Comput. Methods Appl. Mech. Engrg. 253 (2013) 146–159

Contents lists available at SciVerse ScienceDirect

Comput. Methods Appl. Mech. Engrg.

journal homepage: www.elsevier.com/locate/cma

A finite volume formulation for transient convection and diffusion equations with unstructured distorted grids and its applications in fluid flow simulations with a collocated variable arrangement

S.-C. Xue *, G.W. Barton

School of Chemical and Biomolecular Engineering, The University of Sydney, NSW 2006, Australia

ARTICLE INFO

Article history:
Received 19 July 2012
Received in revised form 24 September 2012
Accepted 27 September 2012
Available online 13 October 2012

Keywords:
Convective and diffusive fluxes
Finite volume formulation
Multi-dimensional flow
SIMPLER algorithm
Unstructured grid

ABSTRACT

This paper describes a finite volume method for simulating transport processes governed by convection–diffusion type equations. The formulation is based on a cell-centred, unstructured grid. With an edge-based data structure, discretisation is independent of control volume (or cell) shape. By using a surface vector decomposition at the midpoint of the interface between cells, along with a deferred-correction approach, any cross-diffusion due to grid skewness can be readily accounted for when discretising the diffusive flux. For modelling fluid flow processes, a collocated arrangement of variables is employed so that a single coefficient matrix applies for the momentum equations of each velocity component. To avoid 'checkerboard oscillation' (arising from pressure–velocity decoupling) occurring under the collocated variable arrangement when a pressure-based solution algorithm is employed, a novel pseudo-flux interpolation method is proposed for unstructured grids, ensuring that the solution is both under-relaxation factor and time-step (for transient calculation) independent. The methodology can be formulated within a framework whereby either a coupled or a decoupled solution algorithm can be employed. The features and advantages of the method are demonstrated by solution of the Navier–Stokes equations for two benchmark flow problems.

1. Introduction

A wide range of transport processes in diverse fields can be mathematically described by a generic convection–diffusion equation that requires numerical solution in all but the simplest cases. In many computational scenarios, the solution domain can be very complex. Thus the ability to efficiently handle complex geometry can be a major consideration in developing suitable numerical simulation methods. In this regard, finite element methods (FEMs) [1] are often the preferred option. However, to ensure numerical convergence and solution accuracy, there are a range of restrictions on FEM grid generation, such as using a single element shape (for example, either triangular or a quadrilateral for two-dimensional cases) with conditions placed on the angles within an element [2]. Considerable effort has been devoted to developing FEM mesh generation algorithms that meet these requirements. However, in some modelling situations, such as flow problems involving free surfaces, an initial smooth mesh may be severely distorted during the solution process. Therefore, numerical methods based on unstructured meshes, free of any shape or angle restrictions, are highly desirable. In this regard, the physically meaningful finite volume methods (FVMs) [3], based on the conservation of mass, momentum and energy over an arbitrarily shaped cell (or control volume), provide an attractive alternative to FEMs. Depending on where dependent variables are stored, how the control volume (over which the conservation laws are applied) is defined, what kind of data structure is used for convective–diffusive flux discretisation, and what kind of solution strategy is implemented, a variety of FVMs with different features have been developed.

In a vertex-centred formulation [4–9], dependent variables are stored (and solved for) at each vertex, and a control volume for the vertex is constructed using the 'median dual' of the cell grid associated with the vertex, as shown in Fig. 1. Depending on the data structure used, there are two different ways of discretising the convective–diffusive fluxes across the control volume faces. With an element-based structure [4–6], the local variation of a dependent variable inside a control volume is described by a piecewise polynomial function. This formulation is similar to the FEM approach where the solution within each element is described by local 'shape functions'. To make a FVM free of shape and internal angle restrictions, and to allow hybrid meshes to be used in the formulation, any use of shape functions should be avoided, meaning that the data structure used should be edge-based [7]. Certainly the use of such a data structure within a vertex-centred formulation is computationally more efficient in terms of CPU time and memory

* Corresponding author. Tel.: +61 2 93518176; fax: +61 2 93527060.
E-mail address: shicheng.xue@sydney.edu.au (S.-C. Xue).

0045-7825/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cma.2012.09.016

Nomenclature

A	area
$\mathbf{A}$	area vector (or coefficient matrix)
b	source term for a scalar equation
$\mathbf{b}$	source term vector for a vector equation
$\mathbf{B}$	source term vector for a linear system
d	distance from a cell centre to a vertex
ds	distance from cell centroid to the midpoint of a face
$\hat{d}$	unit vector along a connecting line
$\mathbf{D}$	vector along a connecting line
F	flux
$\mathbf{F}$	flux vector
$\hat{\mathbf{n}}$	unit vector normal to a surface
H	height
$\mathbf{I}$	unit tensor
m	mass flow rate
$\max(x,y)$	maximum value in x and y
$\min(x,y)$	minimum value in x and y
N	total number of control volumes (or cells) in the solution domain
NB	neighbouring cell centroid
p	isotropic pressure
P	cell centroid
R	residual
R_e	Reynolds number
$\mathbf{r}$	distance vector
S	volumetric source term for a scalar equation
$\mathbf{S}$	volumetric source term vector for a vector equation
$\mathbf{T}$	tangential vector to a face
$\hat{\mathbf{t}}$	unit vector tangential to a surface
t	time
U	speed
$\mathbf{V}$	velocity vector
V	velocity component
x, y	Cartesian coordinates

Greek symbols

α, γ	scaling factors
ΔM	mass residual
δ	difference
Φ	dependent variable
Ψ	position-dependent variable
Γ	diffusive coefficient
κ	blending factor
μ	fluid viscosity
Λ	convective coefficient
θ	volume fraction
ρ	fluid density
$\boldsymbol{\tau}$	stress tensor
Ω	volume
ω	under-relaxation factor

Subscripts

bd	boundary face designator
$f\text{-}f'$	from point f to point f'
f	midpoint of a face
f'	intersection point of a connecting line with a face
Φ	dependent variable
i	coordinate index
L	left-hand side
max	maximum value
n	normal direction
NB	neighbouring cell designator
nd	node (or vertex)
P	cell designator
R	right-hand side
t	tangential direction

Superscripts

bd	total number of boundary faces for a control volume (or cell)
0	previous time value
#	intermediate value
$*$	previous iteration
c	convection
d	diffusion
FO	first-order upwind differencing scheme
HO	higher-order scheme
k	cell index
n	normal direction
NC	total number of neighbouring control volumes (or cells)
nd	node or vertex
nf	total number of faces bounding a control volume (or cell)
SO	second-order linear interpolation
T	transpose operation
V	volumetric

usage [8], although there are challenges: the control volumes constructed around the vertices are usually larger than the cells, while often flux evaluation for each edge is simplified using an approach based on the normal vector to the edge corresponding to the centroid dual. As a result, the local truncation error is usually larger [9] than that for a cell-centred formulation [11] where the cells themselves, formed by the grid-generation process, are chosen as the control volumes and dependent variables are solved for (and stored) at the cell's geometric centre. In some situations, there is the additional drawback that a vertex can become a geometrical singularity, where mathematically the values of some field variables are not uniquely defined.

With a cell-centred formulation [10–14], geometrical singularities can be avoided, while construction of dual control volumes is unnecessary. In addition, if the mesh is orthogonal, such an approach is computationally efficient because low-order finite difference approximations along the straight line connecting adjacent

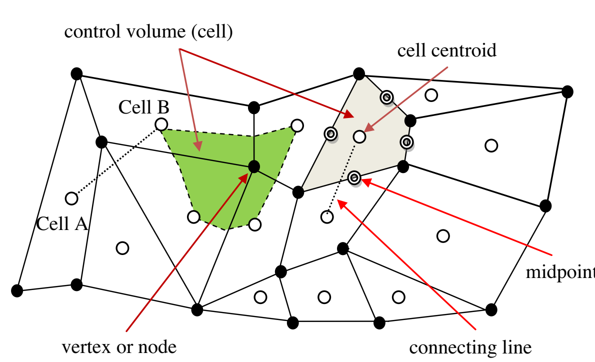

Fig. 1. Schematic representation of an unstructured, hybrid mesh for a 2D discretisation.