phi, \quad (6)$$

where

$$\mathbf{C} = \{C_1, C_2, C_3, C_4\}^T \quad (7)$$

is the vector of the unknown coefficients and

$$\phi = \{\phi_1, \phi_2, \phi_3, \phi_4\}^T \quad (8)$$

is the vector of the four surrounding node values. The matrix \mathbf{V} is the Vandermonde matrix which is expressed as

$$\mathbf{V} = \begin{bmatrix} 1 & x_1 & y_1 & x_1 y_1 \\ 1 & x_2 & y_2 & x_2 y_2 \\ 1 & x_3 & y_3 & x_3 y_3 \\ 1 & x_4 & y_4 & x_4 y_4 \end{bmatrix} \quad (9)$$

In this classical formulation, the unknown coefficient values $C_i, i = 1, \dots, 4$, would depend on the solution at each time step. However, the approach can be reformulated such that new coefficients are only dependent on the coordinates of the image point and the geometry of the grids. The reformulation is discussed in detail in Appendix C. Thus, the image point value can be expressed as

$$\phi_{IP} = \sum_{i=1}^4 \alpha_i \phi_i \quad (10)$$

where $\alpha_i, i = 1, \dots, 4$, are coefficients depending on the coordinates only. They can be established once the grid, immersed boundary and image point coordinates are specified. When a ghost point is close to the immersed boundary, its corresponding image point might not have four surrounding fluid points. One case would be that the ghost point itself is part of the interpolation scheme. Since the ghost point value in an interpolation scheme would be unknown, the ghost point is then replaced by the body intercept point where the values are determined by the boundary conditions, cf. Fig. 3.

For Dirichlet boundary condition in this case, the corresponding row in Eq. (9) is replaced by

$$\phi_{BI}(x, y) = C_1 + C_2 x_{BI} + C_3 y_{BI} + C_4 x_{BI} y_{BI} \quad (11)$$

where x_{BI} and y_{BI} are the coordinates of the body intercept point. Thereby, for a Dirichlet boundary condition the linear system corresponding to Eq.(6) for this case becomes

$$\begin{bmatrix} 1 & x_1 & y_1 & x_1 y_1 \\ 1 & x_2 & y_2 & x_2 y_2 \\ 1 & x_3 & y_3 & x_3 y_3 \\ 1 & x_B & y_B & x_B y_B \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \end{bmatrix} = \begin{bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \phi_{BI} \end{bmatrix} \quad (12)$$

For a Neumann boundary condition, the variable gradient at the body intercept is known instead of the actual value. The most obvious choice in such a case is to use the specified gradient value $\frac{\partial \phi_{BI}}{\partial n}$ to compute the value at the image point. The gradient of ϕ_{BI} at the boundary can be determined by taking the normal derivative of Eq.(11),

$$\frac{\partial \phi_{BI}}{\partial n} = C_2 n_x + C_3 n_y + C_4 (y_{BI} n_x + x_{BI} n_y) = \zeta \quad (13)$$

where n_x and n_y are the components of the unit vector normal to the boundary.

Thus, the linear system corresponding to Eq.(6) for this case becomes

$$\begin{bmatrix} 1 & x_1 & y_1 & x_1 y_1 \\ 1 & x_2 & y_2 & x_2 y_2 \\ 1 & x_3 & y_3 & x_3 y_3 \\ 0 & n_x & n_y & y_{BI} n_x + x_{BI} n_y \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \end{bmatrix} = \begin{bmatrix} \phi_1 \\ \phi_2 \\ \phi_3 \\ \zeta \end{bmatrix} \quad (14)$$

As shown in Fig. 4, it might also be the case that two interpolation points would lie inside the immersed body, one at the corresponding ghost point itself and one at another ghost point. The procedure we used to handle this case is to repeat the above steps for the other ghost point as well, resulting in a Vandermonde matrix where another row is also replaced by Eqs. (11) or (13) in the same way as the fourth row, in contrast to (Ghias *et al.*, 2007; Mittal *et al.*, 2008). Applying our procedure for this case, it is no longer necessary to solve a coupled linear system by using iterative processes like (Ghias *et al.*, 2007; Mittal *et al.*, 2008). This situation does not pose any consistency issues and ensures that the interpolation procedure for the image point is well-posed without affecting the accuracy of the interpolation.

The value of the variable at the ghost point is computed by employing a linear approximation along the normal probe which takes into account the boundary condition at the boundary intercept. For a Dirichlet boundary condition this can generally be expressed as

$$\phi_{GP} = \frac{1}{2}(\phi_{IP} + \phi_{GP}) + O(\Delta l^2) \quad (15)$$

where Δl is the length of the normal probe from GP to IP. Solving for ϕ_{GP} using Eq. (15) and neglecting the truncation gives

$$\phi_{GP} = \left(2 - \sum_{j \in \mathcal{G}} \alpha_j\right) \phi_{BI} - \sum_{i \notin \mathcal{G}} \alpha_i \phi_i \quad (16)$$

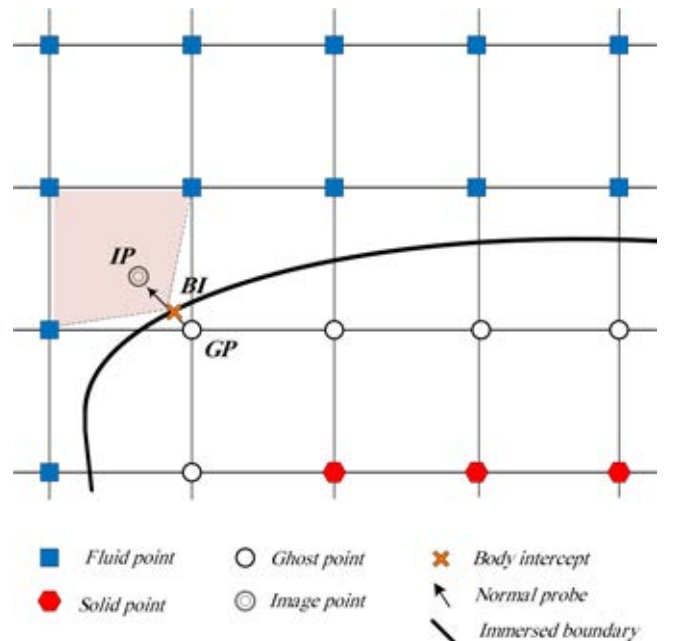


Figure 3: Schematic of the situation when one surrounding interpolation point is the boundary intercept.

where \mathcal{G} is the set of body intercepts that are part of the interpolation stencil. For a Neumann boundary condition on the immersed boundary, the following second-order central-difference is written along the normal probe

$$\left(\frac{\partial\phi}{\partial n}\right)_{BI} = \frac{\phi_{IP} - \phi_{GP}}{\Delta l} + O(\Delta l^2) \quad (17)$$

Thereby, the general formulation for a non-homogeneous Neumann boundary condition is expressed as

$$\phi_{GP} = \left(\sum_{j \in \mathcal{G}} \alpha_j - \Delta l\right) \left(\frac{\partial\phi}{\partial n}\right)_{BI} + \sum_{i \notin \mathcal{G}} \alpha_i \phi_i \quad (18)$$

Boundary conditions

The no-slip boundary condition at the immersed boundary for a stationary body is considered. Thereby, Dirichlet boundary conditions are employed for the velocity components at the IB. For each velocity components, the corresponding value at the body intercept $\phi_{BI} = 0$ is set in Eq. (11). Applying the no-slip condition at the body surface, the convective flux contribution should be zero. The pressure gradient normal to the immersed interface is set zero as a boundary layer approximation, $\frac{\partial p}{\partial n} = \frac{\partial \phi_{BI}}{\partial n} = 0$ in Eq. (13). The boundary condition for the temperature depends on whether the immersed surface of the body is adiabatic or isothermal. Assuming an adiabatic boundary condition at the immersed body, the temperature gradient normal to the surface $\frac{\partial T}{\partial n} = 0$ is set to zero by enforcing a zero density gradient $\frac{\partial \rho}{\partial n} = 0$. Thus, for the variables ρ and p Neumann boundary conditions are employed. According to the boundary conditions considered for the immersed body, the values of the conservative perturbation variables at the ghost points are determined once the flow variables at the image points

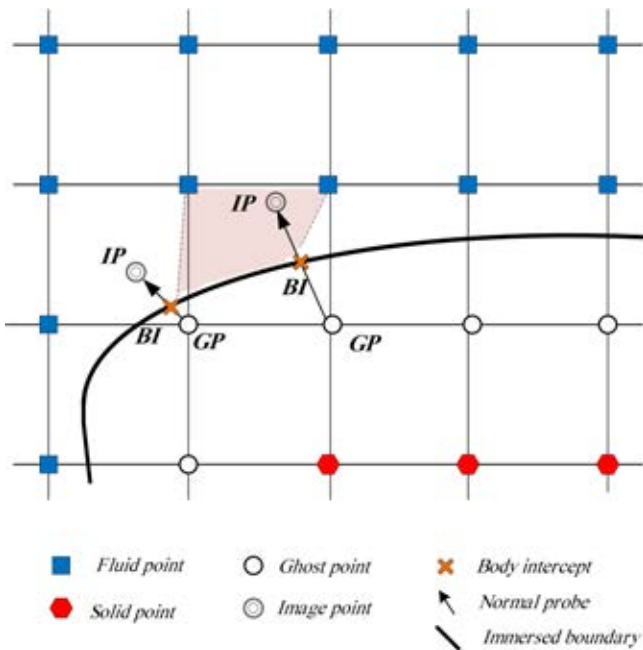


Figure 4: Schematic of the situation when two of the surrounding interpolation points lie inside the immersed body.

are interpolated using:

$$\begin{aligned} \rho'_{GP} &= \rho'_{IP} \\ (\rho u)'_{GP} &= -(\rho u)'_{IP} \\ (\rho v)'_{GP} &= -(\rho v)'_{IP} \\ (\rho E)'_{GP} &= (\rho E)'_{IP} \end{aligned} \quad (19)$$

As mentioned above, applying the first derivative approximation twice for computing the second derivative will make the stencil wider. For the proper treatment of wide stencils in computing the second derivative, the first derivatives of the viscous terms are computed up to and including the ghost points, treating the solid points inside the ghost point layers as domain boundaries when employing the differencing stencil. Using this procedure, we ensure that the derivatives of the viscous fluxes at the fluid points closest to the immersed boundary are computed with high order.

Non-reflecting characteristic boundary conditions are employed at the inflow and outflow boundaries to minimize wave reflections. The Navier–Stokes characteristic boundary conditions (NSCBC) developed by (Poinsot and Lele, 1992) are employed to approximate incoming waves based on local one-dimensional inviscid (LODI) relations. The primitive variables can be related to the wave amplitude (\mathcal{L}_i) by LODI relations. The amplitudes of the characteristic waves are $\mathcal{L}_1 = \lambda_1(\frac{\partial p}{\partial x} - \rho c \frac{\partial u}{\partial x})$, $\mathcal{L}_2 = \lambda_2(c^2 \frac{\partial p}{\partial x} - \frac{\partial p}{\partial x})$, $\mathcal{L}_3 = \lambda_3(\frac{\partial v}{\partial x})$ and $\mathcal{L}_4 = \lambda_4(\frac{\partial p}{\partial x} + \rho c \frac{\partial u}{\partial x})$. Since fully non-reflecting conditions may lead to an ill-posed problem (Poinsot and Lele, 1992), this approach is partially reflecting. Imposing a constant pressure at the outlet requires $\mathcal{L}_1 = -\mathcal{L}_4$. To keep the reflections low and the pressure close to atmospheric pressure, the incoming wave amplitude is set to

$$\mathcal{L}_1 = K(p - p_{\text{atm}}) \quad (20)$$

where K is a relaxation coefficient. Rudy and Strikwerda proposed the relaxation coefficient as $K = \Lambda(1 - \text{Ma}^2)(c/L_t)$ where Ma is the Mach number, c the speed of sound, L_t the total length of the domain and Λ a parameter (Rudy and Strikwerda, 1980). The optimum value $\Lambda = 0.25$ derived by (Rudy and Strikwerda, 1980) is employed. For reverse flow (negative velocity in x -direction) at the outlet, \mathcal{L}_1 , \mathcal{L}_2 and \mathcal{L}_3 are set to zero. A similar boundary treatment at inflow and outflow was used by (Khalili *et al.*, 2016).

RESULTS

In order to assess the accuracy of the immersed boundary methodology, a two-dimensional steady state heat problem is first solved. Then, the IBM is applied to a two-dimensional flow past a circular cylinder at a range Reynolds numbers to demonstrate the ability and performance of the method for simulating compressible viscous flow.

Steady state heat equation

To verify the order of spatial accuracy of the current immersed boundary scheme, a steady state heat transfer problem has been considered. Since the ghost point immersed boundary method is second-order accurate (Ghias *et al.*, 2007; Mittal *et al.*, 2008), care has been taken to maintain a second-order spatial accuracy in the imposition of boundary conditions on the immersed boundary. The steady state heat equation reads

$$\nabla^2 T = \sigma \quad (21)$$

where σ is a source term, i.e. $-\kappa\sigma$ is the rate of heat generation per unit volume. The exact solution for this case in polar coordinates can be expressed as

$$T(r) = \sigma \frac{r^2}{4} + A \ln(r) + B \quad (22)$$

where A and B depend on the boundary conditions type and their values.

The numerical solution by means of IBM is implemented to solve Eq.(21) in Cartesian coordinates. The second and fourth order central finite difference methods for second derivatives are employed for spatial discretization of the regular fluid points.

$$T_{xx}^{(2)} = (T_{i+1} - 2T_i + T_{i-1})/\Delta x^2 \quad (23)$$

$$T_{xx}^{(4)} = (-T_{i+2} + 16T_{i+1} - 30T_i + 16T_{i-1} - T_{i-2})/(12\Delta x^2) \quad (24)$$

The temperature distribution is solved between two concentric cylinders with inner and outer diameters $D_{\text{inner}} = 3.5$ and $D_{\text{outer}} = 8.5$, respectively, embedded in a square domain of edge length $L = 10$. The $\sigma = -0.45$ is chosen and the temperatures of the inner and outer cylinders are $T_{\text{inner}} = 5$ and $T_{\text{outer}} = 10$, respectively. The immersed boundary approach is implemented at the cylinder interfaces. The results from different grids on a uniform Cartesian grid ($N \times N$) from $N = 100$ to 1000 are compared with the exact solution to compute the L_2 and L_∞ norms. Fig. 5 shows the errors for different grids.

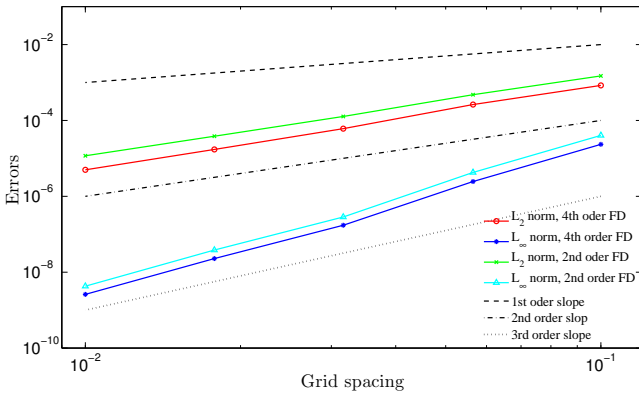


Figure 5: L_2 and L_∞ norms computed at various grid levels with IBM for 2D steady heat Eq. (21).

The first, second and third-order convergence rates are also included in Fig. 5 for reference. This figure indicates that a second-order rate of convergence has been achieved by the Poisson solver for Dirichlet boundary conditions at immersed boundaries. The error are slightly lower for the fourth order difference method than for the second order one.

Flow past circular cylinder

To verify and validate the present immersed boundary treatment for a compressible flow solver, the benchmark flow over a circular cylinder is firstly simulated at the Reynolds numbers of 20 and 40 based on the free-stream velocity and diameter of the cylinder. It is known that steady flow over a circular cylinder can persist up to Reynolds numbers of about 40. The free-stream Mach number for the simulation is set as a small number $Ma = 0.03$ in order to be comparable to the simulations performed using incompressible solvers. Then, the unsteady flow over a circular cylinder has been chosen

to verify the proposed IB method at the Reynolds number of 100 and Mach number 0.25.

The computational domain size is $90D \times 40D$ where D is the diameter of the cylinder. The center of the cylinder is located at the point $(20D, 20D)$ of the coordinate system. The computational domain is sizeable to reduce the effects of domain boundaries and wave reflections form the inlet and outlet boundaries. It has been observed that those could lead to a momentous error when computing the lift and drag coefficients. In the present work, the block structured computational domain has been discretized with non-uniform Cartesian grids, where the block corresponding to the cylinder has a much finer grid spacing of $(\Delta x = \Delta y = D/25)$ at $Re = 20$ and 40, and grid spacing of $(\Delta x = \Delta y = D/50)$ at $Re = 100$. At these grid resolutions, the lift and drag coefficients are sufficiently converged. Sufficient grid resolution around the cylinder is crucial to obtain the drag and lift coefficients accurately. Additionally, to capture the von Kármán vortex shedding, the wake region needs to be resolved properly. The grid spacing Δx and Δy was smoothly stretched from $(\Delta x = \Delta y = D/25)$ at $Re = 20$ and 40, and $\Delta x = \Delta y = D/50$ for $Re=100$ to $\Delta x = \Delta y = D/2$ near the inflow, outflow, top and bottom boundaries. Symmetry boundary conditions are applied on the top and bottom of the computational domain. At the inflow, the velocities in the x - and y -directions are imposed using a uniform inlet profile normal to the boundary, $u(x = 0, t) = U_\infty$ and $v = 0$. In addition, the inlet temperature is set to $T = T_0 = 310$ K. The outlet pressure is set to atmospheric pressure, i.e., $p' = p - p_0 = p - p_{\text{atm}} = 0$ Pa.

The drag and lift coefficients are defined as $C_D = \frac{F_D}{\frac{1}{2}\rho_\infty U_\infty^2 D}$ and $C_L = \frac{F_L}{\frac{1}{2}\rho_\infty U_\infty^2 D}$, respectively, where F_D and F_L are the drag and lift forces. The total force on the cylinder is given by the sum of the pressure and viscous force integrated over the cylinder surface $F = -\oint p_B \cdot \mathbf{n} ds + \oint \tau_B \cdot \mathbf{n} ds$ where \mathbf{n} is the outer unit vector normal to the cylinder, and p_B and τ_B are pressure and the viscous stress tensor on the body surface, respectively. These quantities are based on the evaluation of surface pressure and viscous stress. The procedure used to compute these surface quantities needs some explanation. In the current solver, four nodes surrounding a body-intercept point corresponding to the first layer of ghost points are identified and then a bilinear interpolation is used to estimate the pressure and viscous stress tensor at the body intercept. The viscous stress at the involved ghost points and fluid points are computed in a straightforward manner by using our high order method.

Figs. 6 - 9 show streamlines and vorticity contours for $Re = 20$ and $Re = 40$, respectively. The geometrical properties of the vortices behind the cylinder are schematically illustrated in Fig. 10 (Canuto and Taira, 2015). The quantitative comparison of these parameters as well as the drag coefficient with available numerical and experimental results are given in Table 1.

Fig. 11 presents the instantaneous spanwise vorticity ω_z contours for $Re = 100$ indicating the presence of the von Kármán vortex street. The vortex shedding leads to time-varying lift and drag forces until they reach to a periodic oscillatory form. The Strouhal number $St = \frac{fD}{U_\infty}$, where f is the vortex shedding frequency, is computed from the temporal variation of the lift coefficient. Due to the unsteadiness of the flow, the comparison of the average values of the lift and drag coefficients as well as the amplitude of the sinusoidal variation in time of the lift and drag coefficients is central. The results for the time-averaged lift and drag coefficients, the amplitude of

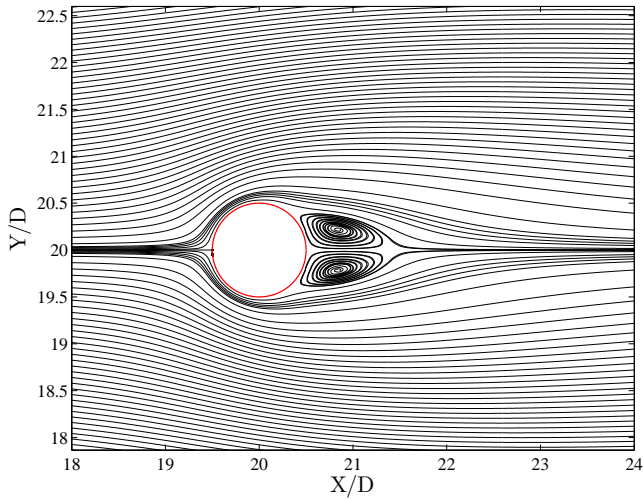


Figure 6: Streamlines for computed flow past a circular cylinder at $Re = 20$ and $Ma = 0.03$.

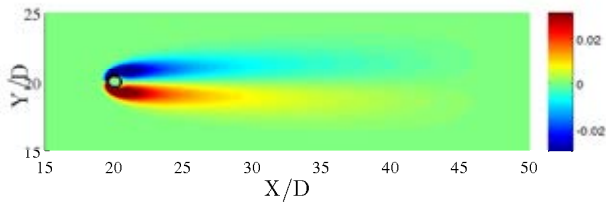


Figure 7: Vorticity contours for computed flow past a circular cylinder at $Re = 20$ and $Ma = 0.03$.

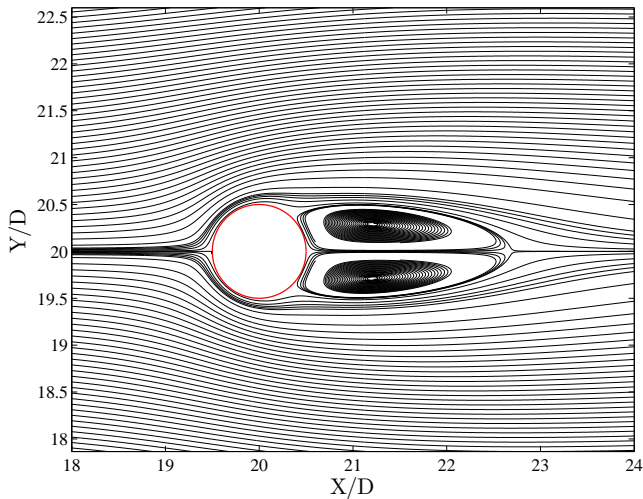


Figure 8: Streamlines for computed flow past a circular cylinder at $Re = 40$ and $Ma = 0.03$.

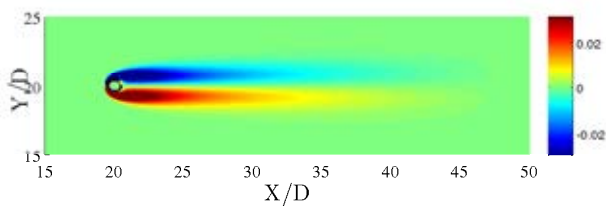


Figure 9: Vorticity contours for computed flow past a circular cylinder at $Re = 40$ and $Ma = 0.03$.

Table 1: Comparison of computed data with available numerical and experimental data at $Re = 20$ and $Re = 40$. (Exp.) indicates the experimental results.

	Re = 20					
	Ma	L	a	b	$\theta(deg)$	C_D
(Tritton, 1959)(Exp.)	-	-	-	-	-	2.09
(Dennis and Chang, 1970)	0	0.94	-	-	43.7	2.05
(Coutanceau and Bouard, 1977)(Exp.)	-	0.93	0.33	0.46	45.0	-
(Fornberg, 1980)	0	0.91	-	-	45.7	2.0
(Linnick and Fasel, 2003)	0	0.93	0.36	0.43	43.5	2.06
(De Palma <i>et al.</i> , 2006)	0.03	0.93	0.36	0.43	44.6	2.05
(Canuto and Taira, 2015)	0	0.92	0.36	0.42	43.7	2.07
Present study	0.03	0.93	0.36	0.43	43.9	2.05
	Re = 40					
	Ma	L	a	b	$\theta(deg)$	C_D
(Tritton, 1959)(Exp.)	-	-	-	-	-	1.59
(Dennis and Chang, 1970)	0	2.35	-	-	53.8	1.52
(Coutanceau and Bouard, 1977)(Exp.)	-	2.13	0.76	0.59	53.8	-
(Fornberg, 1980)	0	2.24	-	-	55.6	1.50
(Linnick and Fasel, 2003)	0	2.28	0.72	0.60	53.6	1.52
(De Palma <i>et al.</i> , 2006)	0.03	2.28	0.72	0.60	53.8	1.55
(Canuto and Taira, 2015)	0	2.24	0.72	0.59	53.7	1.54
Present study	0.03	2.22	0.72	0.59	53.1	1.52

their changes as well as the Strouhal number of the present study are compared to published results in Table 2. Table 1 and 2 confirm that for the present study all results compare very well with results reported in the literature.

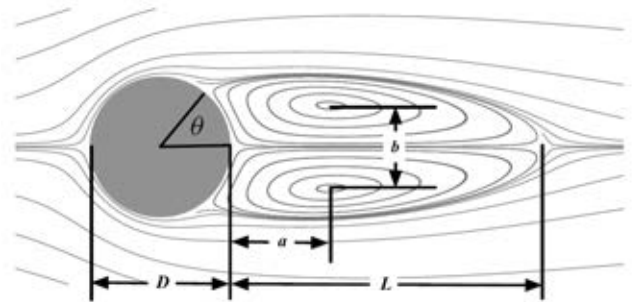


Figure 10: Definitions of the relevant geometrical parameters of the symmetric deperation region behind the cylinder (Canuto and Taira, 2015).

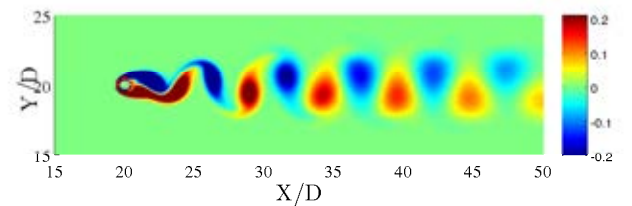


Figure 11: Vorticity contours for computed flow past a circular cylinder at $Re = 100$ and $Ma = 0.25$

Table 2: Comparison of computed data with available numerical and experimental data at $Re = 100$.

	Re = 100			
	Ma	St	C_D	C_L
(Berger and Wille, 1972)	0	0.16-0.17	-	-
(Liu <i>et al.</i> , 1998)	0	0.165	1.35 ± 0.012	± 0.339
(Linnick and Fasel, 2003)	0	0.166	1.34 ± 0.009	± 0.333
(Mittal <i>et al.</i> , 2008)	0	-	1.35	-
(Karagiozis <i>et al.</i> , 2010)	0.25	0.168	1.336	± 0.319
(Canuto and Taira, 2015)	0.25	0.163	1.378	± 0.325
Present study	0.25	0.1667	1.33 ± 0.013	± 0.323

CONCLUSION

In this paper, we have combined highly stable high-order SBP operators with an immersed boundary method which permits us to use Cartesian grids for arbitrary geometries for solving the compressible Navier–Stokes equations accurately and efficiently. SBP operators which are 6th order accurate in the interior and 3rd order accurate near the boundaries is employed. To achieve high accuracy and easy parallelization, the 4th order explicit Runge–Kutta method is applied. The methodology is applied to compute steady and unsteady flow problems to demonstrate its versatility as well as its accuracy. The flow past a circular cylinder for moderate values of Reynolds number and Mach number is assessed. A good agreement with available experimental and numerical results is achieved.

ACKNOWLEDGMENTS

The present research is a part of a research project entitled "Modeling of obstructive sleep apnea by fluid-structure interaction in the upper airways" funded by the Research Council of Norway. The simulations in our work are done on the Vilje cluster at NTNU which is gratefully acknowledged.

REFERENCES

BALARAS, E. (2004). "Modeling complex boundaries using an external force field on fixed Cartesian grids in large-eddy simulations". *Computers & Fluids*, **33**(3), 375–404.

BERGER, E. and WILLE, R. (1972). "Periodic flow phenomena". *Annual Review of Fluid Mechanics*, **4**(1), 313–340.

BREHM, C., HADER, C. and FASEL, H.F. (2015). "A locally stabilized immersed boundary method for the compressible navier–stokes equations". *Journal of Computational Physics*, **295**, 475–504.

CANUTO, D. and TAIRA, K. (2015). "Two-dimensional compressible viscous flow around a circular cylinder". *Journal of Fluid Mechanics*, **785**, 349–371.

COUTANCEAU, M. and BOUARD, R. (1977). "Experimental determination of the main features of the viscous flow in the wake of a circular cylinder in uniform translation. Part 1. Steady flow". *Journal of Fluid Mechanics*, **79**(02), 231–256.

DE PALMA, P., DE TULLIO, M., PASCAZIO, G. and NAPOLITANO, M. (2006). "An immersed-boundary method for compressible viscous flows". *Computers & Fluids*, **35**(7), 693–702.

DE TULLIO, M.D., DE PALMA, P., IACCARINO, G., PASCAZIO, G. and NAPOLITANO, M. (2007). "An immersed boundary method for compressible flows using local grid refinement". *Journal of Computational Physics*, **225**(2), 2098–2117.

DENNIS, S. and CHANG, G.Z. (1970). "Numerical solutions for steady flow past a circular cylinder at Reynolds

numbers up to 100". *Journal of Fluid Mechanics*, **42**(03), 471–489.

FADLUN, E., VERZICCO, R., ORLANDI, P. and MOHD-YUSOF, J. (2000). "Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations". *Journal of Computational Physics*, **161**(1), 35–60.

FORNBERG, B. (1980). "A numerical study of steady viscous flow past a circular cylinder". *Journal of Fluid Mechanics*, **98**(04), 819–855.

GHIAS, R., MITTAL, R. and DONG, H. (2007). "A sharp interface immersed boundary method for compressible viscous flows". *Journal of Computational Physics*, **225**(1), 528–553.

GOLDSTEIN, D., HANDLER, R. and SIROVICH, L. (1993). "Modeling a no-slip flow boundary with an external force field". *Journal of Computational Physics*, **105**(2), 354–366.

GUSTAFSSON, B. (2008). *High order difference methods for time dependent PDE*. Springer, Berlin.

GUSTAFSSON, B., KREISS, H.O. and OLIGER, J. (1995). *Time dependent problems and difference methods*. John Wiley & Sons, New York.

HAERI, S. and SHRIMPTON, J. (2012). "On the application of immersed boundary, fictitious domain and body-conformal mesh methods to many particle multiphase flows". *International Journal of Multiphase Flow*, **40**, 38 – 55.

KARAGIOZIS, K., KAMAKOTI, R. and PANTANO, C. (2010). "A low numerical dissipation immersed interface method for the compressible navier–stokes equations". *Journal of Computational physics*, **229**(3), 701–727.

KHALILI, M., LARSSON, M. and MÜLLER, B. (2016). "Interaction between a simplified soft palate and compressible viscous flow". *Journal of Fluids and Structures*, **67**, 85–105.

LARSSON, M. and MÜLLER, B. (2009). "Numerical simulation of confined pulsating jets in human phonation". *Computers & Fluids*, **38**(7), 1375–1383.

LINNICK, M. and FASEL, H. (2003). "A high-order immersed boundary method for unsteady incompressible flow calculations". *41st Aerospace Sciences Meeting and Exhibit*, 1124.

LIU, C., ZHENG, X. and SUNG, C. (1998). "Pre-conditioned multigrid methods for unsteady incompressible flows". *Journal of Computational Physics*, **139**(1), 35–57.

MITTAL, R. and IACCARINO, G. (2005). "Immersed boundary methods". *Annual Review of Fluid Mechanics*, **37**, 239–261.

MITTAL, R., DONG, H., BOZKURTTAS, M., NAJJAR, F., VARGAS, A. and VON LOEBBECKE, A. (2008). "A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries". *Journal of Computational Physics*, **227**(10), 4825–4852.

MOHD-YUSOF, J. (1997). "Combined immersed-boundary/B-spline methods for simulations of flow in complex geometries". *Center for Turbulence Research Annual Research Briefs*, 317–325.

MÜLLER, B. (2008). "High order numerical simulation of aeolian tones". *Computers & Fluids*, **37**(4), 450–462.

PESKIN, C.S. (1972). "Flow patterns around heart valves: A numerical method". *Journal of Computational Physics*, **10**(2), 252–271.

POINSOT, T.J. and LELE, S. (1992). "Boundary conditions for direct simulations of compressible viscous flows". *Journal of Computational Physics*, **101**(1), 104–129.

RUDY, D.H. and STRIKWERDA, J.C. (1980). “A non-reflecting outflow boundary condition for subsonic Navier-Stokes calculations”. *Journal of Computational Physics*, **36**(1), 55–70.

SAIKI, E. and BIRINGEN, S. (1996). “Numerical simulation of a cylinder in uniform flow: Application of a virtual boundary method”. *Journal of Computational Physics*, **123**(2), 450–465.

SESTERHENN, J., MÜLLER, B. and THOMANN, H. (1999). “On the cancellation problem in calculating compressible low Mach number flows”. *Journal of Computational Physics*, **151**(2), 597–615.

SOTIROPOULOS, F. and YANG, X. (2014). “Immersed boundary methods for simulating fluid-structure interaction”. *Progress in Aerospace Sciences*, **65**, 1–21.

STRAND, B. (1994). “Summation by parts for finite difference approximations for d/dx ”. *Journal of Computational Physics*, **110**(1), 47–67.

SVÄRD, M. and NORDSTRÖM, J. (2014). “Review of summation-by-parts schemes for initial-boundary-value problems”. *Journal of Computational Physics*, **268**, 17–38.

TRITTON, D. (1959). “Experiments on the flow past a circular cylinder at low Reynolds numbers”. *Journal of Fluid Mechanics*, **6**(04), 547–567.

TSENG, Y.H. and FERZIGER, J.H. (2003). “A ghost-cell immersed boundary method for flow in complex geometry”. *Journal of Computational Physics*, **192**(2), 593–623.

VISBAL, M.R. and GAITONDE, D.V. (2002). “On the use of higher-order finite-difference schemes on curvilinear and deforming meshes”. *Journal of Computational Physics*, **181**(1), 155–185.

YE, T., MITTAL, R., UDAYKUMAR, H. and SHYY, W. (1999). “An accurate Cartesian grid method for viscous incompressible flows with complex immersed boundaries”. *Journal of Computational Physics*, **156**(2), 209–240.

APPENDIX A. ENERGY METHOD

To demonstrate the energy method, we apply the procedure to a simplified PDE, that is, rather than analysing the full Navier–Stokes equations in this section, we only focus on the 1D convection–diffusion equation as a model equation.

$$\begin{aligned} u_t + au_x &= bu_{xx}, & 0 \leq x \leq 1 & \quad t \geq 0 \\ u(x, 0) &= f(x) \\ u(0, t) &= u(1, t) = g(t) = 0 \end{aligned} \quad (25)$$

where a and b are assumed to be constant and positive, and u is the dependent variable. The L_2 scalar product for two real functions v and w is defined by

$$(v, w) = \int_0^1 v(x)w(x)dx \quad (26)$$

which then defines the L_2 norm of the continuous solution at time t and energy $E(t) = \|u(\cdot, t)\|^2 = (u, u)$. Using integration by parts $(v, w_x) = v(1, t)w(1, t) - v(0, t)w(0, t) - (v_x, w)$, the energy method leads to

$$\begin{aligned} \frac{dE}{dt} &= \frac{d}{dt} \|u(\cdot, t)\|^2 = (u_t, u) + (u, u_t) \\ &= (-au_x + bu_{xx}, u) + (u, -au_x + bu_{xx}) \\ &= -a[u^2(1, t) - u^2(0, t)] + 2b[u(1, t)u_x(1, t) - u(0, t)u_x(0, t)] \\ &\quad - 2b(u_x, u_x) \leq au^2(0, t) + 2b[u(1, t)u_x(1, t) - u(0, t)u_x(0, t)] \\ &= 0 \end{aligned} \quad (27)$$

which yields a non growing solution, i.e. $E(t) \leq E(0) = \|f(x)\|^2$. Thus, the energy is bounded by the initial condition.

APPENDIX B. SUMMATION BY PARTS OPERATORS

(Khalili *et al.*, 2016)

The SBP operators are constructed to guarantee a discrete energy estimate similar to the continuous energy estimate above.

$$\begin{aligned} u_t + au_x &= bu_{xx}, & 0 \leq x \leq 1 & \quad t \geq 0 \\ u(x, 0) &= f(x) \\ u(0, t) &= u(1, t) = g(t) = 0 \end{aligned} \quad (28)$$

where a and b are assumed to be constant and positive, and u is the dependent variable.

The basis of getting such an energy estimate is to satisfy integration by parts in the discrete sense called Summation–By–Parts (SBP) property (Gustafsson, 2008; Svärd and Nordström, 2014). To outline this technique for model problem (28), we consider $u_j = u_j(t)$ the numerical solution of the convection–diffusion equation at grid point $x_j = jh$, $j = 0, \dots, N$, with grid spacing $h = \frac{1}{N}$. The solution vector containing the solution at the discrete grid points is $\mathbf{u} = [u_0(t), u_1(t), \dots, u_N(t)]^T$. Using a difference operator Q approximating the first derivative in space, the semi-discrete form of the model equation can be expressed as

$$\frac{d\mathbf{u}}{dt} = -aQ\mathbf{u} + bQQ\mathbf{u}, \quad u_j(0) = f(x_j) \quad (29)$$

The discrete scalar product and corresponding norm and energy can be defined by

$$\begin{aligned} (\mathbf{u}, \mathbf{v})_h &= h\mathbf{u}^T H\mathbf{v}, \\ E_h(t) &= \|\mathbf{u}\|_h^2 = (\mathbf{u}, \mathbf{u})_h \end{aligned} \quad (30)$$

where H is a diagonal and positive definite matrix defined by $H = \text{diag}(H_L, I, H_R)$. The SBP property is satisfied by the difference operator Q , if

$$(\mathbf{u}, Q\mathbf{v})_h = u_N v_N - u_0 v_0 - (Q\mathbf{u}, \mathbf{v})_h \quad (31)$$

or if Q can be written on the form $hQ = H^{-1}P$ for P satisfying

$$P + P^T = E_N - E_0 = \text{diag}(-1, 0, \dots, 0, 1) \quad (32)$$

where $E_0 = \text{diag}(1, 0, \dots, 0)$ and $E_N = \text{diag}(0, 0, \dots, 1)$. Using the semi-discrete equation 29, the energy estimate for the semi-discrete problem can be obtained as

$$\begin{aligned} \frac{dE}{dt} &= \frac{d}{dt} \|u(\cdot, t)\|^2 = (u_t, u)_h + (u, u_t)_h \\ &= (-aQu + bQQ\mathbf{u}, \mathbf{u})_h + (u, -aQu + bQQ\mathbf{u})_h \\ &= -a[u_N^2 - u_0^2] + 2b[u_N(Q\mathbf{u})_N - u_0(Q\mathbf{u})_0] \\ &\quad - 2b(Q\mathbf{u}, \mathbf{u})_h \leq au_N^2 + 2b[u_N(Q\mathbf{u})_N - u_0(Q\mathbf{u})_0]. \end{aligned} \quad (33)$$

We would get non-growing energy in time if the homogeneous boundary conditions could directly be imposed in (33). However, this will change the difference operator Q such that its SBP property might be lost. To avoid this problem, boundary conditions are weakly imposed by the simultaneous approximation term (SAT) technique (Gustafsson, 2008). A first derivative SBP operator with diagonal quadrature matrix H in 30 is a $O(h^{2s})$ accurate central difference operator which is $O(h^s)$ accurate at and near boundaries $s = 1, 2, 3$. Such an SBP operator is globally $O(h^{s+1})$ accurate.

APPENDIX C. REFORMULATION OF COEFFICIENTS

The four unknown coefficients C_i , $i = 1, \dots, 4$ can be determined using values of the four variables surrounding the image point. It can be expressed as

$$\mathbf{C} = \mathbf{V}^{-1}\{\phi\} \quad (34)$$

where \mathbf{V} is the Vandermonde matrix corresponding to the bi-linear interpolation scheme for four surrounding nodes. The value at the image point can be expressed as

$$\phi_{IP} = \begin{bmatrix} 1 & x_{IP} & y_{IP} & x_{IP}y_{IP} \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \end{bmatrix} \quad (35)$$

The vector in bracket can be expressed as

$$\mathbf{V}_{IP} = \begin{bmatrix} 1 & x_{IP} & y_{IP} & x_{IP}y_{IP} \end{bmatrix} = \sum_{i=1}^4 \alpha_i V_i \quad (36)$$

where V_i is the i th row of \mathbf{V} and α_i depends on the coordinates of the image point and the four surrounding nodes. Thereby, the matrix equation for α can be written as

$$\begin{bmatrix} 1 \\ x_{IP} \\ y_{IP} \\ x_{IP}y_{IP} \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ x_1y_1 & x_2y_2 & x_3y_3 & x_4y_4 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix} \quad (37)$$

By rearranging, α can be obtained as

$$\alpha = \mathbf{V}^{-T} \mathbf{V}_{IP}^T. \quad (38)$$

Thus, the value at the image point can be expressed as

$$\phi_{IP} = \phi^T \mathbf{V}^{-T} \mathbf{V}_{IP}^T \quad (39)$$

Inserting the result obtained in Eq. (38), the value at the image point can be obtained as

$$\phi_{IP} = \sum_{i=1}^4 \alpha_i \phi_i \quad (40)$$