Journal of Computational Physics 267 (2014) 225–246

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A new approach for conjugate heat transfer problems using immersed boundary method for curvilinear grid based solvers

Krishnamurthy Nagendra [a,b], Danesh K. Tafti [a,b,*], Kamal Viswanath [b,1,2]

[a] *National Energy Technology Laboratory, Pittsburgh PA, USA*
[b] *Dept. of Mechanical Engineering, Virginia Tech, Blacksburg VA, USA*

ARTICLE INFO

Article history:
Received 29 August 2013
Received in revised form 1 January 2014
Accepted 28 February 2014
Available online 11 March 2014

Keywords:
Immersed boundary method
Curvilinear coordinates
Non-staggered grid
Conjugate heat transfer
Micro-channel flows

ABSTRACT

Use of immersed boundary method (IBM) based techniques have helped considerably in easing the grid generation process in flows involving complex geometries and/or large boundary movements. Body fitting grid based techniques still, however, are advantageous in terms of accuracy and efficiency. In this work, we have developed an IBM scheme applicable to curvilinear coordinates, aiming at taking advantage of both the methodologies. The framework uses efficient algorithms for search, locate, and interpolate operations. A new method of implementing the conjugate heat transfer (CHT) boundary condition is proposed which is a direct extension of the method used for other boundary conditions and does not involve any complex interpolations like previous CHT implementations using IBM. The developed scheme is shown to be applicable to complex geometries on curvilinear grids, while also being very efficient, consuming less than 1% of the total simulation time per time-step. Very good scalability on massive computations is demonstrated using strong scaling study up to 1024 cores. Detailed code verification process is undertaken to show that the method is second-order accurate for both the velocity and temperature fields for all the boundary conditions considered. Further, validation studies involving uniform flow over stationary and oscillating cylinders are carried out to demonstrate the accuracy of the developed method. Lastly, simulations are performed to study flow and conjugate heat transfer through thick-walled micro-channels using body-fitted background grids and the results are shown to be in excellent agreement with previously published results.

1. Introduction

In the last couple of decades, the use of boundary non-conforming techniques for computational fluid dynamics problems has gained considerable popularity, owing mainly to the versatile capabilities that such implementations offer in comparison to traditional CFD codes. The time spent on generation of body-fitting meshes requires a huge amount of user input for complex geometries. For moving boundary problems, the structured body-fitting grids require complex grid deformations and/or re-meshing algorithms, most of which still have limited allowable range of boundary movement. While unstructured

* Corresponding author at: 114-I Randolph Hall, Dept. of Mechanical Engg., Virginia Tech, Blacksburg, VA 24061, USA. Tel.: +1 540 231 9975; fax: +1 540 231 9100.
E-mail address: dtafti@exchange.vt.edu (D.K. Tafti).
[1] This research was performed while K. Viswanath was at Virginia Tech.
[2] Present address: Laboratory for Computational Physics and Fluid Dynamics, Naval Research Laboratory, Washington, DC 20375, USA.

http://dx.doi.org/10.1016/j.jcp.2014.02.045
0021-9991/

Nomenclature

A	area
d	distance
D	diameter
L	length
n	surface normal (direction)
Nu	Nusselt number
p	pressure
p'	pressure correction
Pr	Prandtl number
q''	heat flux
R	radius
Re	Reynolds number
St	Strouhal number
t	time
T	temperature
T_c	characteristic temperature
u	velocity
U	cell-face flux

Greek symbols

Γ	surface entity
Δ	grid size
Δ_p	probe distance
κ	thermal conductivity
μ	absolute viscosity
ρ	density
ϕ	flow variable
Ω	volume entity

Subscripts

BC	boundary condition
c	cell
e	nearest element
f	fluid
i	inner wall
IB	IB node
mp	mirror probe
n	node
o	outer wall
p	probe
ref	reference value
s	solid
w	wall

Superscripts

$*$	dimensional value

Miscellaneous

a, b, g	computational space coordinates
i, j, k	structured grid indices
r, θ	radial coordinates
x, y, z	Cartesian coordinates
(bold)	vector quantities
$\sim$	(overhead) intermediate value
$\mathfrak{F}(\cdot)$	time-stepping operator
$\mathfrak{G}(\cdot)$	boundary condition operator
$\mathfrak{M}(\cdot)$	mass conservation operator

solvers are better in terms of ease of grid generation and re-meshing, they are much harder to implement and require large amounts of book-keeping and memory usage. In contrast, the use of boundary non-conforming formulations allows simulations of fluid flow around bodies with complex features, large movements and surface deformations on a fixed background mesh with minimal overhead in setup.

Solvers based on such methodologies have the solid boundary "immersed" in a background mesh and generally utilize a series of special treatments applied near the solid surface. These methods are commonly termed immersed boundary methods (IBM). The idea originated from the work of Peskin [1] where this method was used to simulate blood flow through heart valves using a fixed Cartesian background grid. Following this work, several modifications and improvisations to this method have been proposed over the years showcasing its wide range of applicability. A wealth of literature is available on the range of capabilities that can be developed using immersed boundary techniques [2]. Here, we first present a brief review of the different types of implementations available. The implementations mainly vary in the way the boundary conditions are applied. In general, the use of IBM can be thought of as a modification applied to the governing Navier–Stokes equations. The exact nature of this modification distinguishes the different schemes from each other.

A widely used approach is the use of a forcing function – usually a Dirac delta function – as a source term in the momentum equations. Peskin and coworkers utilized a forcing function that is spread across a few cells neighboring the immersed surface [1,3–5]. In fact, numerous applications exist in the literature, especially for flows with elastic boundaries such as fluid–structure interaction problems involving blood flow through arteries, for which this approach is a natural choice. One drawback of such formulations, however, is the smearing of the boundaries due to the spreading nature of the forcing function. While this may not be an issue for elastic boundaries, this increases the grid resolution requirement substantially for flows with rigid immersed surfaces.

This issue can be avoided by directly computing the forcing function to be applied from the discretized momentum equations and using it as a source term for the nodes that lie in the immediate vicinity of the immersed boundary. This approach was pioneered by Mohd-Yusof and coworkers [6,7] who proposed a second-order accurate scheme based on a locally dependent interpolation stencil. This method is applicable to sharp solid interfaces and does not suffer from the blurring which may occur in a continuous forcing approach. Several improvements have been proposed since, mainly with an aim of developing a general methodology applicable to wider spectrum of problems including arbitrarily shaped 3D surface contours with deformation and/or movement [8–11].