OpenFOAM

The Open Source CFD Toolbox

Programmer's Guide

Version 1.7.1
25th August 2010

Copyright © 2000, 2001, 2002, 2003, 2004, 2005, 2006, 2007, 2008, 2009, 2010 OpenCFD Limited.

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.2 published by the Free Software Foundation; with no Invariant Sections, no Back-Cover Texts and one Front-Cover Text: "Available free from openfoam.org." A copy of the license is included in the section entitled "GNU Free Documentation License".

This document is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.

Typeset in LaTeX.

GNU Free Documentation License

Version 1.2, November 2002
Copyright ©2000,2001,2002 Free Software Foundation, Inc.

59 Temple Place, Suite 330, Boston, MA 02111-1307 USA

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

Preamble

The purpose of this License is to make a manual, textbook, or other functional and useful document "free" in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or noncommercially. Secondarily, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of "copyleft", which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License in order to use it for manuals for free software, because free software needs free documentation: a free program should come with manuals providing the same freedoms that the software does. But this License is not limited to software manuals; it can be used for any textual work, regardless of subject matter or whether it is published as a printed book. We recommend this License principally for works whose purpose is instruction or reference.

1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work, in any medium, that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. Such a notice grants a world-wide, royalty-free license, unlimited in duration, to use that work under the conditions stated herein. The **"Document"**, below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as **"you"**. You accept the license if you copy, modify or distribute the work in a way requiring permission under copyright law.

A **"Modified Version"** of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A **"Secondary Section"** is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document's overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (Thus, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The **"Invariant Sections"** are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released under this License. If a section does not fit the above definition of Secondary then it is not allowed to be designated as Invariant. The Document may contain zero Invariant Sections. If the Document does not identify any Invariant Sections then there are none.

The **"Cover Texts"** are certain short passages of text that are listed, as Front-Cover Texts or Back-Cover Texts, in the notice that says that the Document is released under this License. A Front-Cover Text may be at most 5 words, and a Back-Cover Text may be at most 25 words.

A **"Transparent"** copy of the Document means a machine-readable copy, represented in a format whose specification is available to the general public, that is suitable for revising the document straightforwardly with generic text editors or (for images composed of pixels) generic paint programs or (for drawings) some widely available drawing editor, and that is suitable for input to text formatters or for automatic translation to a variety of formats suitable for input to text formatters. A copy made in an otherwise Transparent file format whose markup, or absence of markup, has been arranged to thwart or discourage subsequent modification by readers is not Transparent. An image format is not Transparent if used for any substantial amount of text. A copy that is not "Transparent" is called **"Opaque"**.

Examples of suitable formats for Transparent copies include plain ASCII without markup, Texinfo input format, LaTeX input format, SGML or XML using a publicly available DTD, and standard-conforming simple HTML, PostScript or PDF designed for human modification. Examples of transparent image formats include PNG, XCF and JPG. Opaque formats include proprietary formats that can be read and edited only by proprietary word processors, SGML or XML for which the DTD and/or processing tools are not generally available, and the machine-generated HTML, PostScript or PDF produced by some word processors for output purposes only.

The **"Title Page"** means, for a printed book, the title page itself, plus such following pages as are needed to hold, legibly, the material this License requires to appear in the title page. For works in formats which do not have any title page as such, "Title Page" means the text near the most prominent appearance of the work's title, preceding the beginning of the body of the text.

A section **"Entitled XYZ"** means a named subunit of the Document whose title either is precisely XYZ or contains XYZ in parentheses following text that translates XYZ in another language. (Here XYZ stands for a specific section name mentioned below, such as **"Acknowledgements"**, **"Dedications"**, **"Endorsements"**, or **"History"**.) To **"Preserve the Title"** of such a section when you modify the Document means that it remains a section "Entitled XYZ" according to this definition.

The Document may include Warranty Disclaimers next to the notice which states that this License applies to the Document. These Warranty Disclaimers are considered to be included by reference in this License, but only as regards disclaiming warranties: any other implication that these Warranty Disclaimers may have is void and has no effect on the meaning of this License.

2. VERBATIM COPYING

You may copy and distribute the Document in any medium, either commercially or noncommercially, provided that this License, the copyright notices, and the license notice saying this License applies to the Document are reproduced in all copies, and that you add no other conditions whatsoever to those of this License. You may not use technical measures to obstruct or control the reading or further copying of the copies you make or distribute. However, you may accept compensation in exchange for copies. If you distribute a large enough number of copies you must also follow the conditions in section 3.

You may also lend copies, under the same conditions stated above, and you may publicly display copies.

3. COPYING IN QUANTITY

If you publish printed copies (or copies in media that commonly have printed covers) of the Document, numbering more than 100, and the Document's license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on

the covers in addition. Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a computer-network location from which the general network-using public has access to download using public-standard network protocols a complete Transparent copy of the Document, free of added material. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any, be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission.

B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has fewer than five), unless they release you from this requirement.

C. State on the Title page the name of the publisher of the Modified Version, as the publisher.

D. Preserve all the copyright notices of the Document.

E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices.

F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below.

G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice.

H. Include an unaltered copy of this License.

I. Preserve the section Entitled "History", Preserve its Title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section Entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence.

J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission.

K. For any section Entitled "Acknowledgements" or "Dedications", Preserve the Title of the section, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein.

L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles.

M. Delete any section Entitled "Endorsements". Such a section may not be included in the Modified Version.

N. Do not retitle any existing section to be Entitled "Endorsements" or to conflict in title with any Invariant Section.

O. Preserve any Warranty Disclaimers.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their titles to the list of Invariant Sections in the Modified Version's license notice. These titles must be distinct from any other section titles.

You may add a section Entitled "Endorsements", provided it contains nothing but endorsements of your Modified Version by various parties–for example, statements of peer review or that the text has been approved by an organization as the authoritative definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be added by (or through arrangements made by) any one entity. If the Document already includes a cover text for the same cover, previously added by you or by arrangement made by the same entity you are acting on behalf of, you may not add another; but you may replace the old one, on explicit permission from the previous publisher that added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission to use their names for publicity for or to assert or imply endorsement of any Modified Version.

5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License, under the terms defined in section 4 above for modified versions, provided that you include in the combination all of the Invariant Sections of all of the original documents, unmodified, and list them all as Invariant Sections of your combined work in its license notice, and that you preserve all their Warranty Disclaimers.

The combined work need only contain one copy of this License, and multiple identical Invariant Sections may be replaced with a single copy. If there are multiple Invariant Sections with the same

name but different contents, make the title of each such section unique by adding at the end of it, in parentheses, the name of the original author or publisher of that section if known, or else a unique number. Make the same adjustment to the section titles in the list of Invariant Sections in the license notice of the combined work.

In the combination, you must combine any sections Entitled "History" in the various original documents, forming one section Entitled "History"; likewise combine any sections Entitled "Acknowledgements", and any sections Entitled "Dedications". You must delete all sections Entitled "Endorsements".

6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released under this License, and replace the individual copies of this License in the various documents with a single copy that is included in the collection, provided that you follow the rules of this License for verbatim copying of each of the documents in all other respects.

You may extract a single document from such a collection, and distribute it individually under this License, provided you insert a copy of this License into the extracted document, and follow this License in all other respects regarding verbatim copying of that document.

7. AGGREGATION WITH INDEPENDENT WORKS

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, is called an "aggregate" if the copyright resulting from the compilation is not used to limit the legal rights of the compilation's users beyond what the individual works permit. When the Document is included in an aggregate, this License does not apply to the other works in the aggregate which are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one half of the entire aggregate, the Document's Cover Texts may be placed on covers that bracket the Document within the aggregate, or the electronic equivalent of covers if the Document is in electronic form. Otherwise they must appear on printed covers that bracket the whole aggregate.

8. TRANSLATION

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License, and all the license notices in the Document, and any Warranty Disclaimers, provided that you also include the original English version of this License and the original versions of those notices and disclaimers. In case of a disagreement between the translation and the original version of this License or a notice or disclaimer, the original version will prevail.

If a section in the Document is Entitled "Acknowledgements", "Dedications", or "History", the requirement (section 4) to Preserve its Title (section 1) will typically require changing the actual title.

9. TERMINATION

You may not copy, modify, sublicense, or distribute the Document except as expressly provided for under this License. Any other attempt to copy, modify, sublicense or distribute the Document is void, and will automatically terminate your rights under this License. However, parties who have

received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

10. FUTURE REVISIONS OF THIS LICENSE

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See http://www.gnu.org/copyleft/.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License "or any later version" applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation.

Trademarks

ANSYS is a registered trademark of ANSYS Inc.
CFX is a registered trademark of Ansys Inc.
CHEMKIN is a registered trademark of Reaction Design Corporation
EnSight is a registered trademark of Computational Engineering International Ltd.
Fieldview is a registered trademark of Intelligent Light
Fluent is a registered trademark of Ansys Inc.
GAMBIT is a registered trademark of Ansys Inc.
Icem-CFD is a registered trademark of Ansys Inc.
I-DEAS is a registered trademark of Structural Dynamics Research Corporation
JAVA is a registered trademark of Sun Microsystems Inc.
Linux is a registered trademark of Linus Torvalds
OpenFOAM is a registered trademark of OpenCFD Ltd
ParaView is a registered trademark of Kitware
STAR-CD is a registered trademark of Computational Dynamics Ltd.
UNIX is a registered trademark of The Open Group

Contents

Copyright Notice **P-2**

GNU Free Documentation Licence **P-3**
1. APPLICABILITY AND DEFINITIONS . . . P-3
2. VERBATIM COPYING . . . P-4
3. COPYING IN QUANTITY . . . P-4
4. MODIFICATIONS . . . P-5
5. COMBINING DOCUMENTS . . . P-6
6. COLLECTIONS OF DOCUMENTS . . . P-7
7. AGGREGATION WITH INDEPENDENT WORKS . . . P-7
8. TRANSLATION . . . P-7
9. TERMINATION . . . P-7
10. FUTURE REVISIONS OF THIS LICENSE . . . P-8

Trademarks **P-9**

Contents **P-11**

1 Tensor mathematics **P-15**
1.1 Coordinate system . . . P-15
1.2 Tensors . . . P-15
1.2.1 Tensor notation . . . P-17
1.3 Algebraic tensor operations . . . P-18
1.3.1 The inner product . . . P-18
1.3.2 The double inner product of two tensors . . . P-19
1.3.3 The triple inner product of two third rank tensors . . . P-19
1.3.4 The outer product . . . P-19
1.3.5 The cross product of two vectors . . . P-20
1.3.6 Other general tensor operations . . . P-20
1.3.7 Geometric transformation and the identity tensor . . . P-21
1.3.8 Useful tensor identities . . . P-21
1.3.9 Operations exclusive to tensors of rank 2 . . . P-22
1.3.10 Operations exclusive to scalars . . . P-23
1.4 OpenFOAM tensor classes . . . P-23
1.4.1 Algebraic tensor operations in OpenFOAM . . . P-24
1.5 Dimensional units . . . P-26

2 Discretisation procedures **P-27**
2.1 Differential operators . . . P-27
2.1.1 Gradient . . . P-27
2.1.2 Divergence . . . P-28
2.1.3 Curl . . . P-28
2.1.4 Laplacian . . . P-28
2.1.5 Temporal derivative . . . P-28
2.2 Overview of discretisation . . . P-29
2.2.1 OpenFOAM lists and fields . . . P-29
2.3 Discretisation of the solution domain . . . P-30
2.3.1 Defining a mesh in OpenFOAM . . . P-31
2.3.2 Defining a geometricField in OpenFOAM . . . P-32
2.4 Equation discretisation . . . P-33
2.4.1 The Laplacian term . . . P-38
2.4.2 The convection term . . . P-38
2.4.3 First time derivative . . . P-39
2.4.4 Second time derivative . . . P-39
2.4.5 Divergence . . . P-39
2.4.6 Gradient . . . P-40
2.4.7 Grad-grad squared . . . P-41
2.4.8 Curl . . . P-41
2.4.9 Source terms . . . P-41
2.4.10 Other explicit discretisation schemes . . . P-41
2.5 Temporal discretisation . . . P-42
2.5.1 Treatment of temporal discretisation in OpenFOAM . . . P-43
2.6 Boundary Conditions . . . P-43
2.6.1 Physical boundary conditions . . . P-44

3 Examples of the use of OpenFOAM **P-45**
3.1 Flow around a cylinder . . . P-45
3.1.1 Problem specification . . . P-46
3.1.2 Note on potentialFoam . . . P-47
3.1.3 Mesh generation . . . P-47
3.1.4 Boundary conditions and initial fields . . . P-49
3.1.5 Running the case . . . P-49
3.1.6 Generating the analytical solution . . . P-50
3.1.7 Exercise . . . P-54
3.2 Steady turbulent flow over a backward-facing step . . . P-54
3.2.1 Problem specification . . . P-54
3.2.2 Mesh generation . . . P-55
3.2.3 Boundary conditions and initial fields . . . P-58
3.2.4 Case control . . . P-58
3.2.5 Running the case and post-processing . . . P-59
3.3 Supersonic flow over a forward-facing step . . . P-59
3.3.1 Problem specification . . . P-60
3.3.2 Mesh generation . . . P-61
3.3.3 Running the case . . . P-62
3.3.4 Exercise . . . P-62

3.4 Decompression of a tank internally pressurised with water P-63
3.4.1 Problem specification P-63
3.4.2 Mesh Generation P-65
3.4.3 Preparing the Run P-66
3.4.4 Running the case P-67
3.4.5 Improving the solution by refining the mesh P-68
3.5 Magnetohydrodynamic flow of a liquid P-69
3.5.1 Problem specification P-69
3.5.2 Mesh generation P-70
3.5.3 Running the case P-71

Index **P-73**

Chapter 1

Tensor mathematics

This Chapter describes tensors and their algebraic operations and how they are represented in mathematical text in this book. It then explains how tensors and tensor algebra are programmed in OpenFOAM.

1.1 Coordinate system

OpenFOAM is primarily designed to solve problems in continuum mechanics, *i.e.* the branch of mechanics concerned with the stresses in solids, liquids and gases and the deformation or flow of these materials. OpenFOAM is therefore based in 3 dimensional space and time and deals with physical entities described by tensors. The coordinate system used by OpenFOAM is the right-handed rectangular Cartesian axes as shown in Figure 1.1. This system of axes is constructed by defining an origin O from which three lines are drawn at right angles to each other, termed the Ox, Oy, Oz axes. A right-handed set of axes is defined such that to an observer looking down the Oz axis (with O nearest them), the arc from a point on the Ox axis to a point on the Oy axis is in a clockwise sense.

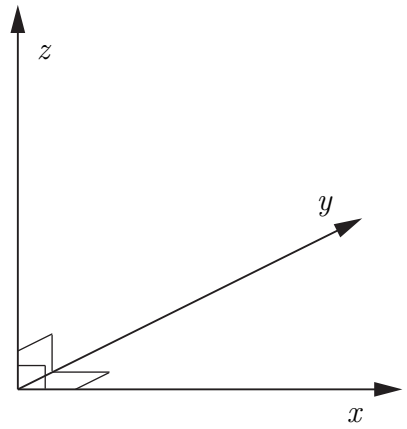


Figure 1.1: Right handed axes

1.2 Tensors

The term tensor describes an entity that belongs to a particular space and obeys certain mathematical rules. Briefly, tensors are represented by a set of *component values* relating

to a set of unit base vectors; in OpenFOAM the unit base vectors $\mathbf{i}_x$, $\mathbf{i}_y$ and $\mathbf{i}_z$ are aligned with the right-handed rectangular Cartesian axes x, y and z respectively. The base vectors are therefore orthogonal, *i.e.* at right-angles to one another. Every tensor has the following attributes:

Dimension d of the particular space to which they belong, *i.e.* $d = 3$ in OpenFOAM;

Rank An integer $r \geq 0$, such that the number of component values $= d^r$.

While OpenFOAM 1.x is set to 3 dimensions, it offers tensors of ranks 0 to 3 as standard while being written in such a way to allow this basic set of ranks to be extended indefinitely. Tensors of rank 0 and 1, better known as scalars and vectors, should be familiar to readers; tensors of rank 2 and 3 may not be so familiar. For completeness all ranks of tensor offered as standard in OpenFOAM 1.x are reviewed below.

Rank 0 'scalar' Any property which can be represented by a single real number, denoted by characters in italics, *e.g.* mass m, volume V, pressure p and viscosity μ.

Rank 1 'vector' An entity which can be represented physically by both magnitude and direction. In component form, the vector $\mathbf{a} = (a_1, a_2, a_3)$ relates to a set of Cartesian axes x, y, z respectively. The *index notation* presents the same vector as a_i, $i = 1, 2, 3$, although the list of indices $i = 1, 2, 3$ will be omitted in this book, as it is intuitive since we are always dealing with 3 dimensions.

Rank 2 'tensor' or second rank tensor, $\mathbf{T}$ has 9 components which can be expressed in array notation as:

$$\mathbf{T} = T_{ij} = \begin{pmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{pmatrix} \tag{1.1}$$

The components T_{ij} are now represented using 2 indices since $r = 2$ and the list of indices $i, j = 1, 2, 3$ is omitted as before. The components for which $i = j$ are referred to as the diagonal components, and those for which $i \neq j$ are referred to as the off-diagonal components. The *transpose* of $\mathbf{T}$ is produced by exchanging components across the diagonal such that

$$\mathbf{T}^{\mathrm{T}} = T_{ji} = \begin{pmatrix} T_{11} & T_{21} & T_{31} \\ T_{12} & T_{22} & T_{32} \\ T_{13} & T_{23} & T_{33} \end{pmatrix} \tag{1.2}$$

Note: a rank 2 tensor is often colloquially termed 'tensor' since the occurrence of higher order tensors is fairly rare.

Symmetric rank 2 The term 'symmetric' refers to components being symmetric about the diagonal, *i.e.* $T_{ij} = T_{ji}$. In this case, there are only 6 independent components since $T_{12} = T_{21}$, $T_{13} = T_{31}$ and $T_{23} = T_{32}$. OpenFOAM distinguishes between symmetric and non-symmetric tensors to save memory by storing 6 components rather than 9 if the tensor is symmetric. Most tensors encountered in continuum mechanics are symmetric.

Rank 3 has 27 components and is represented in index notation as P_{ijk} which is too long to represent in array notation as in Equation 1.1.

Symmetric rank 3 Symmetry of a rank 3 tensor is defined in OpenFOAM to mean that $P_{ijk} = P_{ikj} = P_{jik} = P_{jki} = P_{kij} = P_{kji}$ and therefore has 10 independent components. More specifically, it is formed by the outer product of 3 identical vectors, where the outer product operation is described in Section 1.3.4.

1.2.1 Tensor notation

This is a book on computational continuum mechanics that deals with problems involving complex PDEs in 3 spatial dimensions and in time. It is vital from the beginning to adopt a notation for the equations which is compact yet unambiguous. To make the equations easy to follow, we must use a notation that encapsulates the idea of a tensor as an entity in the own right, rather than a list of scalar components. Additionally, any tensor operation should be perceived as an operation on the entire tensor entity rather than a series of operations on its components.

Consequently, in this book the *tensor notation* is preferred in which any tensor of rank 1 and above, *i.e.* all tensors other than scalars, are represented by letters in bold face, *e.g.* **a**. This actively promotes the concept of a tensor as a entity in its own right since it is denoted by a single symbol, and it is also extremely compact. The potential drawback is that the rank of a bold face symbol is not immediately apparent, although it is clearly not zero. However, in practice this presents no real problem since we are aware of the property each symbol represents and therefore intuitively know its rank, *e.g.* we know velocity **U** is a tensor of rank 1.

A further, more fundamental idea regarding the choice of notation is that the mathematical representation of a tensor should not change depending on our coordinate system, *i.e.* the vector **a**is the same vector irrespective of where we view it from. The tensor notation supports this concept as it implies nothing about the coordinate system. However, other notations, *e.g.* a_i, expose the individual components of the tensor which naturally implies the choice of coordinate system. The unsatisfactory consequence of this is that the tensor is then represented by a set of values which are not unique — they depend on the coordinate system.

That said, the index notation, introduced in Section 1.2, is adopted from time to time in this book mainly to expand tensor operations into the constituent components. When using the index notation, we adopt the *summation convention* which states that whenever the same letter subscript occurs twice in a term, the that subscript is to be given all values, *i.e.* $1, 2, 3$, and the results added together, *e.g.*

$$a_i b_i = \sum_{i=1}^{3} a_i b_i = a_1 b_1 + a_2 b_2 + a_3 b_3 \tag{1.3}$$

In the remainder of the book the symbol $\sum$ is omitted since the repeated subscript indicates the summation.

1.3 Algebraic tensor operations

This section describes all the algebraic operations for tensors that are available in OpenFOAM. Let us first review the most simple tensor operations: addition, subtraction, and scalar multiplication and division. Addition and subtraction are both commutative and associative and are only valid between tensors of the same rank. The operations are performed by addition/subtraction of respective components of the tensors, *e.g.* the subtraction of two vectors $\mathbf{a}$ and $\mathbf{b}$ is

$$\mathbf{a} - \mathbf{b} = a_i - b_i = (a_1 - b_1, a_2 - b_2, a_3 - b_3) \tag{1.4}$$

Multiplication of any tensor $\mathbf{a}$ by a scalar s is also commutative and associative and is performed by multiplying all the tensor components by the scalar. For example,

$$s\mathbf{a} = sa_i = (sa_1, sa_2, sa_3) \tag{1.5}$$

Division between a tensor $\mathbf{a}$ and a scalar is only relevant when the scalar is the second argument of the operation, *i.e.*

$$\mathbf{a}/s = a_i/s = (a_1/s, a_2/s, a_3/s) \tag{1.6}$$

Following these operations are a set of more complex products between tensors of rank 1 and above, described in the following Sections.

1.3.1 The inner product

The inner product operates on any two tensors of rank r_1 and r_2 such that the rank of the result $r = r_1 + r_2 - 2$. Inner product operations with tensors up to rank 3 are described below:

- The inner product of two vectors $\mathbf{a}$ and $\mathbf{b}$ is commutative and produces a scalar $s = \mathbf{a} \cdot \mathbf{b}$ where

$$s = a_i b_i = a_1 b_1 + a_2 b_2 + a_3 b_3 \tag{1.7}$$

- The inner product of a tensor $\mathbf{T}$ and vector $\mathbf{a}$ produces a vector $\mathbf{b} = \mathbf{T} \cdot \mathbf{a}$, represented below as a column array for convenience

$$b_i = T_{ij} a_j = \begin{pmatrix} T_{11}a_1 + T_{12}a_2 + T_{13}a_3 \\ T_{21}a_1 + T_{22}a_2 + T_{23}a_3 \\ T_{31}a_1 + T_{32}a_2 + T_{33}a_3 \end{pmatrix} \tag{1.8}$$

  It is non-commutative if $\mathbf{T}$ is non-symmetric such that $\mathbf{b} = \mathbf{a} \cdot \mathbf{T} = \mathbf{T}^{\mathrm{T}} \cdot \mathbf{a}$ is

$$b_i = a_j T_{ji} = \begin{pmatrix} a_1T_{11} + a_2T_{21} + a_3T_{31} \\ a_1T_{12} + a_2T_{22} + a_3T_{32} \\ a_1T_{13} + a_2T_{23} + a_3T_{33} \end{pmatrix} \tag{1.9}$$

- The inner product of two tensors $\mathbf{T}$ and $\mathbf{S}$ produces a tensor $\mathbf{P} = \mathbf{T} \cdot \mathbf{S}$ whose components are evaluated as:

$$P_{ij} = T_{ik} S_{kj} \tag{1.10}$$

  It is non-commutative such that $\mathbf{T} \cdot \mathbf{S} = \left(\mathbf{S}^{\mathrm{T}} \cdot \mathbf{T}^{\mathrm{T}}\right)^{\mathrm{T}}$

- The inner product of a vector $\mathbf{a}$ and third rank tensor $\mathbf{P}$ produces a second rank tensor $\mathbf{T} = \mathbf{a} \cdot \mathbf{P}$ whose components are

$$T_{ij} = a_k P_{kij} \tag{1.11}$$

  Again this is non-commutative so that $\mathbf{T} = \mathbf{P} \cdot \mathbf{a}$ is

$$T_{ij} = P_{ijk} a_k \tag{1.12}$$

- The inner product of a second rank tensor $\mathbf{T}$ and third rank tensor $\mathbf{P}$ produces a third rank tensor $\mathbf{Q} = \mathbf{T} \cdot \mathbf{P}$ whose components are

$$Q_{ijk} = T_{il} P_{ljk} \tag{1.13}$$

  Again this is non-commutative so that $\mathbf{Q} = \mathbf{P} \cdot \mathbf{T}$ is

$$Q_{ijk} = P_{ijl} T_{lk} \tag{1.14}$$

1.3.2 The double inner product of two tensors

The double inner product of two second-rank tensors $\mathbf{T}$ and $\mathbf{S}$ produces a scalar $s = \mathbf{T} : \mathbf{S}$ which can be evaluated as the sum of the 9 products of the tensor components

$$\begin{aligned} s = T_{ij} S_{ij} = \; & T_{11} S_{11} + T_{12} S_{12} + T_{13} S_{13} + \\ & T_{21} S_{21} + T_{22} S_{22} + T_{23} S_{23} + \\ & T_{31} S_{31} + T_{32} S_{32} + T_{33} S_{33} \end{aligned} \tag{1.15}$$

The double inner product between a second rank tensor $\mathbf{T}$ and third rank tensor $\mathbf{P}$ produces a vector $\mathbf{a} = \mathbf{T} : \mathbf{P}$ with components

$$a_i = T_{jk} P_{jki} \tag{1.16}$$

This is non-commutative so that $\mathbf{a} = \mathbf{P} : \mathbf{T}$ is

$$a_i = P_{ijk} T_{jk} \tag{1.17}$$

1.3.3 The triple inner product of two third rank tensors

The triple inner product of two third rank tensors $\mathbf{P}$ and $\mathbf{Q}$ produces a scalar $s = \mathbf{P} \vdots \mathbf{Q}$ which can be evaluated as the sum of the 27 products of the tensor components

$$s = P_{ijk} Q_{ijk} \tag{1.18}$$

1.3.4 The outer product

The outer product operates between vectors and tensors as follows:

- The outer product of two vectors $\mathbf{a}$ and $\mathbf{b}$ is non-commutative and produces a tensor $\mathbf{T} = \mathbf{ab} = (\mathbf{ba})^{\mathrm{T}}$ whose components are evaluated as:

$$T_{ij} = a_i b_j = \begin{pmatrix} a_1 b_1 & a_1 b_2 & a_1 b_3 \\ a_2 b_1 & a_2 b_2 & a_2 b_3 \\ a_3 b_1 & a_3 b_2 & a_3 b_3 \end{pmatrix} \tag{1.19}$$

- An outer product of a vector $\mathbf{a}$ and second rank tensor $\mathbf{T}$ produces a third rank tensor $\mathbf{P} = \mathbf{aT}$ whose components are

$$P_{ijk} = a_i T_{jk} \tag{1.20}$$

This is non-commutative so that $\mathbf{P} = \mathbf{T}\,\mathbf{a}$ produces

$$P_{ijk} = T_{ij} a_k \tag{1.21}$$

1.3.5 The cross product of two vectors

The cross product operation is exclusive to vectors only. For two vectors $\mathbf{a}$ with $\mathbf{b}$, it produces a vector $\mathbf{c} = \mathbf{a} \times \mathbf{b}$ whose components are

$$c_i = e_{ijk} a_j b_k = (a_2 b_3 - a_3 b_2, a_3 b_1 - a_1 b_3, a_1 b_2 - a_2 b_1) \tag{1.22}$$

where the *permutation symbol* is defined by

$$e_{ijk} = \begin{cases} 0 & \text{when any two indices are equal} \\ +1 & \text{when } i,j,k \text{ are an even permutation of 1,2,3} \\ -1 & \text{when } i,j,k \text{ are an odd permutation of 1,2,3} \end{cases} \tag{1.23}$$

in which the even permutations are 123, 231 and 312 and the odd permutations are 132, 213 and 321.

1.3.6 Other general tensor operations

Some less common tensor operations and terminology used by OpenFOAM are described below.

Square of a tensor is defined as the outer product of the tensor with itself, *e.g.* for a vector $\mathbf{a}$, the square $\mathbf{a}^2 = \mathbf{aa}$.

***n*th power** of a tensor is evaluated by n outer products of the tensor, *e.g.* for a vector $\mathbf{a}$, the 3rd power $\mathbf{a}^3 = \mathbf{aaa}$.

Magnitude squared of a tensor is the rth inner product of the tensor of rank r with itself, to produce a scalar. For example, for a second rank tensor $\mathbf{T}$, $|\mathbf{T}|^2 = \mathbf{T} : \mathbf{T}$.

Magnitude is the square root of the magnitude squared, *e.g.* for a tensor $\mathbf{T}$, $|\mathbf{T}| = \sqrt{\mathbf{T} : \mathbf{T}}$. Vectors of unit magnitude are referred to as *unit vectors*.

Component maximum is the component of the tensor with greatest value, inclusive of sign, *i.e.* not the largest magnitude.

Component minimum is the component of the tensor with smallest value.

Component average is the mean of all components of a tensor.

Scale As the name suggests, the scale function is a tool for scaling the components of one tensor by the components of another tensor of the same rank. It is evaluated as the product of corresponding components of 2 tensors, *e.g.*, scaling vector $\mathbf{a}$ by vector $\mathbf{b}$ would produce vector $\mathbf{c}$ whose components are

$$c_i = \text{scale}(\mathbf{a}, \mathbf{b}) = (a_1 b_1, a_2 b_2, a_3 b_3) \tag{1.24}$$

1.3.7 Geometric transformation and the identity tensor

A second rank tensor $\mathbf{T}$ is strictly defined as a linear vector function, i.e. it is a function which associates an argument vector $\mathbf{a}$ to another vector $\mathbf{b}$ by the inner product $\mathbf{b} = \mathbf{T} \cdot \mathbf{a}$. The components of $\mathbf{T}$ can be chosen to perform a specific geometric transformation of a tensor from the x, y, z coordinate system to a new coordinate system x^*, y^*, z^*; $\mathbf{T}$ is then referred to as the *transformation tensor*. While a scalar remains unchanged under a transformation, the vector $\mathbf{a}$ is transformed to $\mathbf{a}^*$ by

$$\mathbf{a}^* = \mathbf{T} \cdot \mathbf{a} \tag{1.25}$$

A second rank tensor $\mathbf{S}$ is transformed to $\mathbf{S}^*$ according to

$$\mathbf{S}^* = \mathbf{T} \cdot \mathbf{S} \cdot \mathbf{T}^{\mathrm{T}} \tag{1.26}$$

The *identity tensor* $\mathbf{I}$ is defined by the requirement that it transforms another tensor onto itself. For all vectors $\mathbf{a}$

$$\mathbf{a} = \mathbf{I} \cdot \mathbf{a} \tag{1.27}$$

and therefore

$$\mathbf{I} = \delta_{ij} = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix} \tag{1.28}$$

where δ_{ij} is known as the *Kronecker delta* symbol.

1.3.8 Useful tensor identities

Several identities are listed below which can be verified by under the assumption that all the relevant derivatives exist and are continuous. The identities are expressed for scalar s and vector $\mathbf{a}$.

$$\begin{aligned}
&\nabla \cdot (\nabla \times \mathbf{a}) \equiv 0 \\
&\nabla \times (\nabla s) \equiv \mathbf{0} \\
&\nabla \cdot (s\mathbf{a}) \equiv s\nabla \cdot \mathbf{a} + \mathbf{a} \cdot \nabla s \\
&\nabla \times (s\mathbf{a}) \equiv s\nabla \times \mathbf{a} + \nabla s \times \mathbf{a} \\
&\nabla(\mathbf{a} \cdot \mathbf{b}) \equiv \mathbf{a} \times (\nabla \times \mathbf{b}) + \mathbf{b} \times (\nabla \times \mathbf{a}) + (\mathbf{a} \cdot \nabla)\mathbf{b} + (\mathbf{b} \cdot \nabla)\mathbf{a} \\
&\nabla \cdot (\mathbf{a} \times \mathbf{b}) \equiv \mathbf{b} \cdot (\nabla \times \mathbf{a}) - \mathbf{a} \cdot (\nabla \times \mathbf{b}) \\
&\nabla \times (\mathbf{a} \times \mathbf{b}) \equiv \mathbf{a}(\nabla \cdot \mathbf{b}) - \mathbf{b}(\nabla \cdot \mathbf{a}) + (\mathbf{b} \cdot \nabla)\mathbf{a} - (\mathbf{a} \cdot \nabla)\mathbf{b} \\
&\nabla \times (\nabla \times \mathbf{a}) \equiv \nabla(\nabla \cdot \mathbf{a}) - \nabla^2\mathbf{a} \\
&(\nabla \times \mathbf{a}) \times \mathbf{a} \equiv \mathbf{a} \cdot (\nabla\mathbf{a}) - \nabla(\mathbf{a} \cdot \mathbf{a})
\end{aligned} \tag{1.29}$$

It is sometimes useful to know the $e - \delta$ identity to help to manipulate equations in index notation:

$$e_{ijk}e_{irs} = \delta_{jr}\delta_{ks} - \delta_{js}\delta_{kr} \tag{1.30}$$

1.3.9 Operations exclusive to tensors of rank 2

There are several operations that manipulate the components of tensors of rank 2 that are listed below:

Transpose of a tensor $\mathbf{T} = T_{ij}$ is $\mathbf{T}^{\mathrm{T}} = T_{ji}$ as described in Equation 1.2.

Symmetric and skew (antisymmetric) tensors As discussed in section 1.2, a tensor is said to be symmetric if its components are symmetric about the diagonal, i.e. $\mathbf{T} = \mathbf{T}^{\mathrm{T}}$. A skew or antisymmetric tensor has $\mathbf{T} = -\mathbf{T}^{\mathrm{T}}$ which intuitively implies that $T_{11} = T_{22} = T_{33} = 0$. Every second order tensor can be decomposed into symmetric and skew parts by

$$\mathbf{T} = \underbrace{\frac{1}{2}(\mathbf{T} + \mathbf{T}^{\mathrm{T}})}_{symmetric} + \underbrace{\frac{1}{2}(\mathbf{T} - \mathbf{T}^{\mathrm{T}})}_{skew} = \mathrm{symm}\,\mathbf{T} + \mathrm{skew}\,\mathbf{T} \tag{1.31}$$

Trace The trace of a tensor $\mathbf{T}$ is a scalar, evaluated by summing the diagonal components

$$\mathrm{tr}\,\mathbf{T} = T_{11} + T_{22} + T_{33} \tag{1.32}$$

Diagonal returns a vector whose components are the diagonal components of the second rank tensor $\mathbf{T}$

$$\mathrm{diag}\,\mathbf{T} = (T_{11}, T_{22}, T_{33}) \tag{1.33}$$

Deviatoric and hydrostatic tensors Every second rank tensor $\mathbf{T}$ can be decomposed into a deviatoric component, for which $\mathrm{tr}\,\mathbf{T} = 0$ and a hydrostatic component of the form $\mathbf{T} = s\mathbf{I}$ where s is a scalar. Every second rank tensor can be decomposed into deviatoric and hydrostatic parts as follows:

$$\mathbf{T} = \underbrace{\mathbf{T} - \frac{1}{3}(\mathrm{tr}\,\mathbf{T})\,\mathbf{I}}_{deviatoric} + \underbrace{\frac{1}{3}(\mathrm{tr}\,\mathbf{T})\,\mathbf{I}}_{hydrostatic} = \mathrm{dev}\,\mathbf{T} + \mathrm{hyd}\,\mathbf{T} \tag{1.34}$$

Determinant The determinant of a second rank tensor is evaluated by

$$\begin{aligned}\det\mathbf{T} = \begin{vmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{vmatrix} &= \begin{array}{l} T_{11}(T_{22}T_{33} - T_{23}T_{32}) - \\ T_{12}(T_{21}T_{33} - T_{23}T_{31}) + \\ T_{13}(T_{21}T_{32} - T_{22}T_{31}) \end{array} \\ &= \frac{1}{6}e_{ijk}e_{pqr}T_{ip}T_{jq}T_{kr}\end{aligned} \tag{1.35}$$

Cofactors The *minors* of a tensor are evaluated for each component by deleting the row and column in which the component is situated and evaluating the resulting entries as a 2×2 *determinant*. For example, the minor of T_{12} is

$$\begin{vmatrix} \cancel{T_{11}} & \cancel{T_{12}} & \cancel{T_{13}} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{vmatrix} = \begin{vmatrix} T_{21} & T_{23} \\ T_{31} & T_{33} \end{vmatrix} = T_{21}T_{33} - T_{23}T_{31} \tag{1.36}$$

The cofactors are *signed minors* where each minor is component is given a sign based on the rule

$$\begin{array}{l} +\text{ve if } i+j \text{ is even} \\ -\text{ve if } i+j \text{ is odd} \end{array} \tag{1.37}$$

The cofactors of $\mathbf{T}$ can be evaluated as

$$\text{cof}\,\mathbf{T} = \frac{1}{2} e_{jkr} e_{ist} T_{sk} T_{tr} \tag{1.38}$$

Inverse The inverse of a tensor can be evaluated as

$$\text{inv}\,\mathbf{T} = \frac{\text{cof}\,\mathbf{T}^{\text{T}}}{\det \mathbf{T}} \tag{1.39}$$

Hodge dual of a tensor is a vector whose components are

$$*\,\mathbf{T} = (T_{23}, -T_{13}, T_{12}) \tag{1.40}$$

1.3.10 Operations exclusive to scalars

OpenFOAM supports most of the well known functions that operate on scalars, *e.g.* square root, exponential, logarithm, sine, cosine *etc.*., a list of which can be found in Table 1.2. There are 3 additional functions defined within OpenFOAM that are described below:

Sign of a scalar s is

$$\text{sgn}(s) = \begin{cases} 1 & \text{if } s \geq 0, \\ -1 & \text{if } s < 0. \end{cases} \tag{1.41}$$

Positive of a scalar s is

$$\text{pos}(s) = \begin{cases} 1 & \text{if } s \geq 0, \\ 0 & \text{if } s < 0. \end{cases} \tag{1.42}$$

Limit of a scalar s by the scalar n

$$\text{limit}(s, n) = \begin{cases} s & \text{if } s < n, \\ 0 & \text{if } s \geq n. \end{cases} \tag{1.43}$$

1.4 OpenFOAM tensor classes

OpenFOAM contains a C++ class library primitive that contains the classes for the tensor mathematics described so far. The basic tensor classes that are available as standard in OpenFOAM are listed in Table 1.1. The Table also lists the functions that allow the user to access individual components of a tensor, known as access functions.

We can declare the tensor

$$\mathbf{T} = \begin{pmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \\ 7 & 8 & 9 \end{pmatrix} \tag{1.44}$$

in OpenFOAM by the line:

Rank	Common name	Basic class	Access functions
0	Scalar	scalar	
1	Vector	vector	`x()`, `y()`, `z()`
2	Tensor	tensor	`xx()`, `xy()`, `xz()`...

Table 1.1: Basic tensor classes in OpenFOAM

```
tensor T(1, 2, 3, 4, 5, 6, 7, 8, 9);
```

We can then access the component T_{13}, or T_{xz} using the `xz()` access function. For instance the code

```
Info << ''Txz = '' << T.xz() << endl;
```

outputs to the screen:

```
Txz = 3
```

1.4.1 Algebraic tensor operations in OpenFOAM

The algebraic operations described in Section 1.3 are all available to the OpenFOAM tensor classes using syntax which closely mimics the notation used in written mathematics. Some functions are represented solely by descriptive functions, *e.g.*`symm()`, but others can also be executed using symbolic operators, *e.g.*`*`. All functions are listed in Table 1.2.

Operation	Comment	Mathematical Description	Description in OpenFOAM
Addition		$\mathbf{a}+\mathbf{b}$	`a + b`
Subtraction		$\mathbf{a}-\mathbf{b}$	`a - b`
Scalar multiplication		$s\mathbf{a}$	`s * a`
Scalar division		$\mathbf{a}/s$	`a / s`
Outer product	rank $\mathbf{a},\mathbf{b} >= 1$	$\mathbf{ab}$	`a * b`
Inner product	rank $\mathbf{a},\mathbf{b} >= 1$	$\mathbf{a}\cdot\mathbf{b}$	`a & b`
Double inner product	rank $\mathbf{a},\mathbf{b} >= 2$	$\mathbf{a}:\mathbf{b}$	`a && b`
Cross product	rank $\mathbf{a},\mathbf{b} = 1$	$\mathbf{a}\times\mathbf{b}$	`a ^ b`
Square		$\mathbf{a}^2$	`sqr(a)`
Magnitude squared		$\lvert\mathbf{a}\rvert^2$	`magSqr(a)`
Magnitude		$\lvert\mathbf{a}\rvert$	`mag(a)`
Power	$n = 0, 1, ..., 4$	$\mathbf{a}^n$	`pow(a,n)`
Component average	$i = 1, ..., N$	$\overline{a_i}$	`cmptAv(a)`
Component maximum	$i = 1, ..., N$	$\max(a_i)$	`max(a)`
Component minimum	$i = 1, ..., N$	$\min(a_i)$	`min(a)`
Scale		scale($\mathbf{a}$,$\mathbf{b}$)	`scale(a,b)`
Geometric transformation	transforms $\mathbf{a}$ using tensor $\mathbf{T}$		`transform(T,a)`
Operations exclusive to tensors of rank 2			

Continued on next page

Continued from previous page

Operation	Comment	Mathematical Description	Description in OpenFOAM
Transpose		$\mathbf{T}^T$	`T.T()`
Diagonal		$\text{diag}\,\mathbf{T}$	`diag(T)`
Trace		$\text{tr}\,\mathbf{T}$	`tr(T)`
Deviatoric component		$\text{dev}\,\mathbf{T}$	`dev(T)`
Symmetric component		$\text{symm}\,\mathbf{T}$	`symm(T)`
Skew-symmetric component		$\text{skew}\,\mathbf{T}$	`skew(T)`
Determinant		$\det\mathbf{T}$	`det(T)`
Cofactors		$\text{cof}\,\mathbf{T}$	`cof(T)`
Inverse		$\text{inv}\,\mathbf{T}$	`inv(T)`
Hodge dual		$*\,\mathbf{T}$	`*T`
Operations exclusive to scalars			
Sign (boolean)		$\text{sgn}(s)$	`sign(s)`
Positive (boolean)		$s >= 0$	`pos(s)`
Negative (boolean)		$s < 0$	`neg(s)`
Limit	n scalar	$\text{limit}(s, n)$	`limit(s,n)`
Square root		$\sqrt{s}$	`sqrt(s)`
Exponential		$\exp s$	`exp(s)`
Natural logarithm		$\ln s$	`log(s)`
Base 10 logarithm		$\log_{10} s$	`log10(s)`
Sine		$\sin s$	`sin(s)`
Cosine		$\cos s$	`cos(s)`
Tangent		$\tan s$	`tan(s)`
Arc sine		$\text{asin}\, s$	`asin(s)`
Arc cosine		$\text{acos}\, s$	`acos(s)`
Arc tangent		$\text{atan}\, s$	`atan(s)`
Hyperbolic sine		$\sinh s$	`sinh(s)`
Hyperbolic cosine		$\cosh s$	`cosh(s)`
Hyperbolic tangent		$\tanh s$	`tanh(s)`
Hyperbolic arc sine		$\text{asinh}\, s$	`asinh(s)`
Hyperbolic arc cosine		$\text{acosh}\, s$	`acosh(s)`
Hyperbolic arc tangent		$\text{atanh}\, s$	`atanh(s)`
Error function		$\text{erf}\, s$	`erf(s)`
Complement error function		$\text{erfc}\, s$	`erfc(s)`
Logarithm gamma function		$\ln\Gamma s$	`lgamma(s)`
Type 1 Bessel function of order 0		$\text{J}_0\, s$	`j0(s)`
Type 1 Bessel function of order 1		$\text{J}_1\, s$	`j1(s)`
Type 2 Bessel function of order 0		$\text{Y}_0\, s$	`y0(s)`
Type 2 Bessel function of order 1		$\text{Y}_1\, s$	`y1(s)`

$\mathbf{a}, \mathbf{b}$ are tensors of arbitrary rank unless otherwise stated
s is a scalar, N is the number of tensor components

Table 1.2: Algebraic tensor operations in OpenFOAM

1.5 Dimensional units

In continuum mechanics, properties are represented in some chosen units, *e.g.* mass in kilograms (kg), volume in cubic metres (m^3), pressure in Pascals ($kg\,m\,s^{-2}$). Algebraic operations must be performed on these properties using consistent units of measurement; in particular, addition, subtraction and equality are only physically meaningful for properties of the same dimensional units. As a safeguard against implementing a meaningless operation, OpenFOAM encourages the user to attach dimensional units to any tensor and will then perform dimension checking of any tensor operation.

Units are defined using the dimensionSet class, *e.g.*

```
dimensionSet pressureDims(1, -1, -2, 0, 0, 0, 0);
```

No.	Property	Unit	Symbol
1	Mass	kilogram	k
2	Length	metre	m
3	Time	second	s
4	Temperature	Kelvin	K
5	Quantity	moles	mol
6	Current	ampere	A
7	Luminous intensity	candela	cd

Table 1.3: S.I. base units of measurement

where each of the values corresponds to the power of each of the S.I. base units of measurement listed in Table 1.3. The line of code declares `pressureDims` to be the dimensionSet for pressure $kg\,m\,s^{-2}$ since the first entry in the `pressureDims` array, 1, corresponds to k^1, the second entry, -1, corresponds to m^{-1} *etc.*. A tensor with units is defined using the dimensioned<Type> template class, the <Type> being scalar, vector, tensor, *etc.*. The dimensioned<Type> stores a variable name of class word,the value <Type> and a dimensionSet

```
dimensionedTensor sigma
    (
        "sigma",
        dimensionSet(1, -1, -2, 0, 0, 0, 0),
        tensor(1e6,0,0,0,1e6,0,0,0,1e6),
    );
```

creates a tensor with correct dimensions of pressure, or stress

$$\boldsymbol{\sigma} = \begin{pmatrix} 10^6 & 0 & 0 \\ 0 & 10^6 & 0 \\ 0 & 0 & 10^6 \end{pmatrix} \tag{1.45}$$

Chapter 2

Discretisation procedures

So far we have dealt with algebra of tensors at a point. The PDEs we wish to solve involve derivatives of tensors with respect to time and space. We therefore need to extend our description to a *tensor field*, *i.e.* a tensor that varies across time and spatial domains. In this Chapter we will first present a mathematical description of all the differential operators we may encounter. We will then show how a tensor field is constructed in OpenFOAM and how the derivatives of these fields are discretised into a set of algebraic equations.

2.1 Differential operators

Before defining the spatial derivatives we first introduce the nabla *vector operator* ∇, represented in index notation as ∂_i:

$$\nabla \equiv \ \partial_i \equiv \frac{\partial}{\partial x_i} \equiv \left(\frac{\partial}{\partial x_1}, \frac{\partial}{\partial x_2}, \frac{\partial}{\partial x_3} \right) \tag{2.1}$$

The nabla operator is a useful notation that obeys the following rules:

- it operates on the tensors to its right and the conventional rules of a derivative of a product, *e.g.* $\partial_i ab = (\partial_i a)\, b + a\, (\partial_i b)$;
- otherwise the nabla operator behaves like any other vector in an algebraic operation.

2.1.1 Gradient

If a scalar field s is defined and continuously differentiable then the gradient of s, ∇s is a vector field

$$\nabla s = \partial_i s = \left(\frac{\partial s}{\partial x_1}, \frac{\partial s}{\partial x_2}, \frac{\partial s}{\partial x_3} \right) \tag{2.2}$$

The gradient can operate on any tensor field to produce a tensor field that is one rank higher. For example, the gradient of a vector field $\mathbf{a}$ is a second rank tensor field

$$\nabla \mathbf{a} = \partial_i a_j = \begin{pmatrix} \partial a_1/\partial x_1 & \partial a_2/\partial x_1 & \partial a_3/\partial x_1 \\ \partial a_1/\partial x_2 & \partial a_2/\partial x_2 & \partial a_3/\partial x_2 \\ \partial a_1/\partial x_3 & \partial a_2/\partial x_3 & \partial a_3/\partial x_3 \end{pmatrix} \tag{2.3}$$

2.1.2 Divergence

If a vector field $\mathbf{a}$ is defined and continuously differentiable then the divergence of $\mathbf{a}$ is a scalar field

$$\nabla \bullet \mathbf{a} = \partial_i a_i = \frac{\partial a_1}{\partial x_1} + \frac{\partial a_2}{\partial x_2} + \frac{\partial a_3}{\partial x_3} \tag{2.4}$$

The divergence can operate on any tensor field of rank 1 and above to produce a tensor that is one rank lower. For example the divergence of a second rank tensor field $\mathbf{T}$ is a vector field (expanding the vector as a column array for convenience)

$$\nabla \bullet \mathbf{T} = \partial_i T_{ij} = \begin{pmatrix} \partial T_{11}/\partial x_1 + \partial T_{12}/\partial x_1 + \partial T_{13}/\partial x_1 \\ \partial T_{21}/\partial x_2 + \partial T_{22}/\partial x_2 + \partial T_{23}/\partial x_2 \\ \partial T_{31}/\partial x_3 + \partial T_{32}/\partial x_3 + \partial T_{33}/\partial x_3 \end{pmatrix} \tag{2.5}$$

2.1.3 Curl

If a vector field $\mathbf{a}$ is defined and continuously differentiable then the curl of $\mathbf{a}$, $\nabla \times \mathbf{a}$ is a vector field

$$\nabla \times \mathbf{a} = e_{ijk}\partial_j a_k = \left(\frac{\partial a_3}{\partial x_2} - \frac{\partial a_2}{\partial x_3}, \frac{\partial a_1}{\partial x_3} - \frac{\partial a_3}{\partial x_1}, \frac{\partial a_2}{\partial x_1} - \frac{\partial a_1}{\partial x_2} \right) \tag{2.6}$$

The curl is related to the gradient by

$$\nabla \times \mathbf{a} = 2\,(*\,\text{skew}\,\nabla \mathbf{a}) \tag{2.7}$$

2.1.4 Laplacian

The Laplacian is an operation that can be defined mathematically by a combination of the divergence and gradient operators by $\nabla^2 \equiv \nabla \bullet \nabla$. However, the Laplacian should be considered as a single operation that transforms a tensor field into another tensor field of the same rank, rather than a combination of two operations, one which raises the rank by 1 and one which reduces the rank by 1.

In fact, the Laplacian is best defined as a *scalar operator*, just as we defined nabla as a vector operator, by

$$\nabla^2 \equiv \partial^2 \equiv \frac{\partial^2}{\partial x_1^2} + \frac{\partial^2}{\partial x_2^2} + \frac{\partial^2}{\partial x_3^2} \tag{2.8}$$

For example, the Laplacian of a scalar field s is the scalar field

$$\nabla^2 s = \partial^2 s = \frac{\partial^2 s}{\partial x_1^2} + \frac{\partial^2 s}{\partial x_2^2} + \frac{\partial^2 s}{\partial x_3^2} \tag{2.9}$$

2.1.5 Temporal derivative

There is more than one definition of temporal, or time, derivative of a tensor. To describe the temporal derivatives we must first recall that the tensor relates to a property of a volume of material that may be moving. If we track an infinitesimally small volume of material, or

particle, as it moves and observe the change in the tensorial property ϕ in time, we have the *total*, or *material* time derivative denoted by

$$\frac{D\phi}{Dt} = \lim_{\Delta t \to 0} \frac{\Delta \phi}{\Delta t} \tag{2.10}$$

However in continuum mechanics, particularly fluid mechanics, we often observe the change of a ϕ in time at a fixed point in space as different particles move across that point. This change at a point in space is termed the *spatial* time derivative which is denoted by $\partial/\partial t$ and is related to the material derivative by:

$$\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + \mathbf{U} \bullet \nabla \phi \tag{2.11}$$

where $\mathbf{U}$ is the velocity field of property ϕ. The second term on the right is known as the convective rate of change of ϕ.

2.2 Overview of discretisation

The term discretisation means *approximation of a problem into discrete quantities*. The FV method and others, such as the finite element and finite difference methods, all discretise the problem as follows:

Spatial discretisation Defining the solution domain by a set of points that fill and bound a region of space when connected;

Temporal discretisation (For transient problems) dividing the time domain into into a finite number of time intervals, or steps;

Equation discretisation Generating a system of algebraic equations in terms of discrete quantities defined at specific locations in the domain, from the PDEs that characterise the problem.

2.2.1 OpenFOAM lists and fields

OpenFOAM frequently needs to store sets of data and perform functions, such as mathematical operations, on the data. OpenFOAM therefore provides an array template class List<Type>, making it possible to create a list of any object of class Type that inherits the functions of the Type. For example a List of vector is List<vector>.

Lists of the tensor classes are defined as standard in OpenFOAM by the template class Field<Type>. For better code legibility, all instances of Field<Type>, *e.g.*Field<vector>, are renamed using `typedef` declarations as scalarField, vectorField, tensorField, symmTensorField, tensorThirdField and symmTensorThirdField. Algebraic operations can be performed between Fields subject to obvious restrictions such as the fields having the same number of elements. OpenFOAM also supports operations between a field and single tensor, *e.g.* all values of a Field `U` can be multiplied by the scalar 2 with the operation `U = 2.0 * U`.

2.3 Discretisation of the solution domain

Discretisation of the solution domain is shown in Figure 2.1. The space domain is discretised into computational mesh on which the PDEs are subsequently discretised. Discretisation of time, if required, is simple: it is broken into a set of time steps Δt that may change during a numerical simulation, perhaps depending on some condition calculated during the simulation.

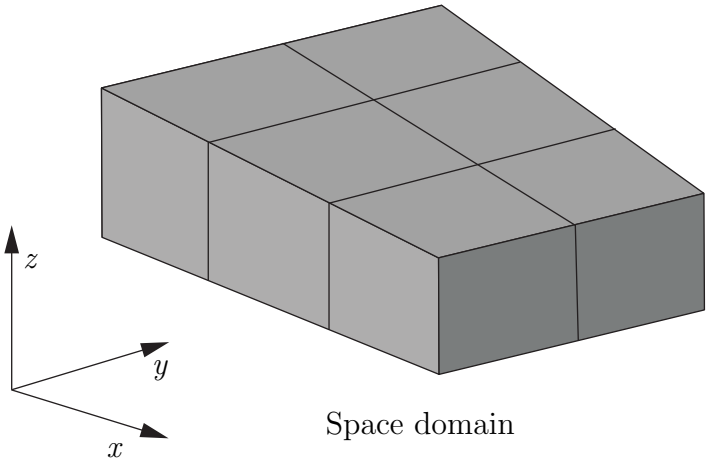


Figure 2.1: Discretisation of the solution domain

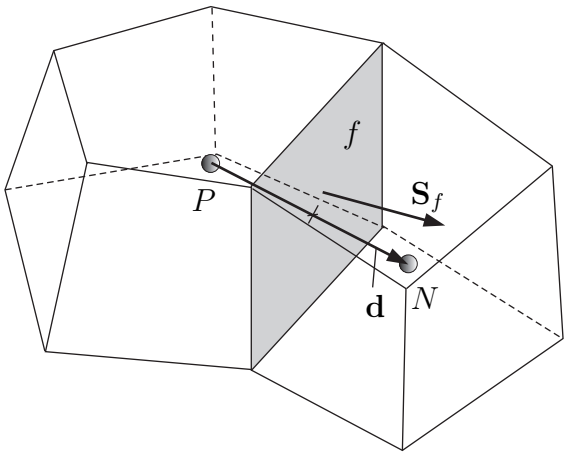


Figure 2.2: Parameters in finite volume discretisation

On a more detailed level, discretisation of space requires the subdivision of the domain into a number of cells, or control volumes. The cells are contiguous, *i.e.* they do not overlap one another and completely fill the domain. A typical cell is shown in Figure 2.2. Dependent variables and other properties are principally stored at the cell centroid P although they

may be stored on faces or vertices. The cell is bounded by a set of flat faces, given the generic label f. In OpenFOAM there is no limitation on the number of faces bounding each cell, nor any restriction on the alignment of each face. This kind of mesh is often referred to as "arbitrarily unstructured" to differentiate it from meshes in which the cell faces have a prescribed alignment, typically with the coordinate axes. Codes with arbitrarily unstructured meshes offer greater freedom in mesh generation and manipulation in particular when the geometry of the domain is complex or changes over time.

Whilst most properties are defined at the cell centroids, some are defined at cell faces. There are two types of cell face.

Internal faces Those faces that connect two cells (and it can never be more than two). For each internal face, OpenFOAM designates one adjoining cell to be the face *owner* and the other to be the *neighbour*;

Boundary faces Those belonging to one cell since they coincide with the boundary of the domain. These faces simply have an owner cell.

2.3.1 Defining a mesh in OpenFOAM

There are different levels of mesh description in OpenFOAM, beginning with the most basic mesh class, named polyMesh since it is based on polyhedra. A polyMesh is constructed using the minimum information required to define the mesh geometry described below and presented in Figure 2.3:

Points A list of cell vertex point coordinate vectors, *i.e.* a vectorField, that is renamed pointField using a `typedef` declaration;

Faces A list of cell faces List<face>, or faceList, where the face class is defined by a list of vertex numbers, corresponding to the pointField;

Cells a list of cells List<cell>, or cellList, where the cell class is defined by a list of face numbers, corresponding to the faceList described previously.

Boundary a polyBoundaryMesh decomposed into a list of patches, polyPatchList representing different regions of the boundary. The boundary is subdivided in this manner to allow different boundary conditions to be specified on different patches during a solution. All the faces of any polyPatch are stored as a single block of the faceList, so that its faces can be easily accessed using the slice class which stores references to the first and last face of the block. Each polyPatch is then constructed from

- a slice;
- a word to assign it a name.

FV discretisation uses specific data that is derived from the mesh geometry stored in polyMesh. OpenFOAM therefore extends the polyMesh class to fvMesh which stores the additional data needed for FV discretisation. fvMesh is constructed from polyMesh and stores the data in Table 2.1 which can be updated during runtime in cases where the mesh moves, is refined *etc.*.

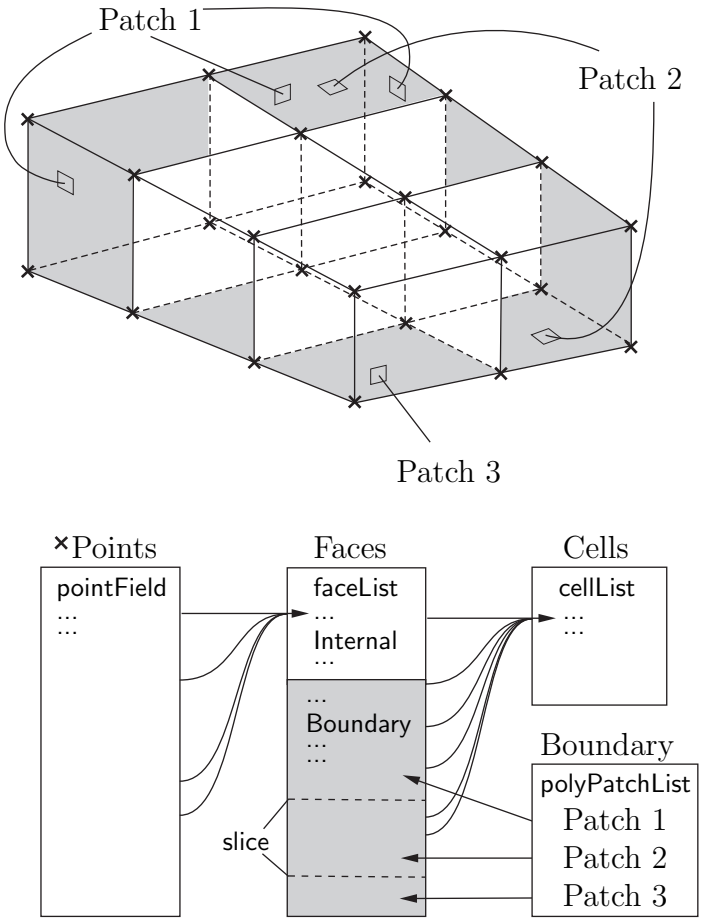


Figure 2.3: Schematic of the basic mesh description used in OpenFOAM

2.3.2 Defining a geometricField in OpenFOAM

So far we can define a field, *i.e.* a list of tensors, and a mesh. These can be combined to define a tensor field relating to discrete points in our domain, specified in OpenFOAM by the template class geometricField<Type>. The Field values are separated into those defined within the internal region of the domain, *e.g.* at the cell centres, and those defined on the domain boundary, *e.g.* on the boundary faces. The geometricField<Type> stores the following information:

Internal field This is simply a Field<Type>, described in Section 2.2.1;

BoundaryField This is a GeometricBoundaryField, in which a Field is defined for the faces of each patch and a Field is defined for the patches of the boundary. This is then a field of fields, stored within an object of the FieldField<Type> class. A reference to the fvBoundaryMesh is also stored [**].

Mesh A reference to an fvMesh, with some additional detail as to the whether the field is defined at cell centres, faces, *etc.*.

Dimensions A dimensionSet, described in Section 4.2.6.

Old values Discretisation of time derivatives requires field data from previous time steps.

Class	Description	Symbol	Access function
volScalarField	Cell volumes	V	`V()`
surfaceVectorField	Face area vectors	$\mathbf{S}_f$	`Sf()`
surfaceScalarField	Face area magnitudes	$\|\mathbf{S}_f\|$	`magSf()`
volVectorField	Cell centres	$\mathbf{C}$	`C()`
surfaceVectorField	Face centres	$\mathbf{C}_f$	`Cf()`
surfaceScalarField	Face motion fluxes **	ϕ_g	`phi()`

Table 2.1: fvMesh stored data.

The geometricField<Type> will store references to stored fields from the previous, or old, time step and its previous, or old-old, time step where necessary.

Previous iteration values The iterative solution procedures can use under-relaxation which requires access to data from the previous iteration. Again, if required, geometricField<Type> stores a reference to the data from the previous iteration.

As discussed in Section 2.3, we principally define a property at the cell centres but quite often it is stored at the cell faces and on occasion it is defined on cell vertices. The geometricField<Type> is renamed using `typedef` declarations to indicate where the field variable is defined as follows:

volField<Type> A field defined at cell centres;

surfaceField<Type> A field defined on cell faces;

pointField<Type> A field defined on cell vertices.

These `typedef` field classes of geometricField<Type>are illustrated in Figure 2.4. A geometricField<Type> inherits all the tensor algebra of Field<Type> and has all operations subjected to dimension checking using the dimensionSet. It can also be subjected to the FV discretisation procedures described in the following Section. The class structure used to build geometricField<Type> is shown in Figure 2.5[1].

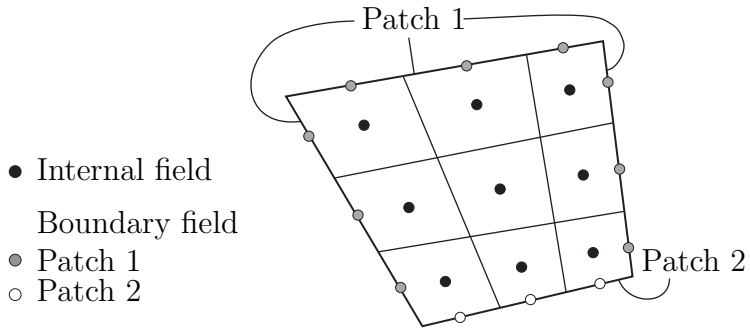
2.4 Equation discretisation

Equation discretisation converts the PDEs into a set of algebraic equations that are commonly expressed in matrix form as:

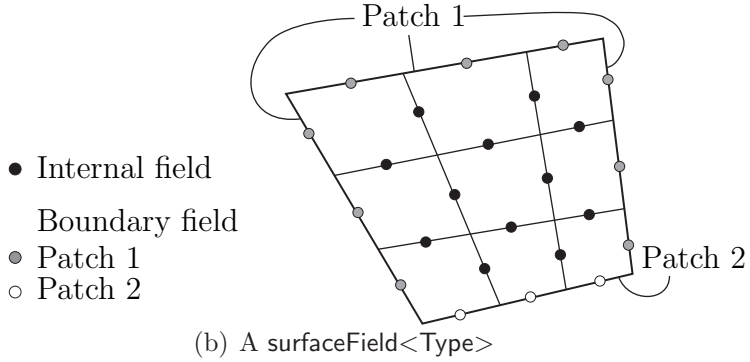
$$[A]\,[x] = [b] \tag{2.12}$$

where $[A]$ is a square matrix, $[x]$ is the column vector of dependent variable and $[b]$ is the source vector. The description of $[x]$ and $[b]$ as 'vectors' comes from matrix terminology rather than being a precise description of what they truly are: a list of values defined at locations in the geometry, *i.e.* a geometricField<Type>, or more specifically a volField<Type> when using FV discretisation.

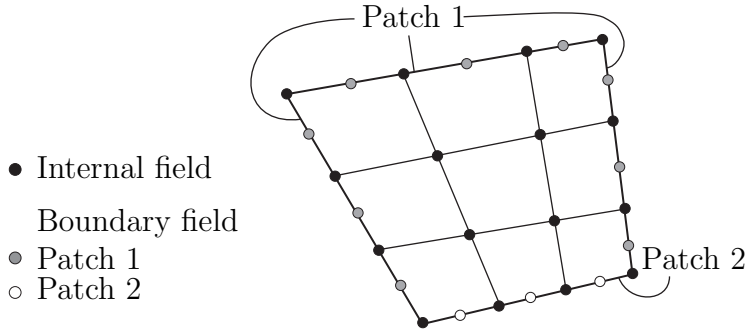
[1]The diagram is not an exact description of the class hierarchy, rather a representation of the general structure leading from some primitive classes to geometric<Type>Field.



(a) A volField<Type>



(b) A surfaceField<Type>



(c) A pointField<Type>

Figure 2.4: Types of geometricField<Type> defined on a mesh with 2 boundary patches (in 2 dimensions for simplicity)

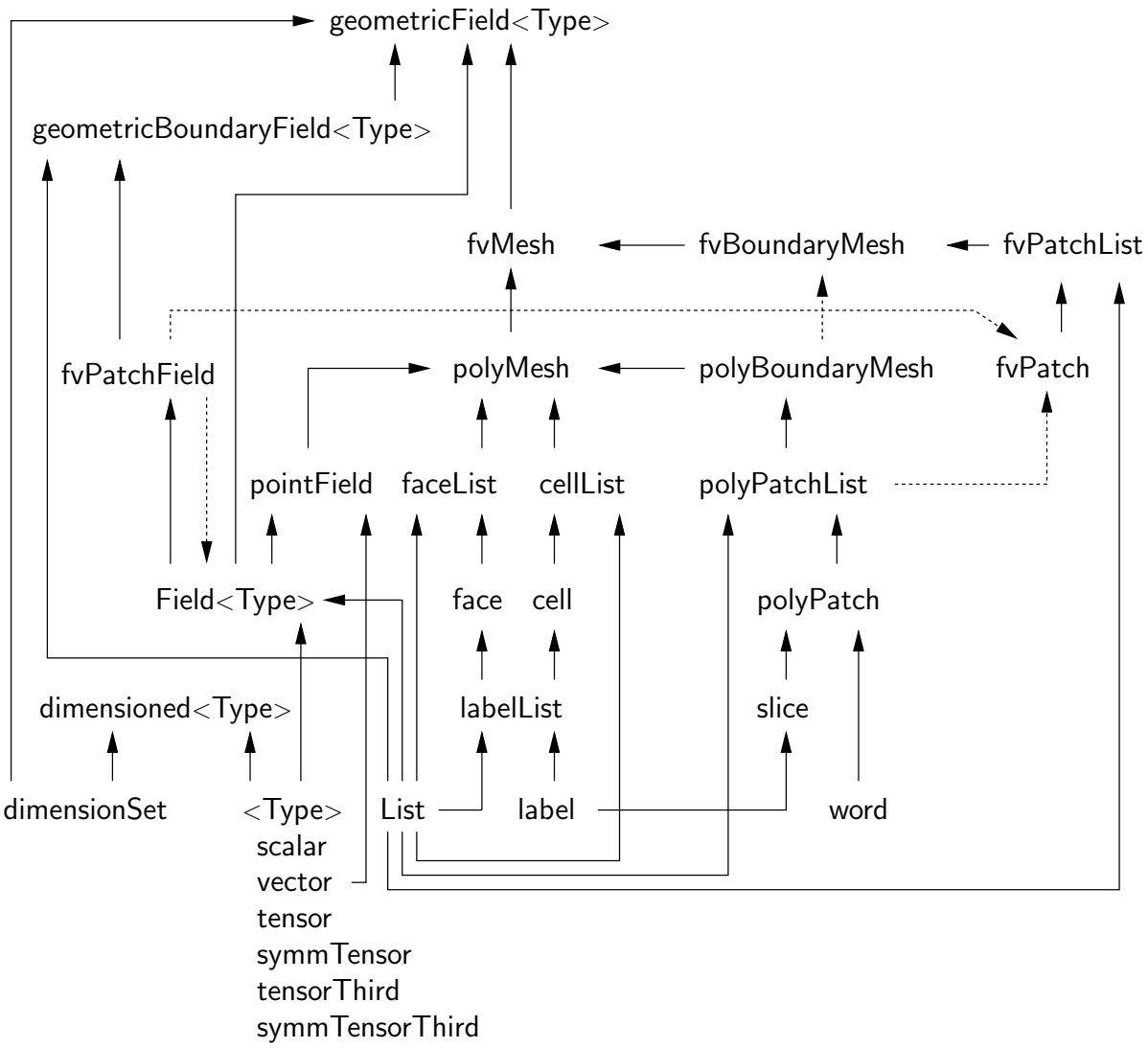


Figure 2.5: Basic class structure leading to geometricField<Type>

$[A]$ is a list of coefficients of a set of algebraic equations, and cannot be described as a geometricField<Type>. It is therefore given a class of its own: fvMatrix. fvMatrix<Type> is created through discretisation of a geometric<Type>Field and therefore inherits the <Type>. It supports many of the standard algebraic matrix operations of addition +, subtraction - and multiplication *.

Each term in a PDE is represented individually in OpenFOAM code using the classes of static functions finiteVolumeMethod and finiteVolumeCalculus, abbreviated by a `typedef` to fvm and fvc respectively. fvm and fvc contain static functions, representing differential operators, *e.g.* ∇^2, $\nabla\bullet$ and $\partial/\partial t$, that discretise geometricField<Type>s. The purpose of defining these functions within two classes, fvm and fvc, rather than one, is to distinguish:

- functions of fvm that calculate implicit derivatives of and return an fvMatrix<Type>
- some functions of fvc that calculate explicit derivatives and other explicit calculations, returning a geometricField<Type>.

Figure 2.6 shows a geometricField<Type> defined on a mesh with 2 boundary patches and illustrates the explicit operations merely transform one field to another and drawn in 2D for simplicity.

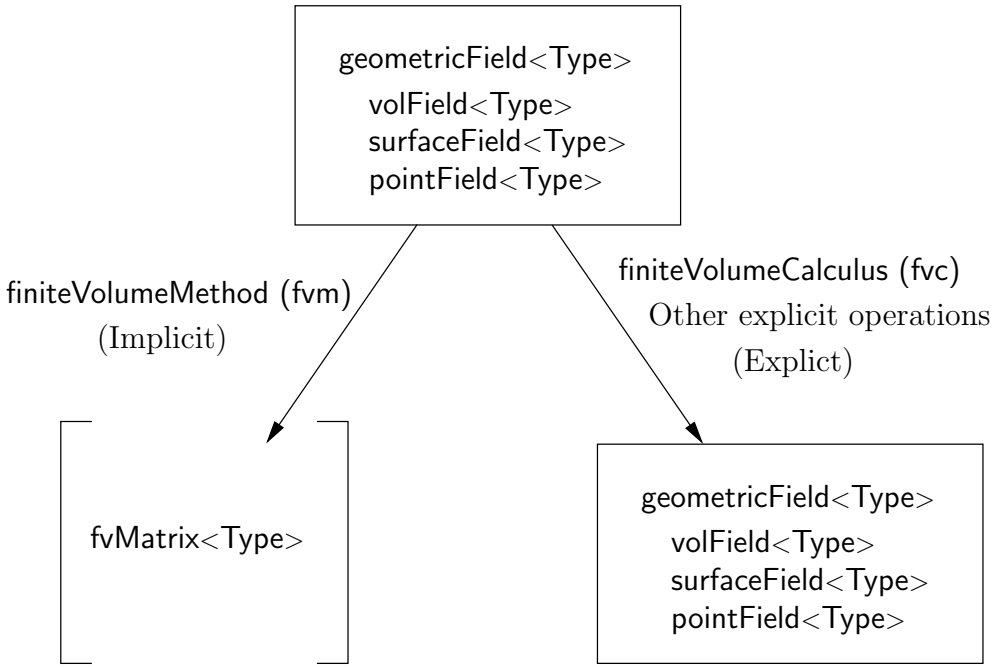


Figure 2.6: A geometricField<Type> and its operators

Table 2.2 lists the main functions that are available in fvm and fvc to discretise terms that may be found in a PDE. FV discretisation of each term is formulated by first integrating the term over a cell volume V. Most spatial derivative terms are then converted to integrals over the cell surface S bounding the volume using Gauss's theorem

$$\int_V \nabla \star \phi \; dV = \int_S d\mathbf{S} \star \phi \tag{2.13}$$

where $\mathbf{S}$ is the surface area vector, ϕ can represent any tensor field and the star notation $\star$ is used to represent any tensor product, *i.e.* inner, outer and cross and the respective

Term description	Implicit / Explicit	Text expression	fvm::/fvc:: functions
Laplacian	Imp/Exp	$\nabla^2\phi$	`laplacian(phi)`
		$\nabla \bullet \Gamma\nabla\phi$	`laplacian(Gamma, phi)`
Time derivative	Imp/Exp	$\frac{\partial\phi}{\partial t}$	`ddt(phi)`
		$\frac{\partial\rho\phi}{\partial t}$	`ddt(rho,phi)`
Second time derivative	Imp/Exp	$\frac{\partial}{\partial t}\left(\rho\frac{\partial\phi}{\partial t}\right)$	`d2dt2(rho, phi)`
Convection	Imp/Exp	$\nabla \bullet (\psi)$	`div(psi,scheme)`*
		$\nabla \bullet (\psi\phi)$	`div(psi, phi, word)`*
			`div(psi, phi)`
Divergence	Exp	$\nabla \bullet \chi$	`div(chi)`
Gradient	Exp	$\nabla\chi$	`grad(chi)`
		$\nabla\phi$	`gGrad(phi)`
			`lsGrad(phi)`
			`snGrad(phi)`
			`snGradCorrection(phi)`
Grad-grad squared	Exp	$\lvert\nabla\nabla\phi\rvert^2$	`sqrGradGrad(phi)`
Curl	Exp	$\nabla \times \phi$	`curl(phi)`
Source	Imp	$\rho\phi$	`Sp(rho,phi)`
	Imp/Exp†		`SuSp(rho,phi)`

†`fvm::SuSp` source is discretised implicit or explicit depending on the sign of `rho`.
†An explicit source can be introduced simply as a vol<Type>Field, *e.g.*`rho*phi`.
Function arguments can be of the following classes:
`phi`: vol<Type>Field
`Gamma`: scalar volScalarField, surfaceScalarField, volTensorField, surfaceTensorField.
`rho`: scalar, volScalarField
`psi`: surfaceScalarField.
`chi`: surface<Type>Field, vol<Type>Field.

Table 2.2: Discretisation of PDE terms in OpenFOAM

derivatives: divergence $\nabla \bullet \phi$, gradient $\nabla \phi$ and $\nabla \times \phi$. Volume and surface integrals are then linearised using appropriate schemes which are described for each term in the following Sections. Some terms are always discretised using one scheme, a selection of schemes is offered in OpenFOAM for the discretisation of other terms. The choice of scheme is either made by a direct specification within the code or it can be read from an input file at job run-time and stored within an fvSchemes class object.

2.4.1 The Laplacian term

The Laplacian term is integrated over a control volume and linearised as follows:

$$\int_V \nabla \bullet (\Gamma \nabla \phi) \, dV = \int_S d\mathbf{S} \bullet (\Gamma \nabla \phi) = \sum_f \Gamma_f \mathbf{S}_f \bullet (\nabla \phi)_f \tag{2.14}$$

The face gradient discretisation is implicit when the length vector $\mathbf{d}$ between the centre of the cell of interest P and the centre of a neighbouring cell N is orthogonal to the face plane, *i.e.* parallel to $\mathbf{S}_f$:

$$\mathbf{S}_f \bullet (\nabla \phi)_f = |S_f| \frac{\phi_N - \phi_P}{|\mathbf{d}|} \tag{2.15}$$

In the case of non-orthogonal meshes, an additional explicit term is introduced which is evaluated by interpolating cell centre gradients, themselves calculated by central differencing cell centre values.

2.4.2 The convection term

The convection term is integrated over a control volume and linearised as follows:

$$\int_V \nabla \bullet (\rho \mathbf{U} \phi) \, dV = \int_S d\mathbf{S} \bullet (\rho \mathbf{U} \phi) = \sum_f \mathbf{S}_f \bullet (\rho \mathbf{U})_f \phi_f = \sum_f F \phi_f \tag{2.16}$$

The face field ϕ_f can be evaluated using a variety of schemes:

Central differencing (CD) is second-order accurate but unbounded

$$\phi_f = f_x \phi_P + (1 - f_x) \phi_N \tag{2.17}$$

where $f_x \equiv \overline{fN}/\overline{PN}$ where $\overline{fN}$ is the distance between f and cell centre N and $\overline{PN}$ is the distance between cell centres P and N.

Upwind differencing (UD) determines ϕ_f from the direction of flow and is bounded at the expense of accuracy

$$\phi_f = \begin{cases} \phi_P & \text{for } F \geq 0 \\ \phi_N & \text{for } F < 0 \end{cases} \tag{2.18}$$

Blended differencing (BD) schemes combine UD and CD in an attempt to preserve boundedness with reasonable accuracy,

$$\phi_f = (1 - \gamma) (\phi_f)_{UD} + \gamma (\phi_f)_{CD} \tag{2.19}$$

OpenFOAM has several implementations of the Gamma differencing scheme to select the blending coefficient γ but it offers other well-known schemes such as van Leer, SUPERBEE, MINMOD *etc.*.

2.4.3 First time derivative

The first time derivative $\partial/\partial t$ is integrated over a control volume as follows:

$$\frac{\partial}{\partial t}\int_V \rho\phi \; dV \tag{2.20}$$

The term is discretised by simple differencing in time using:

new values $\phi^n \equiv \phi(t + \Delta t)$ at the time step we are solving for;

old values $\phi^o \equiv \phi(t)$ that were stored from the previous time step;

old-old values $\phi^{oo} \equiv \phi(t - \Delta t)$ stored from a time step previous to the last.

One of two discretisation schemes can be declared using the `timeScheme` keyword in the appropriate input file, described in detail in section 4.4 of the User Guide.

Euler implicit scheme, `timeScheme EulerImplicit`, that is first order accurate in time:

$$\frac{\partial}{\partial t}\int_V \rho\phi \; dV = \frac{(\rho_P\phi_P V)^n - (\rho_P\phi_P V)^o}{\Delta t} \tag{2.21}$$

Backward differencing scheme, `timeScheme BackwardDifferencing`, that is second order accurate in time by storing the old-old values and therefore with a larger overhead in data storage than `EulerImplicit`:

$$\frac{\partial}{\partial t}\int_V \rho\phi \; dV = \frac{3\,(\rho_P\phi_P V)^n - 4\,(\rho_P\phi_P V)^o + (\rho_P\phi_P V)^{oo}}{2\Delta t} \tag{2.22}$$

2.4.4 Second time derivative

The second time derivative is integrated over a control volume and linearised as follows:

$$\frac{\partial}{\partial t}\int_V \rho\frac{\partial\phi}{\partial t} \; dV = \frac{(\rho_P\phi_P V)^n - 2\,(\rho_P\phi_P V)^o + (\rho_P\phi_P V)^{oo}}{\Delta t^2} \tag{2.23}$$

It is first order accurate in time.

2.4.5 Divergence

The divergence term described in this Section is strictly an explicit term that is distinguished from the convection term of Section 2.4.2, *i.e.* in that it is not the divergence of the product of a velocity and dependent variable. The term is integrated over a control volume and linearised as follows:

$$\int_V \nabla \bullet \phi \; dV = \int_S d\mathbf{S} \bullet \phi = \sum_f \mathbf{S}_f \bullet \phi_f \tag{2.24}$$

The `fvc::div` function can take as its argument either a surface<Type>Field, in which case ϕ_f is specified directly, or a vol<Type>Field which is interpolated to the face by central differencing as described in Section 2.4.10:

2.4.6 Gradient

The gradient term is an explicit term that can be evaluated in a variety of ways. The scheme can be evaluated either by selecting the particular grad function relevant to the discretisation scheme, *e.g.*`fvc::gGrad`, `fvc::lsGrad` *etc.*, or by using the `fvc::grad` function combined with the appropriate `timeScheme` keyword in an input file

Gauss integration is invoked using the `fvc::grad` function with `timeScheme Gauss` or directly using the `fvc::gGrad` function. The discretisation is performed using the standard method of applying Gauss's theorem to the volume integral:

$$\int_V \nabla\phi \; dV = \int_S d\mathbf{S}\, \phi = \sum_f \mathbf{S}_f \phi_f \tag{2.25}$$

As with the `fvc::div` function, the Gaussian integration `fvc::grad` function can take either a surfaceField<Type> or a volField<Type> as an argument.

Least squares method is based on the following idea:

1. a value at point P can be extrapolated to neighbouring point N using the gradient at P;
2. the extrapolated value at N can be compared to the actual value at N, the difference being the error;
3. if we now minimise the sum of the square of weighted errors at all neighbours of P with the respect to the gradient, then the gradient should be a good approximation.

Least squares is invoked using the `fvc::grad` function with `timeScheme leastSquares` or directly using the `fvc::lsGrad` function. The discretisation is performed as by first calculating the tensor $\mathbf{G}$ at every point P by summing over neighbours N:

$$\mathbf{G} = \sum_N w_N^2 \mathbf{dd} \tag{2.26}$$

where $\mathbf{d}$ is the vector from P to N and the weighting function $w_N = 1/|\mathbf{d}|$. The gradient is then evaluated as:

$$(\nabla\phi)_P = \sum_N w_N^2 \mathbf{G}^{-1} \bullet \mathbf{d}\,(\phi_N - \phi_P) \tag{2.27}$$

Surface normal gradient The gradient normal to a surface $\mathbf{n}_f \bullet (\nabla\phi)_f$ can be evaluated at cell faces using the scheme

$$(\nabla\phi)_f = \frac{\phi_N - \phi_P}{|\mathbf{d}|} \tag{2.28}$$

This gradient is called by the function `fvc::snGrad` and returns a surfaceField<Type>. The scheme is directly analogous to that evaluated for the Laplacian discretisation scheme in Section 2.4.1, and in the same manner, a correction can be introduced to improve the accuracy of this face gradient in the case of non-orthogonal meshes. This correction is called using the function `fvc::snGradCorrection` [Check**].

2.4.7 Grad-grad squared

The grad-grad squared term is evaluated by: taking the gradient of the field; taking the gradient of the resulting gradient field; and then calculating the magnitude squared of the result. The mathematical expression for grad-grad squared of ϕ is $|\nabla(\nabla\phi)|^2$.

2.4.8 Curl

The curl is evaluated from the gradient term described in Section 2.4.6. First, the gradient is discretised and then the curl is evaluated using the relationship from Equation 2.7, repeated here for convenience

$$\nabla \times \phi = 2\,*(\text{skew}\,\nabla\phi)$$

2.4.9 Source terms

Source terms can be specified in 3 ways

Explicit Every explicit term is a volField<Type>. Hence, an explicit source term can be incorporated into an equation simply as a field of values. For example if we wished to solve Poisson's equation $\nabla^2\phi = f$, we would define `phi` and `f` as volScalarField and then do

```
solve(fvm::laplacian(phi) == f)
```

Implicit An implicit source term is integrated over a control volume and linearised by

$$\int_V \rho\phi \; dV = \rho_P V_P \phi_P \tag{2.29}$$

Implicit/Explicit The implicit source term changes the coefficient of the diagonal of the matrix. Depending on the sign of the coefficient and matrix terms, this will either increase or decrease diagonal dominance of the matrix. Decreasing the diagonal dominance could cause instability during iterative solution of the matrix equation. Therefore OpenFOAM provides a mixed source discretisation procedure that is implicit when the coefficients that are greater than zero, and explicit for the coefficients less than zero. In mathematical terms the matrix coefficient for node P is $V_P \max(\rho_P, 0)$ and the source term is $V_P\phi_P \min(\rho_P, 0)$.

2.4.10 Other explicit discretisation schemes

There are some other discretisation procedures that convert volField<Type>s into surface<Type>Fields and visa versa.

Surface integral `fvc::surfaceIntegrate` performs a summation of surface<Type>Field face values bounding each cell and dividing by the cell volume, *i.e.* $(\sum_f \phi_f)/V_P$. It returns a volField<Type>.

Surface sum `fvc::surfaceSum` performs a summation of surface<Type>Field face values bounding each cell, *i.e.* $\sum_f \phi_f$ returning a volField<Type>.

Average `fvc::average` produces an area weighted average of surface<Type>Field face values, *i.e.* $(\sum_f S_f\phi_f)/\sum_f S_f$, and returns a volField<Type>.

Reconstruct

Face interpolate The geometric<Type>Field function `faceInterpolate()` interpolates volField<Type> cell centre values to cell faces using central differencing, returning a surface<Type>Field.

2.5 Temporal discretisation

Although we have described the discretisation of temporal derivatives in Sections 2.4.3 and 2.4.4, we need to consider how to treat the spatial derivatives in a transient problem. If we denote all the spatial terms as $\mathcal{A}\phi$ where $\mathcal{A}$ is any spatial operator, *e.g.* Laplacian, then we can express a transient PDE in integral form as

$$\int_t^{t+\Delta t}\left[\frac{\partial}{\partial t}\int_V \rho\phi \; dV + \int_V \mathcal{A}\phi \; dV\right] \; dt = 0 \tag{2.30}$$

Using the Euler implicit method of Equation 2.21, the first term can be expressed as

$$\begin{aligned}\int_t^{t+\Delta t}\left[\frac{\partial}{\partial t}\int_V \rho\phi \; dV\right] \; dt &= \int_t^{t+\Delta t}\frac{(\rho_P\phi_P V)^n - (\rho_P\phi_P V)^o}{\Delta t} \; dt \\ &= \frac{(\rho_P\phi_P V)^n - (\rho_P\phi_P V)^o}{\Delta t}\Delta t\end{aligned} \tag{2.31}$$

The second term can be expressed as

$$\int_t^{t+\Delta t}\left[\int_V \mathcal{A}\phi \; dV\right] \; dt = \int_t^{t+\Delta t}\mathcal{A}^*\phi \; dt \tag{2.32}$$

where $\mathcal{A}^*$ represents the spatial discretisation of $\mathcal{A}$. The time integral can be discretised in three ways:

Euler implicit uses implicit discretisation of the spatial terms, thereby taking current values ϕ^n.

$$\int_t^{t+\Delta t}\mathcal{A}^*\phi \; dt = \mathcal{A}^*\phi^n\Delta t \tag{2.33}$$

It is first order accurate in time, guarantees boundedness and is unconditionally stable.

Explicit uses explicit discretisation of the spatial terms, thereby taking old values ϕ^o.

$$\int_t^{t+\Delta t}\mathcal{A}^*\phi \; dt = \mathcal{A}^*\phi^o\Delta t \tag{2.34}$$

It is first order accurate in time and is unstable if the Courant number Co is greater than 1. The Courant number is defined as

$$Co = \frac{\mathbf{U}_f \bullet \mathbf{d}}{|\mathbf{d}|^2\Delta t} \tag{2.35}$$

where $\mathbf{U}_f$ is a characteristic velocity, *e.g.* velocity of a wave front, velocity of flow.

Crank Nicholson uses the trapezoid rule to discretise the spatial terms, thereby taking a mean of current values ϕ^n and old values ϕ^o.

$$\int_t^{t+\Delta t} \mathcal{A}^*\phi \; dt = \mathcal{A}^* \left(\frac{\phi^n + \phi^o}{2} \right) \Delta t \tag{2.36}$$

It is second order accurate in time, is unconditionally stable but does not guarantee boundedness.

2.5.1 Treatment of temporal discretisation in OpenFOAM

At present the treatment of the temporal discretisation is controlled by the implementation of the spatial derivatives in the PDE we wish to solve. For example, let us say we wish to solve a transient diffusion equation

$$\frac{\partial \phi}{\partial t} = \kappa \nabla^2 \phi \tag{2.37}$$

An Euler implicit implementation of this would read

```
solve(fvm::ddt(phi) == kappa*fvm::laplacian(phi))
```

where we use the fvm class to discretise the `Laplacian` term implicitly. An explicit implementation would read

```
solve(fvm::ddt(phi) == kappa*fvc::laplacian(phi))
```

where we now use the fvc class to discretise the `Laplacian` term explicitly. The Crank Nicholson scheme can be implemented by the mean of implicit and explicit terms:

```
solve
    (
    fvm::ddt(phi)
    ==
    kappa*0.5*(fvm::laplacian(phi) + fvc::laplacian(phi))
    )
```

2.6 Boundary Conditions

Boundary conditions are required to complete the problem we wish to solve. We therefore need to specify boundary conditions on all our boundary faces. Boundary conditions can be divided into 2 types:

Dirichlet prescribes the value of the dependent variable on the boundary and is therefore termed 'fixed value' in this guide;

Neumann prescribes the gradient of the variable normal to the boundary and is therefore termed 'fixed gradient' in this guide.

When we perform discretisation of terms that include the sum over faces $\sum_f$, we need to consider what happens when one of the faces is a boundary face.

Fixed value We specify a fixed value at the boundary ϕ_b

- We can simply substitute ϕ_b in cases where the discretisation requires the value on a boundary face ϕ_f, *e.g.* in the convection term in Equation 2.16.
- In terms where the face gradient $(\nabla\phi)_f$ is required, *e.g.* Laplacian, it is calculated using the boundary face value and cell centre value,

$$\mathbf{S}_f \bullet (\nabla\phi)_f = |S_f| \frac{\phi_b - \phi_P}{|\mathbf{d}|} \tag{2.38}$$

Fixed gradient The fixed gradient boundary condition g_b is a specification on inner product of the gradient and unit normal to the boundary, or

$$g_b = \left(\frac{\mathbf{S}}{|\mathbf{S}|} \bullet \nabla\phi\right)_f \tag{2.39}$$

- When discretisation requires the value on a boundary face ϕ_f we must interpolate the cell centre value to the boundary by

$$\begin{aligned} \phi_f &= \phi_P + \mathbf{d} \bullet (\nabla\phi)_f \\ &= \phi_P + |\mathbf{d}|\, g_b \end{aligned} \tag{2.40}$$

- ϕ_b can be directly substituted in cases where the discretisation requires the face gradient to be evaluated,

$$\mathbf{S}_f \bullet (\nabla\phi)_f = |S_f|\, g_b \tag{2.41}$$

2.6.1 Physical boundary conditions

The specification of boundary conditions is usually an engineer's interpretation of the true behaviour. Real boundary conditions are generally defined by some physical attributes rather than the numerical description as described of the previous Section. In incompressible fluid flow there are the following physical boundaries

Inlet The velocity field at the inlet is supplied and, for consistency, the boundary condition on pressure is zero gradient.

Outlet The pressure field at the outlet is supplied and a zero gradient boundary condition on velocity is specified.

No-slip impermeable wall The velocity of the fluid is equal to that of the wall itself, *i.e.* a fixed value condition can be specified. The pressure is specified zero gradient since the flux through the wall is zero.

In a problem whose solution domain and boundary conditions are symmetric about a plane, we only need to model half the domain to one side of the symmetry plane. The boundary condition on the plane must be specified according to

Symmetry plane The symmetry plane condition specifies the component of the gradient normal to the plane should be zero. [Check**]

Chapter 3

Examples of the use of OpenFOAM

In this section we shall describe several test cases supplied with the OpenFOAM distribution. The intention is to provide example cases, including those in the tutorials in chapter 2 of the User Guide, for every standard solver. The examples are designed to introduce certain tools and features of OpenFOAM, *e.g.* within pre-/post-processing, numerical schemes, algorithms. They also provide a means for validation of solvers although that is not their principal function.

Each example contains a description of the problem: the geometry, initial and boundary conditions, a brief description of the equations being solved, models used, and physical properties required. The solution domain is selected which may be a portion of the original geometry, *e.g.* if we introduce symmetry planes. The method of meshing, usually blockMesh, is specified; of course the user can simply view the mesh since every example is distributed with the *polyMesh* directory containing the data files that describe the mesh.

The examples coexist with the tutorials in the *tutorials* subdirectory of the OpenFOAM installation. They are organised into a set of subdirectories by solver, *e.g.* all the icoFoam cases are stored within a subdirectory *icoFoam*. Before running a particular example, the user is urged to copy it into their user account. We recommend that the user stores all OpenFOAM cases in a directory we recommend that the tutorials are copied into a directory *$FOAM_RUN*. If this directory structure has not yet been created in the user's account, it can be created with

```
mkdir -p $FOAM_RUN
```

The tutorials can then be copied into this directory with

```
cp -r $FOAM_TUTORIALS/* $FOAM_RUN
```

3.1 Flow around a cylinder

In this example we shall investigate potential flow around a cylinder using potentialFoam. This example introduces the following OpenFOAM features:

- non-orthogonal meshes;
- generating an analytical solution to a problem in OpenFOAM.

3.1.1 Problem specification

The problem is defined as follows:

Solution domain The domain is 2 dimensional and consists of a square domain with a cylinder collocated with the centre of the square as shown in Figure 3.1.

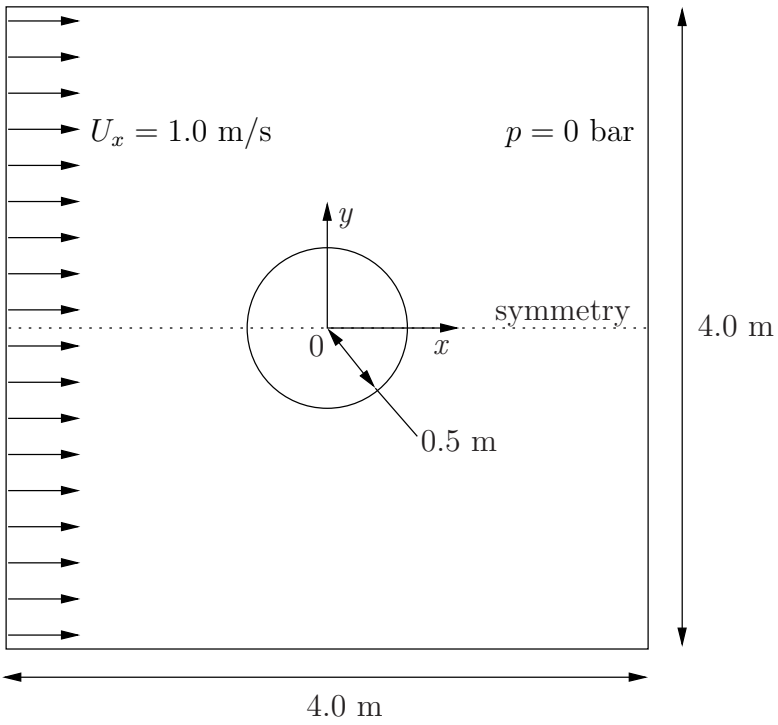


Figure 3.1: Geometry of flow round a cylinder

Governing equations

- Mass continuity for an incompressible fluid

$$\nabla \cdot \mathbf{U} = 0 \tag{3.1}$$

- Pressure equation for an incompressible, irrotational fluid assuming steady-state conditions

$$\nabla^2 p = 0 \tag{3.2}$$

Boundary conditions

- Inlet (left) with fixed velocity $\mathbf{U} = (1, 0, 0)$ m/s.
- Outlet (right) with a fixed pressure $p = 0$ Pa.
- No-slip wall (bottom);
- Symmetry plane (top).

Initial conditions $U = 0$ m/s, $p = 0$ Pa — required in OpenFOAM input files but not necessary for the solution since the problem is steady-state.

Solver name potentialFoam: a potential flow code, *i.e.* assumes the flow is incompressible, steady, irrotational, inviscid and it ignores gravity.

Case name *cylinder* case located in the *$FOAM_TUTORIALS/potentialFoam* directory.

3.1.2 Note on potentialFoam

potentialFoam is a useful solver to validate OpenFOAM since the assumptions of potential flow are such that an analytical solution exists for cases whose geometries are relatively simple. In this example of flow around a cylinder an analytical solution exists with which we can compare our numerical solution. potentialFoam can also be run more like a utility to provide a (reasonably) conservative initial **U** field for a problem. When running certain cases, this can useful for avoiding instabilities due to the initial field being unstable. In short, potentialFoam creates a conservative field from a non-conservative initial field supplied by the user.

3.1.3 Mesh generation

Mesh generation using blockMesh has been described in tutorials in the User Guide. In this case, the mesh consists of 10 blocks as shown in Figure 3.2. Remember that all meshes

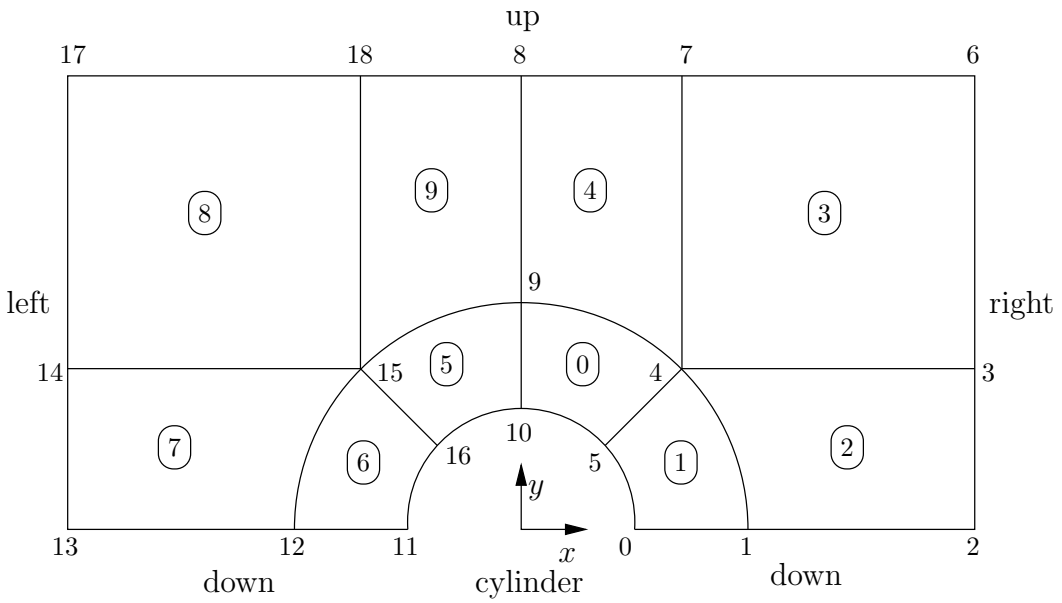


Figure 3.2: Blocks in cylinder geometry

are treated as 3 dimensional in OpenFOAM. If we wish to solve a 2 dimensional problem, we must describe a 3 dimensional mesh that is only one cell thick in the third direction that is not solved. In Figure 3.2 we show only the back plane of the geometry, along $z = -0.5$, in which the vertex numbers are numbered 0-18. The other 19 vertices in the front plane, $z = +0.5$, are numbered in the same order as the back plane, as shown in the mesh description file below:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
```

```
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices
(
    (0.5 0 -0.5)
    (1 0 -0.5)
    (2 0 -0.5)
    (2 0.707107 -0.5)
    (0.707107 0.707107 -0.5)
    (0.353553 0.353553 -0.5)
    (2 2 -0.5)
    (0.707107 2 -0.5)
    (0 2 -0.5)
    (0 1 -0.5)
    (0 0.5 -0.5)
    (-0.5 0 -0.5)
    (-1 0 -0.5)
    (-2 0 -0.5)
    (-2 0.707107 -0.5)
    (-0.707107 0.707107 -0.5)
    (-0.353553 0.353553 -0.5)
    (-2 2 -0.5)
    (-0.707107 2 -0.5)
    (0.5 0 0.5)
    (1 0 0.5)
    (2 0 0.5)
    (2 0.707107 0.5)
    (0.707107 0.707107 0.5)
    (0.353553 0.353553 0.5)
    (2 2 0.5)
    (0.707107 2 0.5)
    (0 2 0.5)
    (0 1 0.5)
    (0 0.5 0.5)
    (-0.5 0 0.5)
    (-1 0 0.5)
    (-2 0 0.5)
    (-2 0.707107 0.5)
    (-0.707107 0.707107 0.5)
    (-0.353553 0.353553 0.5)
    (-2 2 0.5)
    (-0.707107 2 0.5)
);

blocks
(
    hex (5 4 9 10 24 23 28 29) (10 10 1) simpleGrading (1 1 1)
    hex (0 1 4 5 19 20 23 24) (10 10 1) simpleGrading (1 1 1)
    hex (1 2 3 4 20 21 22 23) (20 10 1) simpleGrading (1 1 1)
    hex (4 3 6 7 23 22 25 26) (20 20 1) simpleGrading (1 1 1)
    hex (9 4 7 8 28 23 26 27) (10 20 1) simpleGrading (1 1 1)
    hex (15 16 10 9 34 35 29 28) (10 10 1) simpleGrading (1 1 1)
    hex (12 11 16 15 31 30 35 34) (10 10 1) simpleGrading (1 1 1)
    hex (13 12 15 14 32 31 34 33) (20 10 1) simpleGrading (1 1 1)
    hex (14 15 18 17 33 34 37 36) (20 20 1) simpleGrading (1 1 1)
    hex (15 9 8 18 34 28 27 37) (10 20 1) simpleGrading (1 1 1)
);

edges
(
    arc 0 5 (0.469846 0.17101 -0.5)
    arc 5 10 (0.17101 0.469846 -0.5)
    arc 1 4 (0.939693 0.34202 -0.5)
    arc 4 9 (0.34202 0.939693 -0.5)
    arc 19 24 (0.469846 0.17101 0.5)
    arc 24 29 (0.17101 0.469846 0.5)
    arc 20 23 (0.939693 0.34202 0.5)
    arc 23 28 (0.34202 0.939693 0.5)
    arc 11 16 (-0.469846 0.17101 -0.5)
    arc 16 10 (-0.17101 0.469846 -0.5)
    arc 12 15 (-0.939693 0.34202 -0.5)
    arc 15 9 (-0.34202 0.939693 -0.5)
    arc 30 35 (-0.469846 0.17101 0.5)
    arc 35 29 (-0.17101 0.469846 0.5)
    arc 31 34 (-0.939693 0.34202 0.5)
```

```
92      arc 34 28 (-0.34202 0.939693 0.5)
93  );
94
95  patches
96  (
97      symmetryPlane down
98      (
99          (0 1 20 19)
100         (1 2 21 20)
101         (12 11 30 31)
102         (13 12 31 32)
103     )
104     patch right
105     (
106         (2 3 22 21)
107         (3 6 25 22)
108     )
109     symmetryPlane up
110     (
111         (7 8 27 26)
112         (6 7 26 25)
113         (8 18 37 27)
114         (18 17 36 37)
115     )
116     patch left
117     (
118         (14 13 32 33)
119         (17 14 33 36)
120     )
121     symmetryPlane cylinder
122     (
123         (10 5 24 29)
124         (5 0 19 24)
125         (16 10 29 35)
126         (11 16 35 30)
127     )
128 );
129
130 mergePatchPairs
131 (
132 );
133
134 // ************************************************************************* //
```

3.1.4 Boundary conditions and initial fields

Using FoamX or editing case files by hand, set the boundary conditions in accordance with the problem description in Figure 3.1, *i.e.* the left boundary should be an Inlet, the right boundary should be an Outlet and the down and cylinder boundaries should be symmetryPlane. The top boundary conditions is chosen so that we can make the most genuine comparison with our analytical solution which uses the assumption that the domain is infinite in the y direction. The result is that the normal gradient of **U** is small along a plane coinciding with our boundary. We therefore impose the condition that the normal component is zero, *i.e.* specify the boundary as a symmetryPlane, thereby ensuring that the comparison with the analytical is reasonable.

3.1.5 Running the case

No fluid properties need be specified in this problem since the flow is assumed to be incompressible and inviscid. In the *system* subdirectory, the *controlDict* specifies the control parameters for the run. Note that since we assume steady flow, we only run for 1 time step:

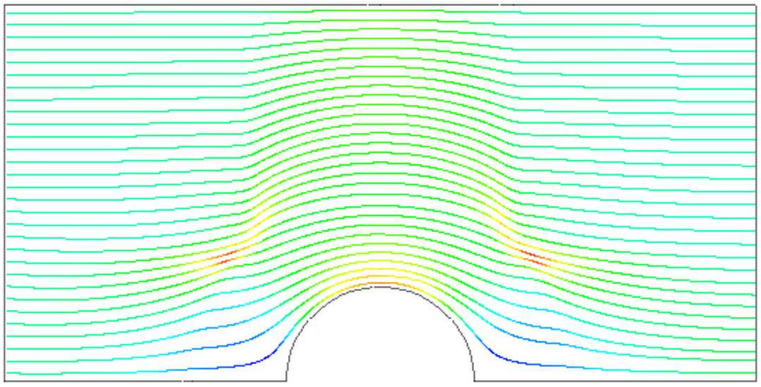
```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  1.7.1                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
```

```
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     potentialFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         1;

deltaT          1;

writeControl    timeStep;

writeInterval   1;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression uncompressed;

timeFormat      general;

timePrecision   6;

runTimeModifiable yes;


// ************************************************************************* //
```

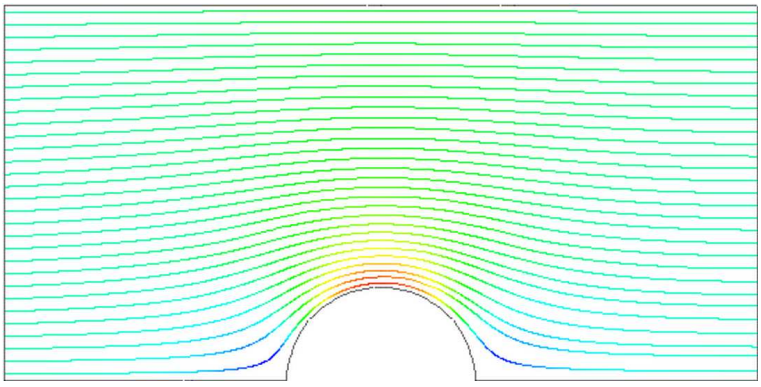
potentialFoam executes an iterative loop around the pressure equation which it solves in order that explicit terms relating to non-orthogonal correction in the Laplacian term may be updated in successive iterations. The number of iterations around the pressure equation is controlled by the `nNonOrthogonalCorrectors` keyword in *controlDict*. In the first instance we can set `nNonOrthogonalCorrectors` to 0 so that no loops are performed, *i.e.* the pressure equation is solved once, and there is no non-orthogonal correction. The solution is shown in Figure 3.3(a) (at $t = 1$, when the steady-state simulation is complete). We expect the solution to show smooth streamlines passing across the domain as in the analytical solution in Figure 3.3(c), yet there is clearly some error in the regions where there is high non-orthogonality in the mesh, *e.g.* at the join of blocks 0, 1 and 3. The case can be run a second time with some non-orthogonal correction by setting `nNonOrthogonalCorrectors` to 3. The solution shows smooth streamlines with no significant error due to non-orthogonality as shown in Figure 3.3(b).

3.1.6 Generating the analytical solution

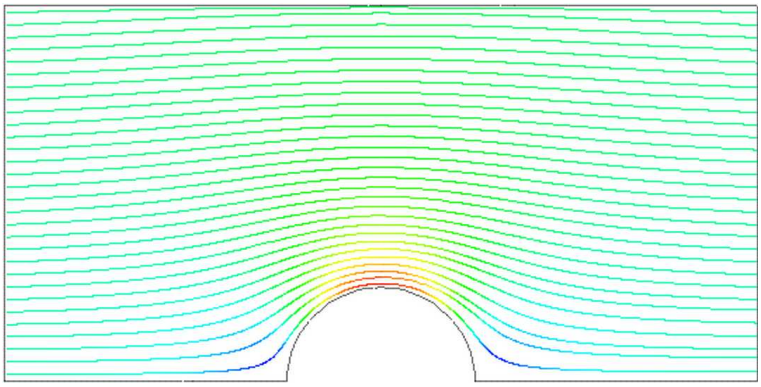
Source code is included in the *$FOAM_TUTORIALS/potentialFoam/analyticalCylinder* directory to generate the analytical solution for the potential flow case. The velocity at any point



(a) With no non-orthogonal correction



(b) With non-orthogonal correction



(c) Analytical solution

Figure 3.3: Streamlines of potential flow

at a distance d and angle θ from the cylinder centre is described analytically as

$$U_x = U_\infty \left[1 - \left(\frac{r}{d}\right)^2 \cos 2\theta\right]$$

$$U_y = U_\infty \left(\frac{r}{d}\right)^2 \sin 2\theta \quad (3.3)$$

where r is the cylinder radius and U_∞ is the inlet flow velocity. Here, θ describes the angle subtended from the x-axis.

Let us examine some details of the source code in the *analyticalCylinder* directory. In *createFields.H*, the velocity field is read in using the `IOobject::NO_WRITE` option to ensure that the field data can never be overwritten during execution of analyticalCylinder. The inlet velocity and cylinder radius are taken from data read from the mesh and a field `UA` is set up to store the analytical solution:

```
Info<< "Reading field U\n" << endl;
volVectorField U
(
    IOobject
    (
        "U",
        runTime.timeName(),
        mesh,
        IOobject::MUST_READ,
        IOobject::NO_WRITE
    ),
    mesh
);

Info<< "Reading inlet velocity  uInfX\n" << endl;

dimensionedScalar uInfX
(
    "uInfx",
    dimensionSet(0, 1, -1, 0, 0),
    U.boundaryField()[3][0].x()
);
Info << "U at inlet = " << uInfX.value() << " m/s" << endl;

dimensionedScalar radius
(
    "radius",
    dimensionSet(0, 1, 0, 0, 0),
    mag(U.mesh().boundary()[4].Cf()[0])
);

Info << "Cylinder radius = " << radius.value() << " m" << endl;

volVectorField UA
(
    IOobject
    (
        "UA",
        runTime.timeName(),
        mesh,
        IOobject::NO_READ,
        IOobject::AUTO_WRITE
    ),
    U
);
```

Thea main code *analyticalCylinder.C* performs the following tasks:

- increments the time step by `runTime++`;
- generates the analytical solution for field `UA` using tensor arithmetic;
- writes the solution to file by `runTime.writeObjects()`.

```
/*---------------------------------------------------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     |
    \\  /    A nd           | Copyright (C) 1991-2010 OpenCFD Ltd.
     \\/     M anipulation  |
-------------------------------------------------------------------------------
License
    This file is part of OpenFOAM.

    OpenFOAM is free software: you can redistribute it and/or modify it
    under the terms of the GNU General Public License as published by
    the Free Software Foundation, either version 3 of the License, or
    (at your option) any later version.

    OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
    ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
    FITNESS FOR A PARTICULAR PURPOSE.  See the GNU General Public License
    for more details.

    You should have received a copy of the GNU General Public License
    along with OpenFOAM.  If not, see <http://www.gnu.org/licenses/>.

Application
    analyticalCylinder

Description
    Generates an analytical solution for potential flow around a cylinder.
    Can be compared with the solution from the potentialFlow/cylinder example.

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

int main(int argc, char *argv[])
{

#   include "setRootCase.H"

#   include "createTime.H"
#   include "createMesh.H"
#   include "createFields.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info << "\nEvaluating analytical solution" << endl;

    volVectorField centres = UA.mesh().C();
    volScalarField magCentres = mag(centres);
    volScalarField theta = acos((centres & vector(1,0,0))/magCentres);

    volVectorField cs2theta =
        cos(2*theta)*vector(1,0,0)
      + sin(2*theta)*vector(0,1,0);

    UA = uInfX*(dimensionedVector(vector(1,0,0))
      - pow((radius/magCentres),2)*cs2theta);

    runTime.write();

    Info<< "end" << endl;

    return 0;
}

// ************************************************************************* //
```

The utility must be compiled with wmake as normal. It can then be run by typing

```
analyticalCylinder $FOAM_RUN/potentialFoam cylinder
```

The analytical solution is plotted as streamlines as shown in Figure 3.3(c). Note that differences in the analytical and numerical solutions at the top plane are due to the fact

that the analytical solution assumes an infinite boundary and the numerical solution specifies a zeroGradient boundary condition at that boundary.

3.1.7 Exercise

Investigate the accuracy of the numerical solution by implementing some measure of comparison between the numercial and analytical in analyticalCylinder.

3.2 Steady turbulent flow over a backward-facing step

In this example we shall investigate steady turbulent flow over a backward-facing step. The problem description is taken from one used by Pitz and Daily in an experimental investigation [**] against which the computed solution can be compared. This example introduces the following OpenFOAM features for the first time:

- generation of a mesh using blockMesh using full mesh grading capability;
- steady turbulent flow.

3.2.1 Problem specification

The problem is defined as follows:

Solution domain The domain is 2 dimensional, consisting of a short inlet, a backward-facing step and converging nozzle at outlet as shown in Figure 3.4.

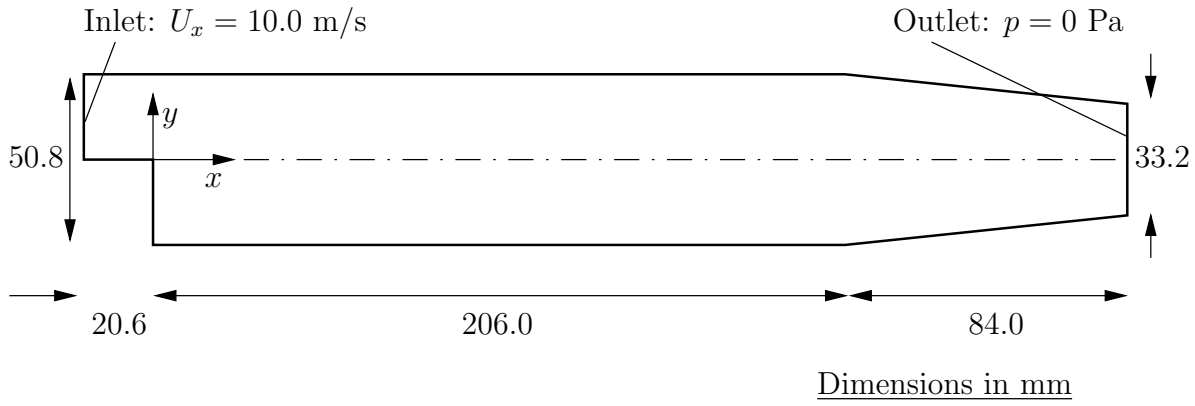


Figure 3.4: Geometry of backward-facing step

Governing equations

- Mass continuity for incompressible flow

$$\nabla \bullet \mathbf{U} = 0 \tag{3.4}$$

- Steady flow momentum equation

$$\nabla \bullet (\mathbf{U}\mathbf{U}) + \nabla \bullet \mathbf{R} = -\nabla p \tag{3.5}$$

where p is kinematic pressure and (in slightly over-simplistic terms) $\mathbf{R} = \nu_{eff}\nabla\mathbf{U}$ is the viscous stress term with an effective kinematic viscosity ν_{eff}, calculated from selected transport and turbulence models.

Initial conditions $U = 0$ m/s, $p = 0$ Pa — required in OpenFOAM input files but not necessary for the solution since the problem is steady-state.

Boundary conditions

- Inlet (left) with fixed velocity $\mathbf{U} = (10, 0, 0)$ m/s;
- Outlet (right) with fixed pressure $p = 0$ Pa;
- No-slip walls on other boundaries.

Transport properties

- Kinematic viscosity of air $\nu = \mu/\rho = 18.1 \times 10^{-6}/1.293 = 14.0$ μm^2/s

Turbulence model

- Standard $k - \epsilon$;
- Coefficients: $C_\mu = 0.09; C_1 = 1.44; C_2 = 1.92; \alpha_k = 1; \alpha_\epsilon = 0.76923$.

Solver name simpleFoam: an implementation for steady incompressible flow.

Case name *pitzDaily*, located in the *$FOAM_TUTORIALS/simpleFoam* directory.

The problem is solved using simpleFoam, so-called as it is an implementation for steady flow using the SIMPLE algorithm [**]. The solver has full access to all the turbulence models in the incompressibleTurbulenceModels library and the non-Newtonian models incompressibleTransportModels library of the standard OpenFOAM release.

3.2.2 Mesh generation

We expect that the flow in this problem is reasonably complex and an optimum solution will require grading of the mesh. In general, the regions of highest shear are particularly critical, requiring a finer mesh than in the regions of low shear. We can anticipate where high shear will occur by considering what the solution might be in advance of any calculation. At the inlet we have strong uniform flow in the x direction and, as it passes over the step, it generates shear on the fluid below, generating a vortex in the bottom half of the domain. The regions of high shear will therefore be close to the centreline of the domain and close to the walls.

The domain is subdivided into 12 blocks as shown in Figure 3.5.

The mesh is 3 dimensional, as always in OpenFOAM, so in Figure 3.5 we are viewing the back plane along $z = -0.5$. The full set of vertices and blocks are given in the mesh description file below:

```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  1.7.1                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
13     object      blockMeshDict;
```

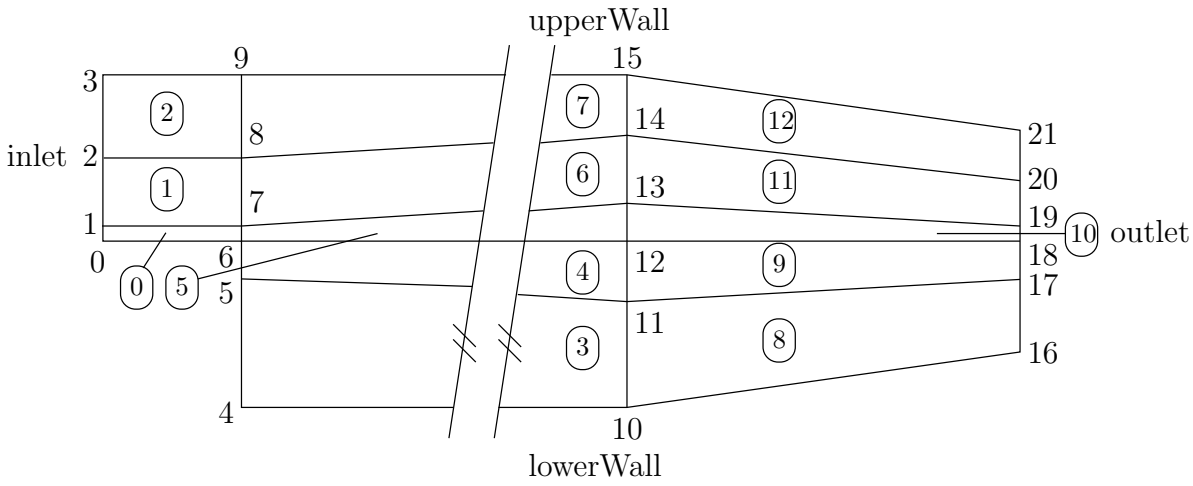


Figure 3.5: Blocks in backward-facing step

```
14  }
15  // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
16
17  convertToMeters 0.001;
18
19  vertices
20  (
21      (-20.6 0 -0.5)
22      (-20.6 3 -0.5)
23      (-20.6 12.7 -0.5)
24      (-20.6 25.4 -0.5)
25      (0 -25.4 -0.5)
26      (0 -5 -0.5)
27      (0 0 -0.5)
28      (0 3 -0.5)
29      (0 12.7 -0.5)
30      (0 25.4 -0.5)
31      (206 -25.4 -0.5)
32      (206 -8.5 -0.5)
33      (206 0 -0.5)
34      (206 6.5 -0.5)
35      (206 17 -0.5)
36      (206 25.4 -0.5)
37      (290 -16.6 -0.5)
38      (290 -6.3 -0.5)
39      (290 0 -0.5)
40      (290 4.5 -0.5)
41      (290 11 -0.5)
42      (290 16.6 -0.5)
43      (-20.6 0 0.5)
44      (-20.6 3 0.5)
45      (-20.6 12.7 0.5)
46      (-20.6 25.4 0.5)
47      (0 -25.4 0.5)
48      (0 -5 0.5)
49      (0 0 0.5)
50      (0 3 0.5)
51      (0 12.7 0.5)
52      (0 25.4 0.5)
53      (206 -25.4 0.5)
54      (206 -8.5 0.5)
55      (206 0 0.5)
56      (206 6.5 0.5)
57      (206 17 0.5)
58      (206 25.4 0.5)
59      (290 -16.6 0.5)
60      (290 -6.3 0.5)
61      (290 0 0.5)
62      (290 4.5 0.5)
63      (290 11 0.5)
64      (290 16.6 0.5)
65  );
66
67  blocks
68  (
```

```
    hex (0 6 7 1 22 28 29 23) (18 7 1) simpleGrading (0.5 1.8 1)
    hex (1 7 8 2 23 29 30 24) (18 10 1) simpleGrading (0.5 4 1)
    hex (2 8 9 3 24 30 31 25) (18 13 1) simpleGrading (0.5 0.25 1)
    hex (4 10 11 5 26 32 33 27) (180 18 1) simpleGrading (4 1 1)
    hex (5 11 12 6 27 33 34 28) (180 9 1) edgeGrading (4 4 4 4 0.5 1 1 0.5 1 1 1 1)
    hex (6 12 13 7 28 34 35 29) (180 7 1) edgeGrading (4 4 4 4 1.8 1 1 1.8 1 1 1 1)
    hex (7 13 14 8 29 35 36 30) (180 10 1) edgeGrading (4 4 4 4 4 1 1 4 1 1 1 1)
    hex (8 14 15 9 30 36 37 31) (180 13 1) simpleGrading (4 0.25 1)
    hex (10 16 17 11 32 38 39 33) (25 18 1) simpleGrading (2.5 1 1)
    hex (11 17 18 12 33 39 40 34) (25 9 1) simpleGrading (2.5 1 1)
    hex (12 18 19 13 34 40 41 35) (25 7 1) simpleGrading (2.5 1 1)
    hex (13 19 20 14 35 41 42 36) (25 10 1) simpleGrading (2.5 1 1)
    hex (14 20 21 15 36 42 43 37) (25 13 1) simpleGrading (2.5 0.25 1)
);

edges
(
);

patches
(
    patch inlet
    (
        (0 22 23 1)
        (1 23 24 2)
        (2 24 25 3)
    )
    patch outlet
    (
        (16 17 39 38)
        (17 18 40 39)
        (18 19 41 40)
        (19 20 42 41)
        (20 21 43 42)
    )
    wall upperWall
    (
        (3 25 31 9)
        (9 31 37 15)
        (15 37 43 21)
    )
    wall lowerWall
    (
        (0 6 28 22)
        (6 5 27 28)
        (5 4 26 27)
        (4 10 32 26)
        (10 16 38 32)
    )
    empty frontAndBack
    (
        (22 28 29 23)
        (23 29 30 24)
        (24 30 31 25)
        (26 32 33 27)
        (27 33 34 28)
        (28 34 35 29)
        (29 35 36 30)
        (30 36 37 31)
        (32 38 39 33)
        (33 39 40 34)
        (34 40 41 35)
        (35 41 42 36)
        (36 42 43 37)
        (0 1 7 6)
        (1 2 8 7)
        (2 3 9 8)
        (4 5 11 10)
        (5 6 12 11)
        (6 7 13 12)
        (7 8 14 13)
        (8 9 15 14)
        (10 11 17 16)
        (11 12 18 17)
        (12 13 19 18)
        (13 14 20 19)
        (14 15 21 20)
```

```
146         )
147     );
148
149     mergePatchPairs
150     (
151     );
152
153     // ************************************************************************* //
```

A major feature of this problem is the use of the full mesh grading capability of blockMesh that is described in section 5.3.1 of the User Guide. The user can see that blocks 4,5 and 6 use the full list of 12 expansion ratios. The expansion ratios correspond to each edge of the block, the first 4 to the edges aligned in the local x_1 direction, the second 4 to the edges in the local x_2 direction and the last 4 to the edges in the local x_3 direction. In blocks 4, 5, and 6, the ratios are equal for all edges in the local x_1 and x_3 directions but not for the edges in the x_2 direction that corresponds in all blocks to the global y. If we consider the ratios used in relation to the block definition in section 5.3.1 of the User Guide, we realize that different gradings have been prescribed along the left and right edges in blocks 4,5 and 6 in Figure 3.5. The purpose of this differential grading is to generate a fine mesh close to the most critical region of flow, the corner of the step, and allow it to expand into the rest of the domain.

The mesh can be generated using blockMesh from the command line or from within FoamX and viewed as described in previous examples.

3.2.3 Boundary conditions and initial fields

The case files can be viewed, or edited from within FoamX or by hand. In this case, we are required to set the initial and boundary fields for velocity **U**, pressure p, turbulent kinetic energy k and dissipation rate ε. The boundary conditions can be specified by setting the physical patch types in FoamX: the upper and lower walls are set to `Wall`, the left patch to `Inlet` and the right patch to `Outlet`. These physical boundary conditions require us to specify a fixedValue at the inlet on **U**, k and ε. **U** is given in the problem specification, but the values of k and ϵ must be chosen by the user in a similar manner to that described in section 2.1.8.1 of the User Guide. We assume that the inlet turbulence is isotropic and estimate the fluctuations to be 5% of **U** at the inlet. We have

$$U'_x = U'_y = U'_z = \frac{5}{100}10 = 0.5 \text{ m/s} \tag{3.6}$$

and

$$k = \frac{3}{2}(0.5)^2 = 0.375 \text{ m}^2/\text{s}^2 \tag{3.7}$$

If we estimate the turbulent length scale l to be 10% of the width of the inlet then

$$\varepsilon = \frac{C_\mu^{0.75} k^{1.5}}{l} = \frac{0.09^{0.75} 0.375^{1.5}}{0.1 \times 25.4 \times 10^{-3}} = 14.855 \,\text{m}^2/\text{s}^3 \tag{3.8}$$

At the outlet we need only specify the pressure $p = 0$Pa.

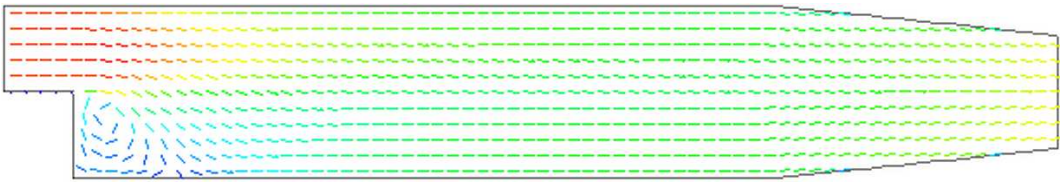
3.2.4 Case control

The choices of *fvSchemes* are as follows: the `timeScheme` should be `SteadyState`; the `gradScheme` and `laplacianScheme` should be set as default to `Gauss`; and, the `divScheme` should be set to `UD` to ensure boundedness.

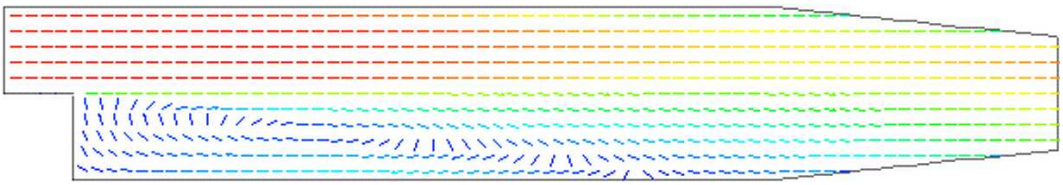
Special attention should be paid to the settings of *fvTolerances*. Although the top level simpleFoam code contains only equations for p and $\mathbf{U}$, the turbulent model solves equations for k, ε and $\mathbf{R}$, and tolerance settings are required for all 5 equations. A `solverTolerance` of 10^{-5} and `solverRelativeTolerance` of 0.1 are acceptable for all variables with the exception of p when 10^{-6} and 0.01 are recommended. Under-relaxation of the solution is required since the problem is steady. A `relaxationFactor` of 0.7 is acceptable for $\mathbf{U}$, k, ε and $\mathbf{R}$ but 0.3 is required for p to avoid numerical instability.

Finally, in *controlDict*, the time step `deltaT` should be set to 1 since in steady state cases such as this is effectively an iteration counter. With benefit of hindsight we know that the solution requires 1000 iterations reach reasonable convergence, hence `endTime` is set to 1000. Ensure that the `writeFrequency` is sufficiently high, *e.g.* 50, that you will not fill the hard disk with data during run time.

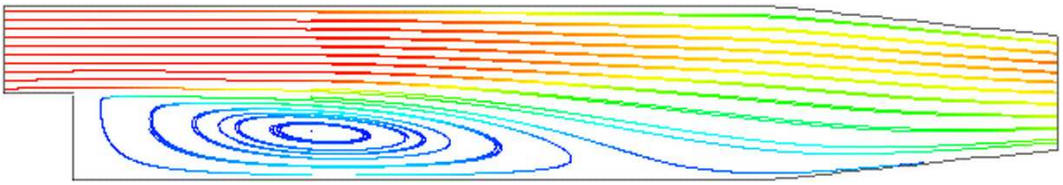
3.2.5 Running the case and post-processing



(a) Velocity vectors after 50 iterations



(b) Velocity vectors at 1000 iterations



(c) Streamlines at 1000 iterations

Figure 3.6: Development of a vortex in the backward-facing step.

Run the case and post-process the results. After a few iterations, *e.g.* 50, a vortex develops beneath the corner of the step that is the height of the step but narrow in the x-direction as shown by the vector plot of velocities is shown Figure 3.6(a). Over several iterations the vortex stretches in the x-direction from the step to the outlet until at 1000 iterations the system reaches a steady-state in which the vortex is fully developed as shown in Figure 3.6(b-c).

3.3 Supersonic flow over a forward-facing step

In this example we shall investigate supersonic flow over a forward-facing step. The problem description involves a flow of Mach 3 at an inlet to a rectangular geometry with a step near

the inlet region that generates shock waves.

This example introduces the following OpenFOAM features for the first time:

- supersonic flow;

3.3.1 Problem specification

The problem is defined as follows:

Solution domain The domain is 2 dimensional and consists of a short inlet section followed by a forward-facing step of 20% the height of the section as shown in Figure 3.7

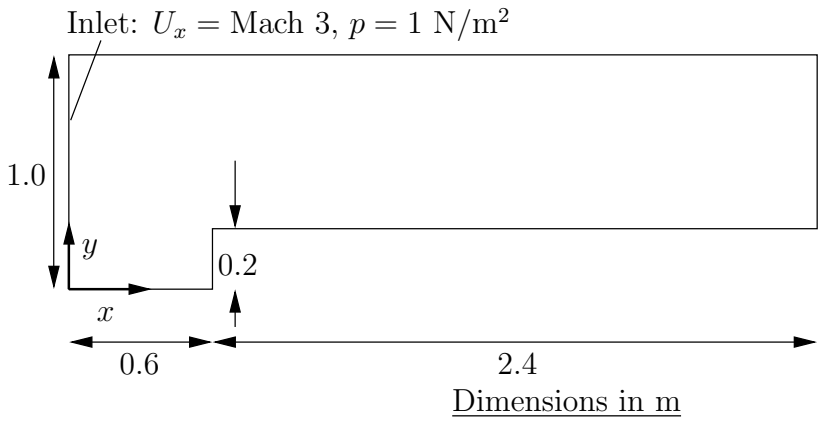


Figure 3.7: Geometry of the forward step geometry

Governing equations

- Mass continuity

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathbf{U}) = 0 \tag{3.9}$$

- Ideal gas

$$p = \rho R T \tag{3.10}$$

- Momentum equation for Newtonian fluid

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \bullet (\rho \mathbf{U}\mathbf{U}) - \nabla \bullet \mu \nabla \mathbf{U} = -\nabla p \tag{3.11}$$

- Energy equation for fluid (ignoring some viscous terms), $e = C_v T$, with Fourier's Law $\mathbf{q} = -k \nabla T$

$$\frac{\partial \rho e}{\partial t} + \nabla \bullet (\rho \mathbf{U} e) - \nabla \bullet \left(\frac{k}{C_v} \right) \nabla e = p \nabla \bullet \mathbf{U} \tag{3.12}$$

Initial conditions $U = 0$ m/s, $p = 1$ Pa, $T = 1$ K.

Boundary conditions

- Inlet (left) with fixedValue for velocity $U = 3$ m/s = Mach 3, pressure $p = 1$ Pa and temperature $T = 1$ K;
- Outlet (right) with zeroGradient on U, p and T;
- No-slip adiabatic wall (bottom);
- Symmetry plane (top).

Transport properties

- Dynamic viscosity of air $\mu = 18.1\mu\text{Pa}\,\text{s}$

Thermodynamic properties

- Specific heat at constant volume $C_v = 1.78571$ J/kg K
- Gas constant $R = 0.714286$ J/kg K
- Conductivity $k = 32.3$ μW/m K

Case name *forwardStep* case located in the *$FOAM_TUTORIALS/sonicFoam* directory.

Solver name sonicFoam: an implementation for compressible trans-sonic/supersonic laminar gas flow.

The case is designed such that the speed of sound of the gas $c = \sqrt{\gamma RT} = 1$ m/s, the consequence being that the velocities are directly equivalent to the Mach number, *e.g.* the inlet velocity of 3 m/s is equivalent to Mach 3. This speed of sound calculation can be verified using the relationship for a perfect gas, $C_p - Cv = R$, *i.e.* the ratio of specific heats

$$\gamma = C_p/C_v = \frac{R}{C_v} + 1 \tag{3.13}$$

3.3.2 Mesh generation

The mesh used in this case is relatively simple, specified with uniform rectangular cells of length 0.06 m in the x direction and 0.05 m in the y direction. The geometry can simply be divided into 3 blocks, one below the top of the step, and two above the step, one either side of the step front. The full set of vertices and blocks are given in the mesh description file below:

```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  1.7.1                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
13     object      blockMeshDict;
14 }
15 // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
16
17 convertToMeters 1;
18
19 vertices
20 (
```

```
21      (0 0 -0.05)
22      (0.6 0 -0.05)
23      (0 0.2 -0.05)
24      (0.6 0.2 -0.05)
25      (3 0.2 -0.05)
26      (0 1 -0.05)
27      (0.6 1 -0.05)
28      (3 1 -0.05)
29      (0 0 0.05)
30      (0.6 0 0.05)
31      (0 0.2 0.05)
32      (0.6 0.2 0.05)
33      (3 0.2 0.05)
34      (0 1 0.05)
35      (0.6 1 0.05)
36      (3 1 0.05)
37  );
38
39  blocks
40  (
41      hex (0 1 3 2 8 9 11 10) (25 10 1) simpleGrading (1 1 1)
42      hex (2 3 6 5 10 11 14 13) (25 40 1) simpleGrading (1 1 1)
43      hex (3 4 7 6 11 12 15 14) (100 40 1) simpleGrading (1 1 1)
44  );
45
46  edges
47  (
48  );
49
50  patches
51  (
52      patch inlet
53      (
54          (0 8 10 2)
55          (2 10 13 5)
56      )
57      patch outlet
58      (
59          (4 7 15 12)
60      )
61      symmetryPlane bottom
62      (
63          (0 1 9 8)
64      )
65      symmetryPlane top
66      (
67          (5 13 14 6)
68          (6 14 15 7)
69      )
70      patch obstacle
71      (
72          (1 3 11 9)
73          (3 4 12 11)
74      )
75  );
76
77  mergePatchPairs
78  (
79  );
80
81  // ************************************************************************* //
```

3.3.3 Running the case

The case approaches a steady-state at some time after 5 s. The results for pressure at 10 s are shown in Figure 3.8. The results clearly show discontinuities in pressure, *i.e.* shock waves, emanating from ahead of the base of the step.

3.3.4 Exercise

The user can examine the effect on the solution of increasing the inlet velocity.

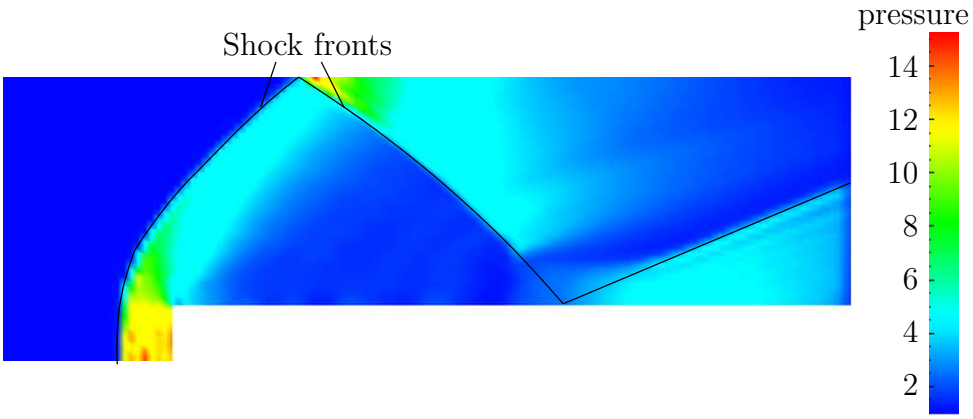


Figure 3.8: Shock fronts in the forward step problem

3.4 Decompression of a tank internally pressurised with water

In this example we shall investigate a problem of rapid opening of a pipe valve close to a pressurised liquid-filled tank. The prominent feature of the result in such cases is the propagation of pressure waves which must therefore be modelled as a compressible liquid.

This tutorial introduces the following OpenFOAM features for the first time:

- Mesh refinement
- Pressure waves in liquids

3.4.1 Problem specification

Solution domain The domain is 2 dimensional and consists of a tank with a small outflow pipe as shown in Figure 3.9

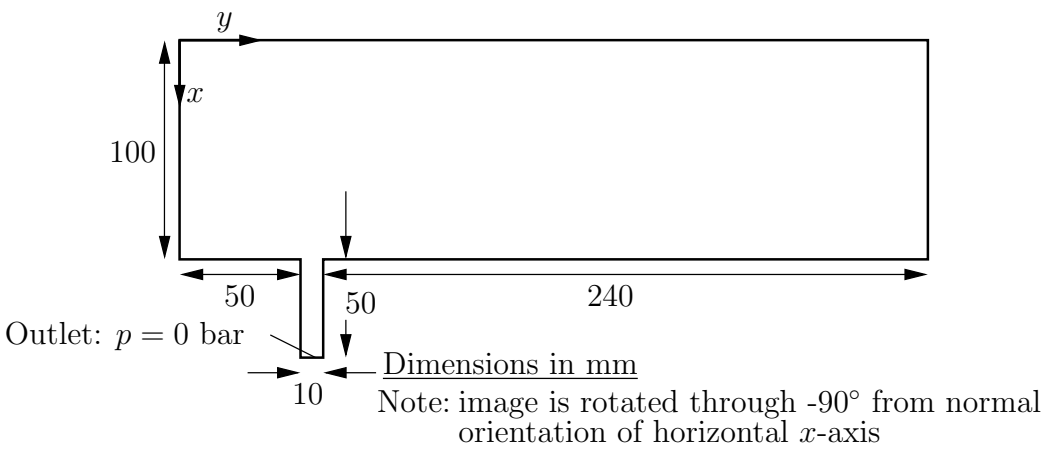


Figure 3.9: Geometry of a tank with outflow pipe

Governing equations This problem requires a model for compressibility ψ in the fluid in order to be able to resolve waves propagating at a finite speed. A barotropic relationship is used to relate density ρ and pressure p are related to ψ.

- Mass continuity

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathbf{U}) = 0 \tag{3.14}$$

- The barotropic relationship

$$\frac{\partial \rho}{\partial p} = \frac{\rho}{K} = \psi \tag{3.15}$$

  where K is the bulk modulus

- Equation 3.15 is linearised as

$$\rho \approx \rho_0 + \psi\,(p - p_0) \tag{3.16}$$

  where ρ_0 and p_0 are the reference density and pressure respectively such that $\rho(p_0) = \rho_0$.

- Momentum equation for Newtonian fluid

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \bullet (\rho \mathbf{U}\mathbf{U}) - \nabla \bullet \mu \nabla \mathbf{U} = -\nabla p \tag{3.17}$$

Boundary conditions Using FoamX the following physical boundary conditions can be set:

- `outerWall` is specified the wall condition;
- `axis` is specified as the symmetryPlane;
- `nozzle` is specified as a pressureOutlet where $p = 0$ bar.
- `front` and `back` boundaries are specified as empty.

Initial conditions $\mathbf{U} = \mathbf{0}$ m/s, $p = 100$ bar.

Transport properties

- Dynamic viscosity of water $\mu = 1.0$ mPa s

Thermodynamic properties

- Density of water $\rho = 1000$ kg/m^3
- Reference pressure $p_0 = 1$ bar
- Compressibility of water $\psi = 4.54 \times 10^{-7}$ s^2/m^2

Solver name sonicLiquidFoam: a compressible sonic laminar liquid flow code.

Case name *decompressionTank* case located in the *$FOAM_TUTORIALS/sonicLiquidFoam* directory.

3.4.2 Mesh Generation

The full geometry is modelled in this case; the set of vertices and blocks are given in the mesh description file below:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 0.1;

vertices
(
    (0 0 -0.1)
    (1 0 -0.1)
    (0 0.5 -0.1)
    (1 0.5 -0.1)
    (1.5 0.5 -0.1)
    (0 0.6 -0.1)
    (1 0.6 -0.1)
    (1.5 0.6 -0.1)
    (0 3 -0.1)
    (1 3 -0.1)
    (0 0 0.1)
    (1 0 0.1)
    (0 0.5 0.1)
    (1 0.5 0.1)
    (1.5 0.5 0.1)
    (0 0.6 0.1)
    (1 0.6 0.1)
    (1.5 0.6 0.1)
    (0 3 0.1)
    (1 3 0.1)
);

blocks
(
    hex (0 1 3 2 10 11 13 12) (30 20 1) simpleGrading (1 1 1)
    hex (2 3 6 5 12 13 16 15) (30 5 1) simpleGrading (1 1 1)
    hex (3 4 7 6 13 14 17 16) (25 5 1) simpleGrading (1 1 1)
    hex (5 6 9 8 15 16 19 18) (30 95 1) simpleGrading (1 1 1)
);

edges
(
);

patches
(
    wall outerWall
    (
        (0 1 11 10)
        (1 3 13 11)
        (3 4 14 13)
        (7 6 16 17)
        (6 9 19 16)
        (9 8 18 19)
    )
    symmetryPlane axis
    (
        (0 10 12 2)
        (2 12 15 5)
        (5 15 18 8)
    )
    patch nozzle
```

```
73      (
74          (4 7 17 14)
75      )
76      empty back
77      (
78          (0 2 3 1)
79          (2 5 6 3)
80          (3 6 7 4)
81          (5 8 9 6)
82      )
83      empty front
84      (
85          (10 11 13 12)
86          (12 13 16 15)
87          (13 14 17 16)
88          (15 16 19 18)
89      )
90  );
91
92  mergePatchPairs
93  (
94  );
95
96  // ************************************************************************* //
```

In order to improve the numerical accuracy, we shall use the reference level of 1 bar for the pressure field. Note that both the internal field level and the boundary conditions are offset by the reference level.

3.4.3 Preparing the Run

Before we commence the setup of the calculation, we need to consider the characteristic velocity of the phenomenon we are trying to capture. In the case under consideration, the fluid velocity will be very small, but the pressure wave will propagate with the speed of sound in water. The speed of sound is calculated as:

$$c = \sqrt{\frac{1}{\psi}} = \sqrt{\frac{1}{4.54 \times 10^{-7}}} = 1483.2\text{m/s}. \tag{3.18}$$

For the mesh described above, the characteristic mesh size is approximately 2 mm (note the scaling factor of 0.1 in the *blockMeshDict* file). Using

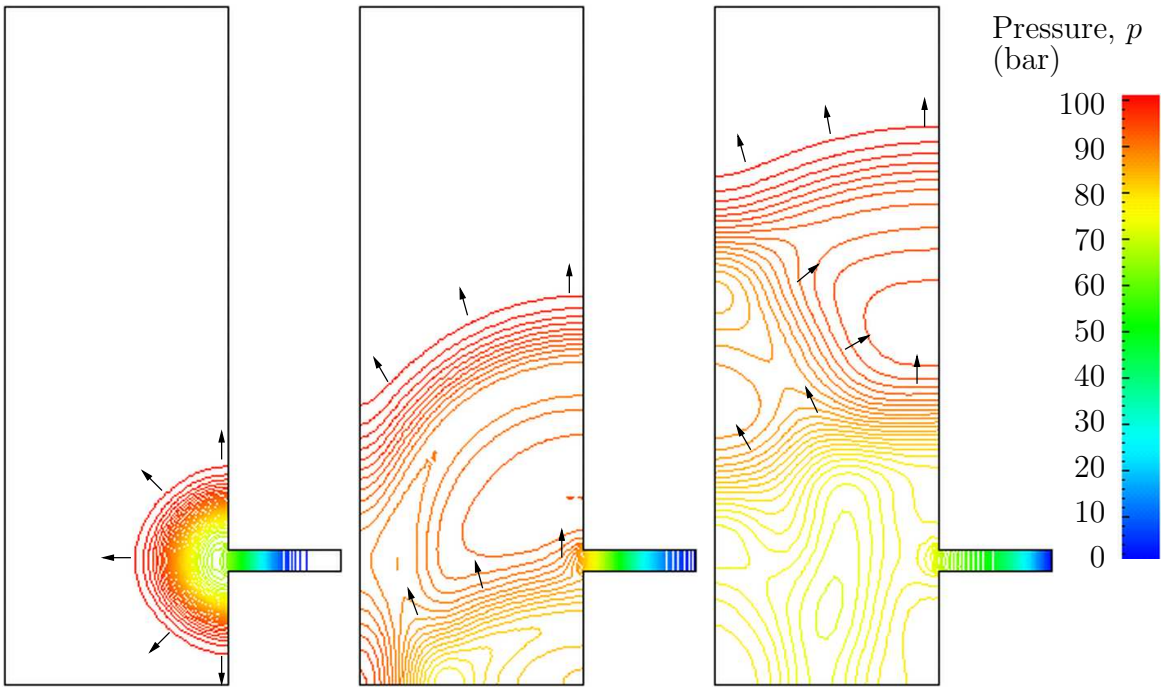
$$Co = \frac{U\,\Delta t}{\Delta x} \tag{3.19}$$

a reasonable time step is around $\Delta t = 5 \times 10^{-7}$s, giving the Co number of 0.35, based on the speed of sound. Also, note that the reported Co number by the code (associated with the convective velocity) will be two orders of magnitude smaller. As we are interested in the pressure wave propagation, we shall set the simulation time to 0.25 ms. For reference, the *controlDict* file is quoted below.

```
1   /*--------------------------------*- C++ -*----------------------------------*\
2   | =========                 |                                                 |
3   | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4   |  \\    /   O peration     | Version:  1.7.1                                 |
5   |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6   |    \\/     M anipulation  |                                                 |
7   \*---------------------------------------------------------------------------*/
8   FoamFile
9   {
10      version     2.0;
11      format      ascii;
12      class       dictionary;
13      location    "system";
```

```
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     sonicLiquidFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         0.0001;

deltaT          5e-07;

writeControl    timeStep;

writeInterval   20;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression compressed;

timeFormat      general;

timePrecision   6;

runTimeModifiable yes;


// ************************************************************************* //
```

3.4.4 Running the case



(a) At $t = 50$ µs (b) At $t = 100$ µs (c) At $t = 150$ µs

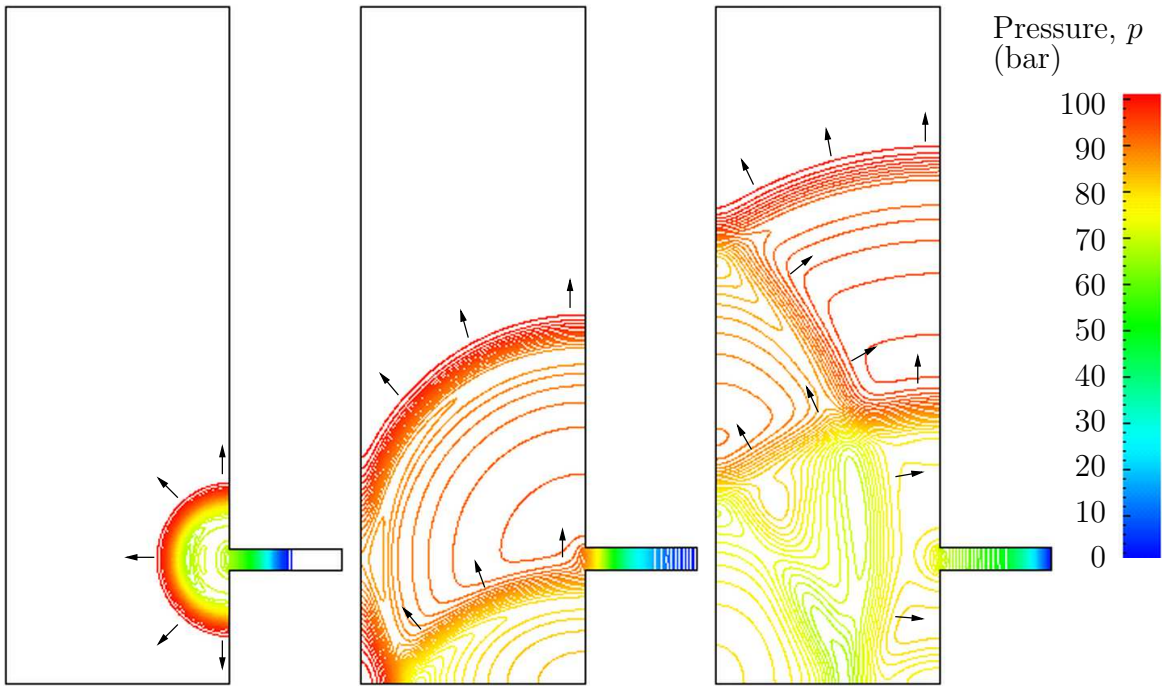
Figure 3.10: Propagation of pressure waves

The user can run the case and view results in dxFoam. The liquid flows out through

the nozzle causing a wave to move along the nozzle. As it reaches the inlet to the tank, some of the wave is transmitted into the tank and some of it is reflected. While a wave is reflected up and down the inlet pipe, the waves transmitted into the tank expand and propagate through the tank. In Figure 3.10, the pressures are shown as contours so that the wave fronts are more clearly defined than if plotted as a normal isoline plot.

If the simulation is run for a long enough time for the reflected wave to return to the pipe, we can see that negative absolute pressure is detected. The modelling permits this and has some physical basis since liquids can support tension, *i.e.* negative pressures. In reality, however, impurities or dissolved gases in liquids act as sites for cavitation, or vapourisation/boiling, of the liquid due to the low pressure. Therefore in practical situations, we generally do not observe pressures falling below the vapourisation pressure of the liquid; not at least for longer than it takes for the cavitation process to occur.

3.4.5 Improving the solution by refining the mesh



(a) At $t = 50$ µs (b) At $t = 100$ µs (c) At $t = 150$ µs

Figure 3.11: Propagation of pressure waves with refined mesh

Looking at the evolution of the resulting pressure field in time, we can clearly see the propagation of the pressure wave into the tank and numerous reflections from the inside walls. It is also obvious that the pressure wave is smeared over a number of cells. We shall now refine the mesh and reduce the time step to obtain a sharper front resolution. Simply edit the *blockMeshDict* and increase the number of cells by a factor of 4 in the x and y directions, *i.e.* block 0 becomes `(120 80 1)` from `(30 20 1)` and so on. Run blockMesh on this file. In addition, in order to maintain a Courant number below 1, the time step must be reduced accordingly to $\Delta t = 10^{-7}$ s. The second simulation gives considerably better resolution of the pressure waves as shown in Figure 3.11.

3.5 Magnetohydrodynamic flow of a liquid

In this example we shall investigate an flow of an electrically-conducting liquid through a magnetic field. The problem is one belonging to the branch of fluid dynamics known as magnetohydrodynamics (MHD) that uses mhdFoam.

3.5.1 Problem specification

The problem is known as the Hartmann problem, chosen as it contains an analytical solution with which mhdFoam can be validated. It is defined as follows:

Solution domain The domain is 2 dimensional and consists of flow along two parallel plates as shown in Fig. 3.12.

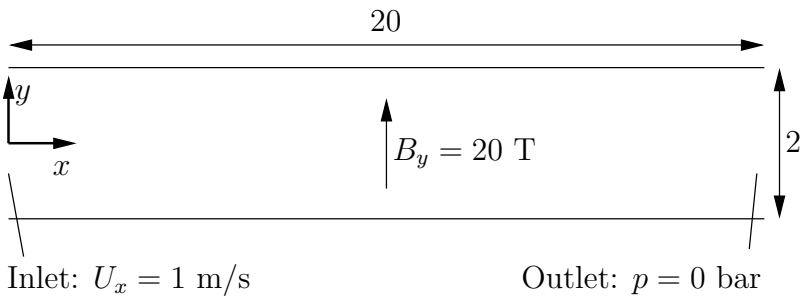


Figure 3.12: Geometry of the Hartmann problem

Governing equations

- Mass continuity for incompressible fluid

$$\nabla \cdot \mathbf{U} = 0 \tag{3.20}$$

- Momentum equation for incompressible fluid

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U}\mathbf{U}) + \nabla \cdot (2\mathbf{B}\Gamma_{\mathbf{BU}}\mathbf{B}) + \nabla \cdot (\nu \mathbf{U}) + \nabla (\Gamma_{\mathbf{BU}}\mathbf{B} : \mathbf{B}) = -\nabla p \tag{3.21}$$

  where $\mathbf{B}$ is the magnetic flux density, $\Gamma_{\mathbf{BU}} = (2\mu\rho)^{-1}$.

- Maxwell's equations

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \tag{3.22}$$

  where $\mathbf{E}$ is the electric field strength.

$$\nabla \cdot \mathbf{B} = 0 \tag{3.23}$$

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} = \mathbf{J} \tag{3.24}$$

  assuming $\partial \mathbf{D}/\partial t \ll \mathbf{J}$. Here, $\mathbf{H}$ is the magnetic field strength, $\mathbf{J}$ is the current density and $\mathbf{D}$ is the electric flux density.

- Charge continuity

$$\nabla \bullet \mathbf{J} = 0 \tag{3.25}$$

- Constitutive law

$$\mathbf{B} = \mu \mathbf{H} \tag{3.26}$$

- Ohm's law

$$\mathbf{J} = \sigma \left(\mathbf{E} + \mathbf{U} \times \mathbf{B} \right) \tag{3.27}$$

- Combining Equation 3.22, Equation 3.24, Equation 3.27, and taking the curl

$$\frac{\partial \mathbf{B}}{\partial t} + \nabla \bullet \left(\mathbf{U}\mathbf{B} \right) - \nabla \bullet \left(\phi_{\mathbf{B}} \mathbf{U} \right) - \nabla \bullet \left(\Gamma_{\mathbf{B}} \mathbf{B} \right) = 0 \tag{3.28}$$

Boundary conditions

- `inlet` is specified the inlet condition with fixed velocity $\mathbf{U} = (1, 0, 0)$ m/s;
- `outlet` is specified as the outlet with with fixed pressure $p = 0$ Pa;
- `upperWall` is specified as a wall where $\mathbf{B} = (0, 20, 0)$ T.
- `lowerWall` is specified as a wall where $\mathbf{B} = (0, 20, 0)$ T.
- `front` and `back` boundaries are specified as empty.

Initial conditions $\mathbf{U} = \mathbf{0}$ m/s, $p = 100$ Pa, $\mathbf{B} = (0, 20, 0)$ T.

Transport properties

- Kinematic viscosity $\nu = 1$ Pa s
- Density $\rho = 1$ kg m/s
- Electrical conductivity $\sigma = 1$ $(\Omega\,\mathrm{m})^{-1}$
- Permeability $\mu = 1$ H/m

Solver name mhdFoam: an incompressible laminar magneto-hydrodynamics code.

Case name *hartmann* case located in the *$FOAM_TUTORIALS/mhdFoam* directory.

3.5.2 Mesh generation

The geometry is simply modelled with 100 cells in the x-direction and 40 cells in the y-direction; the set of vertices and blocks are given in the mesh description file below:

```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  1.7.1                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
```

```
11      format      ascii;
12      class       dictionary;
13      object      blockMeshDict;
14  }
15  // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
16
17  convertToMeters 1;
18
19  vertices
20  (
21      (0 -1 0)
22      (20 -1 0)
23      (20 1 0)
24      (0 1 0)
25      (0 -1 0.1)
26      (20 -1 0.1)
27      (20 1 0.1)
28      (0 1 0.1)
29  );
30
31  blocks
32  (
33      hex (0 1 2 3 4 5 6 7) (100 40 1) simpleGrading (1 1 1)
34  );
35
36  edges
37  (
38  );
39
40  patches
41  (
42      patch inlet
43      (
44          (0 4 7 3)
45      )
46      patch outlet
47      (
48          (2 6 5 1)
49      )
50      patch lowerWall
51      (
52          (1 5 4 0)
53      )
54      patch upperWall
55      (
56          (3 7 6 2)
57      )
58      empty frontAndBack
59      (
60          (0 3 2 1)
61          (4 5 6 7)
62      )
63  );
64
65  mergePatchPairs
66  (
67  );
68
69  // ************************************************************************* //
```

3.5.3 Running the case

The user can run the case and view results in dxFoam. It is also useful at this stage to run the Ucomponents utility to convert the **U** vector field into individual scalar components. MHD flow is governed by, amongst other things, the Hartmann number which is a measure of the ratio of electromagnetic body force to viscous force

$$M = BL\sqrt{\frac{\sigma}{\rho\nu}} \tag{3.29}$$

where L is the characteristic length scale. In this case with $B_y = 20$ T, $M = 20$ and the electromagnetic body forces dominate the viscous forces. Consequently with the flow fairly

steady at $t = 2$ s the velocity profile is almost planar, viewed at a cross section midway along the domain $x = 10$ m. The user can plot a graph of the profile of U_x in dxFoam. Now the user should reduce the magnetic flux density **B** to 1 Tand re-run the code and

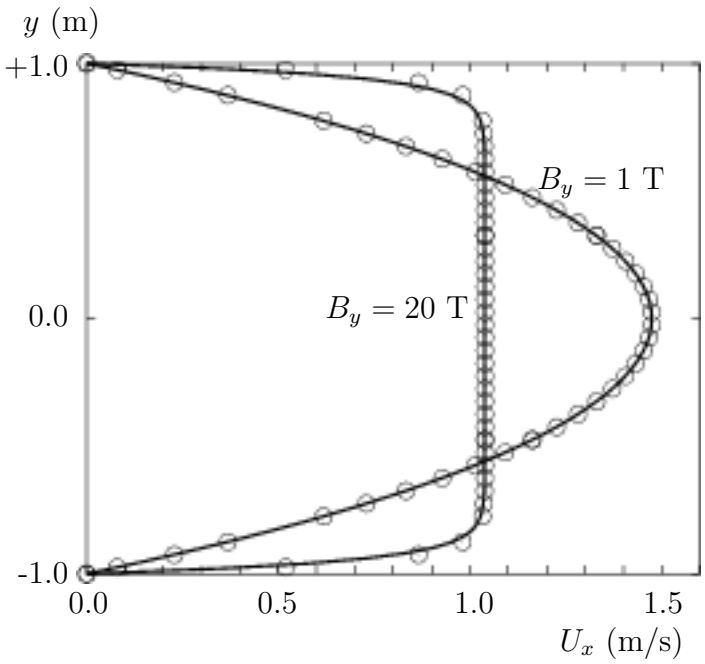


Figure 3.13: Velocity profile in the Hartmann problem for $B_y = 1$ T and $B_y = 20$ T.

Ucomponents. In this case, $M = 1$ and the electromagnetic body forces no longer dominate. The velocity profile consequently takes on the parabolic form, characteristic of Poiseuille flow as shown in Figure 3.13. To validate the code the analytical solution for the velocity profile U_x is superimposed in Figure 3.13, given by:

$$\frac{U_x(y)}{U_x(0)} = \frac{\cosh M - \cosh M(y/L)}{\cosh M - 1} \tag{3.30}$$

where the characteristic length L is half the width of the domain, *i.e.* 1 m.

Index

Symbols Numbers A B C D E F G H I J K L M N O P Q R S T U V W X Z

Symbols

*
    tensor member function, P-25
+
    tensor member function, P-25
-
    tensor member function, P-25
/
    tensor member function, P-25
/*...*/
    C++ syntax, U-81
//
    C++ syntax, U-81
    OpenFOAM file syntax, U-106
include
    C++ syntax, U-74, U-81
&
    tensor member function, P-25
&&
    tensor member function, P-25
^
    tensor member function, P-25
<LESModel>Coeffs keyword, U-190
<RASModel>Coeffs keyword, U-189
<delta>Coeffs keyword, U-190
cellSet utility, U-93
faceSet utility, U-93
pointSet utility, U-93
0.000000e+00 directory, U-106
1-dimensional mesh, U-133
1D mesh, U-133
2-dimensional mesh, U-133
2D mesh, U-133

Numbers

0 directory, U-106

A

access functions, P-23
addLayersControls keyword, U-150
adiabaticFlameT utility, U-97
adjustableRunTime
    keyword entry, U-63, U-113
adjustTimeStep keyword, U-63
agglomerator keyword, U-125
algorithms tools, U-98
alphaContactAngle
    boundary condition, U-60
analytical solution, P-45
anisotropicFilter model, U-102
Annotation window panel, U-26, U-172
ansysToFoam utility, U-92
APIfunctions model, U-101
applications, U-71
Apply button, U-168, U-172
applyBoundaryLayer utility, U-92
applyWallFunctionBoundaryConditions utility, U-92
arbitrarily unstructured, P-31
arc
    keyword entry, U-144
arc keyword, U-143
ascii
    keyword entry, U-113
attachMesh utility, U-93
Auto Accept button, U-172
autoMesh
    library, U-99
autoPatch utility, U-93
autoRefineMesh utility, U-94
axes
    right-handed, U-142
    right-handed rectangular Cartesian, P-15, U-20
axi-symmetric cases, U-138, U-148
axi-symmetric mesh, U-133

B

background
process, U-27, U-83
`backward`
keyword entry, U-121
Backward differencing, P-39
barotropicCompressibilityModels
library, U-101
basicMultiComponentMixture model, U-100, U-186
basicThermophysicalModels
library, U-99
`binary`
keyword entry, U-113
BirdCarreau model, U-103
blended differencing, P-38
block
expansion ratio, U-144
`block` keyword, U-143
blockMesh solver, P-47
blockMesh utility, U-39, U-92, U-139
blockMesh executable
vertex numbering, U-144
blockMeshDict
dictionary, U-20, U-22, U-37, U-50, U-139, U-148
`blocks` keyword, U-22, U-32, U-144
boundaries, U-133
boundary, U-133
boundary
dictionary, U-132, U-139
boundary condition
alphaContactAngle, U-60
calculated, U-140
cyclic, U-138
directionMixed, U-140
empty, P-64, P-70, U-20, U-133, U-138
fixedGradient, U-140
fixedValue, U-140
fluxCorrectedVelocity, U-141
inlet, P-70
inletOutlet, U-141
mixed, U-140
movingWallVelocity, U-141
outlet, P-70
outletInlet, U-141
partialSlip, U-141
patch, U-137
pressureDirectedInletVelocity, U-141
pressureInletVelocity, U-141
pressureOutlet, P-64
pressureTransmissive, U-141
processor, U-138
setup, U-22
slip, U-141
supersonicFreeStream, U-141
surfaceNormalFixedValue, U-141
symmetryPlane, P-64, U-138
totalPressure, U-141
turbulentInlet, U-141
wall, U-42
wall, P-64, P-70, U-60, U-137
wallBuoyantPressure, U-141
wedge, U-133, U-138, U-148
zeroGradient, U-140
boundary conditions, P-43
Dirichlet, P-43
inlet, P-44
Neumann, P-43
no-slip impermeable wall, P-44
outlet, P-44
physical, P-44
symmetry plane, P-44
`boundaryField` keyword, U-23, U-111
boundaryFoam solver, U-88
`bounded`
keyword entry, U-119, U-120
`boxToCell` keyword, U-61
boxTurb utility, U-92
breaking of a dam, U-58
bubbleFoam solver, U-89
buoyantBoussinesqPimpleFoam solver, U-90
buoyantBoussinesqSimpleFoam solver, U-90
buoyantPimpleFoam solver, U-90
buoyantSimpleFoam solver, U-90
buoyantSimpleRadiationFoam solver, U-90
button
Apply, U-168, U-172
Auto Accept, U-172
Choose Preset, U-170
Delete, U-168
Edit Color Map, U-170
Enable Line Series, U-36
Orientation Axes, U-26, U-172
Rescale to Data Range, U-27
Reset, U-168
Set Solid Color, U-171
Update GUI, U-27, U-169
Use Parallel Projection, U-26
Use parallel projection, U-171

C

C++ syntax
/*...*/, U-81
//, U-81
include, U-74, U-81
cacheAgglomeration keyword, U-125
calculated
boundary condition, U-140
cAlpha keyword, U-65
cases, U-105
castellatedMesh keyword, U-150
castellatedMeshControls
dictionary, U-151–U-153
castellatedMeshControls keyword, U-150
cavitatingFoam solver, U-89
cavity flow, U-19
CEI_ARCH
environment variable, U-177
CEI_HOME
environment variable, U-177
cell
expansion ratio, U-144
cell class, P-31
cell
keyword entry, U-179
cellLimited
keyword entry, U-119
cellPoint
keyword entry, U-179
cellPointFace
keyword entry, U-179
cells
dictionary, U-139
central differencing, P-38
cfdTools tools, U-98
cfx4ToFoam utility, U-92, U-156
changeDictionary utility, U-92
channelFoam solver, U-88
Chart Options window, U-36
checkMesh utility, U-93, U-159
chemistryModel
library, U-101
chemistryModel model, U-101
chemistrySolver model, U-101
chemkinToFoam utility, U-97
Choose Preset button, U-170
chtMultiRegionFoam solver, U-90
chtMultiRegionSimpleFoam solver, U-90
Chung
library, U-101
class
cell, P-31
dimensionSet, P-26, P-32, P-33
face, P-31
finiteVolumeCalculus, P-36
finiteVolumeMethod, P-36
fvMesh, P-31
fvSchemes, P-38
fvc, P-36
fvm, P-36
pointField, P-31
polyBoundaryMesh, P-31
polyMesh, P-31, U-129, U-131
polyPatchList, P-31
polyPatch, P-31
scalarField, P-29
scalar, P-24
slice, P-31
symmTensorField, P-29
symmTensorThirdField, P-29
tensorField, P-29
tensorThirdField, P-29
tensor, P-24
vectorField, P-29
vector, P-24, U-109
word, P-26, P-31
class keyword, U-107
clockTime
keyword entry, U-113
cloud keyword, U-180
cmptAv
tensor member function, P-25
Co utility, U-95
coalChemistryFoam solver, U-91
coalCombustion
library, U-99
cofactors
tensor member function, P-25
coldEngineFoam solver, U-90
collapseEdges utility, U-94
Color By menu, U-171
Color Legend window, U-29
Color Legend window panel, U-170
Color Scale window panel, U-170
combinePatchFaces utility, U-94
comments, U-81
compressed
keyword entry, U-114
compressibleInterFoam solver, U-89
compressibleLESModels

library, U-103
compressibleRASModels
library, U-102
constant directory, U-105, U-185
constLaminarFlameSpeed model, U-100
constTransport model, U-101, U-186
containers tools, U-98
continuum
mechanics, P-15
control
of time, U-112
controlDict
dictionary, P-66, U-24, U-33, U-43, U-53, U-63, U-105, U-164
controlDict file, P-49
convection, *see* divergence, P-38
convergence, U-41
conversion
library, U-99
`convertToMeters` keyword, U-142, U-143
coordinate
system, P-15
coordinate system, U-20
`corrected`
keyword entry, U-119, U-120
Courant number, P-42, U-24
`cpuTime`
keyword entry, U-113
Crank Nicholson
temporal discretisation, P-43
`CrankNicholson`
keyword entry, U-121
createBaffles utility, U-93
createPatch utility, U-93
createTurbulenceFields utility, U-96
cross product, *see* tensor, vector cross product
`CrossPowerLaw`
keyword entry, U-61
CrossPowerLaw model, U-103
cubeRootVolDelta model, U-102
`cubicCorrected`
keyword entry, U-121
`cubicCorrection`
keyword entry, U-118
curl, P-37
`curl`
fvc member function, P-37
`Current Time Controls` menu, U-27, U-169
`curve` keyword, U-180
cyclic
boundary condition, U-138
`cyclic`
keyword entry, U-138
cylinder
flow around a, P-45

D

`d2dt2`
fvc member function, P-37
fvm member function, P-37
dam
breaking of a, U-58
db tools, U-98
`ddt`
fvc member function, P-37
fvm member function, P-37
DeardorffDiffStress model, U-103
`debug` keyword, U-150
decomposePar utility, U-84, U-85, U-96
decomposeParDict
dictionary, U-84
decomposition
of field, U-84
of mesh, U-84
decompositionMethods
library, U-99
decompression of a tank, P-63
`defaultFieldValues` keyword, U-61
deformedGeom utility, U-93
Delete button, U-168
`delta` keyword, U-86, U-190
`deltaT` keyword, U-113
dependencies, U-74
dependency lists, U-74
`det`
tensor member function, P-25
determinant, *see* tensor, determinant
`dev`
tensor member function, P-25
`diag`
tensor member function, P-25
`diagonal`
keyword entry, U-124
`DIC`
keyword entry, U-124
`DICGaussSeidel`
keyword entry, U-124
dictionary
LESProperties, U-189
PISO, U-25

blockMeshDict, U-20, U-22, U-37, U-50, U-139, U-148
boundary, U-132, U-139
castellatedMeshControls, U-151–U-153
cells, U-139
controlDict, P-66, U-24, U-33, U-43, U-53, U-63, U-105, U-164
decomposeParDict, U-84
faces, U-131, U-139
fvSchemes, U-64, U-105, U-115
fvSolution, U-105, U-122
mechanicalProperties, U-52
neighbour, U-132
owner, U-131
points, U-131, U-139
thermalProperties, U-52, U-53
thermophysicalProperties, U-185
transportProperties, U-23, U-40, U-43
turbulenceProperties, U-43, U-62, U-189
dieselEngineFoam solver, U-90
dieselFoam solver, U-90
dieselMixture model, U-100, U-186
dieselSpray
library, U-99
differencing
Backward, P-39
blended, P-38
central, P-38
Euler implicit, P-39
Gamma, P-38
MINMOD, P-38
SUPERBEE, P-38
upwind, P-38
van Leer, P-38
`DILU`
keyword entry, U-124
dimension
checking in OpenFOAM, P-26, U-109
dimensional units, U-109
dimensioned<Type> template class, P-26
dimensionedTypes tools, U-98
`dimensions` keyword, U-23, U-111
dimensionSet class, P-26, P-32, P-33
dimensionSet tools, U-98
direct numerical simulation, U-64
directionMixed
boundary condition, U-140
directory
0.000000e+00, U-106
0, U-106
Make, U-75
constant, U-105, U-185
fluentInterface, U-174
polyMesh, U-105, U-131
processorN, U-85
run, U-105
system, P-49, U-105
tutorials, P-45, U-19
discretisation
equation, P-33
Display window panel, U-26, U-27, U-168, U-169
`distance`
keyword entry, U-154, U-180
`distributed` keyword, U-86, U-87
`div`
fvc member function, P-37
fvm member function, P-37
divergence, P-37, P-39
`divSchemes` keyword, U-115
dnsFoam solver, U-90
`doLayers` keyword, U-150
double inner product, *see* tensor,double inner product
dsmc
library, U-99
dsmcFieldsCalc utility, U-96
dsmcFoam solver, U-91
dsmcInitialise utility, U-92
`dx`
keyword entry, U-179
dynamicFvMesh
library, U-99
dynamicMesh
library, U-99
dynMixedSmagorinsky model, U-103
dynOneEqEddy model, U-103
dynSmagorinsky model, U-103

E

eConstThermo model, U-101, U-185
`edgeGrading` keyword, U-145
edgeMesh
library, U-99
`edges` keyword, U-143
`Edit` menu, U-171, U-172
Edit Color Map button, U-170
egrMixture model, U-100, U-186
electrostaticFoam solver, U-91
empty
boundary condition, P-64, P-70, U-20, U-133, U-138

empty
keyword entry, U-138
Enable Line Series button, U-36
endTime keyword, U-24, U-113
engine
library, U-99
engineCompRatio utility, U-96
engineFoam solver, U-90
engineSwirl utility, U-92
ensight74FoamExec utility, U-176
ENSIGHT7_INPUT
environment variable, U-177
ENSIGHT7_READER
environment variable, U-177
ensightFoamReader utility, U-94
enstrophy utility, U-95
environment variable
CEI_ARCH, U-177
CEI_HOME, U-177
ENSIGHT7_INPUT, U-177
ENSIGHT7_READER, U-177
FOAM_RUN, U-105
WM_ARCH_OPTION, U-78
WM_ARCH, U-78
WM_COMPILER_BIN, U-78
WM_COMPILER_DIR, U-78
WM_COMPILER_LIB, U-78
WM_COMPILER, U-78
WM_COMPILE_OPTION, U-78
WM_DIR, U-78
WM_JAVAC_OPTION, U-78
WM_LINK_LANGUAGE, U-78
WM_MPLIB, U-78
WM_OPTIONS, U-78
WM_PRECISION_OPTION, U-78
WM_PROJECT_DIR, U-78
WM_PROJECT_INST_DIR, U-78
WM_PROJECT_LANGUAGE, U-78
WM_PROJECT_USER_DIR, U-78
WM_PROJECT_VERSION, U-78
WM_PROJECT, U-78
wmake, U-78
ePsiThermo model, U-100, U-186
equilibriumCO utility, U-97
equilibriumFlameT utility, U-97
errorEstimation
library, U-99
errorReduction keyword, U-158
estimateScalarError utility, U-97
Euler
keyword entry, U-121
Euler implicit
differencing, P-39
temporal discretisation, P-42
examples
decompression of a tank, P-63
flow around a cylinder, P-45
flow over backward step, P-54
Hartmann problem, P-69
supersonic flow over forward step, P-59
execFlowFunctionObjects utility, U-96
expandDictionary utility, U-97
expansionRatio keyword, U-157
explicit
temporal discretisation, P-42
exponential model, U-101
extrude2DMesh utility, U-92
extrudeMesh utility, U-92

F

face class, P-31
face keyword, U-180
faceAreaPair
keyword entry, U-125
faceLimited
keyword entry, U-119
faces
dictionary, U-131, U-139
FDIC
keyword entry, U-124
featureAngle keyword, U-157
features keyword, U-151, U-152
field
U, U-24
p, U-24
decomposition, U-84
FieldField<Type> template class, P-32
fieldFunctionObjects
library, U-98
fields, P-29
mapping, U-164
fields tools, U-98
fields keyword, U-179
Field<Type> template class, P-29
fieldValues keyword, U-61
fieldview9Reader utility, U-94
file
Make/files, U-77
controlDict, P-49
files, U-75
g, U-61

options, U-75
snappyHexMeshDict, U-149
transportProperties, U-61
file format, U-106
files file, U-75
`finalLayerRatio` keyword, U-157
financialFoam solver, U-91
finite volume
discretisation, P-27
mesh, P-31
finiteVolume
library, U-98
finiteVolume tools, U-98
finiteVolumeCalculus class, P-36
finiteVolumeMethod class, P-36
fireFoam solver, U-90
`firstTime` keyword, U-113
`fixed`
keyword entry, U-114
fixedGradient
boundary condition, U-140
fixedValue
boundary condition, U-140
flattenMesh utility, U-93
flow
free surface, U-58
laminar, U-19
steady, turbulent, P-54
supersonic, P-60
turbulent, U-19
flow around a cylinder, P-45
flow over backward step, P-54
flowType utility, U-95
fluent3DMeshToFoam utility, U-92
fluentInterface directory, U-174
fluentMeshToFoam utility, U-92, U-156
fluxCorrectedVelocity
boundary condition, U-141
`fluxRequired` keyword, U-115
OpenFOAM
cases, U-105
FOAM_RUN
environment variable, U-105
foamCalc utility, U-34
foamCalcFunctions
library, U-98
foamCorrectVrt script/alias, U-163
foamDataToFluent utility, U-94, U-174
foamDebugSwitches utility, U-97
`FoamFile` keyword, U-107
`foamFile`
keyword entry, U-179
foamFormatConvert utility, U-97
foamInfoExec utility, U-97
foamJob script/alias, U-181
foamLog script/alias, U-182
foamMeshToFluent utility, U-92, U-174
foamToEnsight utility, U-94
foamToEnsightParts utility, U-94
foamToFieldview9 utility, U-95
foamToGMV utility, U-95
foamToStarMesh utility, U-92
foamToTecplot360 utility, U-95
foamToVTK utility, U-95
foamUpgradeFvSolution utility, U-92
forces
library, U-98
foreground
process, U-27
`format` keyword, U-107
`fourth`
keyword entry, U-119, U-120
`functions` keyword, U-114
fvc class, P-36
fvc member function
`curl`, P-37
`d2dt2`, P-37
`ddt`, P-37
`div`, P-37
`gGrad`, P-37
`grad`, P-37
`laplacian`, P-37
`lsGrad`, P-37
`snGrad`, P-37
`snGradCorrection`, P-37
`sqrGradGrad`, P-37
fvDOM
library, U-100
fvm class, P-36
fvm member function
`d2dt2`, P-37
`ddt`, P-37
`div`, P-37
`laplacian`, P-37
`Su`, P-37
`SuSp`, P-37
fvMatrices tools, U-98
fvMatrix template class, P-36
fvMesh class, P-31
fvMesh tools, U-98

fvMotionSolver
library, U-99
fvSchemes
dictionary, U-64, U-105, U-115
fvSchemes class, P-38
`fvSchemes`
menu entry, U-54
fvSolution
dictionary, U-105, U-122

G

g file, U-61
gambitToFoam utility, U-93, U-156
`GAMG`
keyword entry, U-54, U-123, U-124
`Gamma`
keyword entry, U-118
Gamma differencing, P-38
`Gauss`
keyword entry, U-119
Gauss's theorem, P-36
`GaussSeidel`
keyword entry, U-124
General window panel, U-171, U-172
general model, U-101
`general`
keyword entry, U-114
geometric-algebraic multi-grid, U-125
GeometricBoundaryField template class, P-32
geometricField<Type> template class, P-32
`geometry` keyword, U-150
`gGrad`
fvc member function, P-37
global tools, U-98
gmshToFoam utility, U-93
`gnuplot`
keyword entry, U-114, U-179
`grad`
fvc member function, P-37
(Grad Grad) squared, P-37
gradient, P-37, P-40
Gauss scheme, P-40
Gauss's theorem, U-54
least square fit, U-54
least squares method, P-40, U-54
surface normal, P-40
`gradSchemes` keyword, U-115
graph tools, U-98
`graphFormat` keyword, U-114
GuldersEGRLaminarFlameSpeed model, U-101
GuldersLaminarFlameSpeed model, U-100

H

hConstThermo model, U-101, U-185
`Help` menu, U-171
HerschelBulkley model, U-103
hhuMixtureThermo model, U-100, U-187
`hierarchical`
keyword entry, U-85, U-86
homogeneousMixture model, U-100, U-186
hPolynomialThermo model, U-101, U-185
hPsiMixtureThermo model, U-100, U-186
hPsiThermo model, U-99, U-186
hRhoMixtureThermo model, U-100, U-187
hRhoThermo model, U-100, U-186
hsPsiMixtureThermo model, U-100, U-186
hsPsiThermo model, U-99, U-186
hsRhoMixtureThermo model, U-100, U-187

I

`I`
tensor member function, P-25
icoErrorEstimate utility, U-97
icoFoam solver, U-19, U-23, U-24, U-27, U-88
icoMomentError utility, U-97
icoPolynomial model, U-101, U-185
ideasToFoam utility, U-156
ideasUnvToFoam utility, U-93
identities, *see* tensor, identities
identity, *see* tensor, identity
IFCLookUpTableGen utility, U-97
incompressibleLESModels
library, U-103
incompressibleRASModels
library, U-102
incompressibleTransportModels
library, P-55, U-103
incompressibleTurbulenceModels
library, P-55
index
notation, P-16, P-17
Information window panel, U-168
inhomogeneousMixture model, U-100, U-186
inlet
boundary condition, P-70
inletOutlet
boundary condition, U-141
inner product, *see* tensor, inner product
`inside`
keyword entry, U-153
insideCells utility, U-93
interfaceProperties model, U-103
interFoam solver, U-89

interMixingFoam solver, U-89
internalField keyword, U-23, U-111
interPhaseChangeFoam solver, U-90
interpolation tools, U-98
interpolationScheme keyword, U-179
interpolations tools, U-98
interpolationSchemes keyword, U-115
inv
    tensor member function, P-25

J

janafThermo model, U-101, U-186
jplot
    keyword entry, U-114, U-179

K

kEpsilon model, U-102
keyword
    FoamFile, U-107
    LESModel, U-190
    RASModel, U-189
    addLayersControls, U-150
    adjustTimeStep, U-63
    agglomerator, U-125
    arc, U-143
    blocks, U-22, U-32, U-144
    block, U-143
    boundaryField, U-23, U-111
    boxToCell, U-61
    cAlpha, U-65
    cacheAgglomeration, U-125
    castellatedMeshControls, U-150
    castellatedMesh, U-150
    class, U-107
    cloud, U-180
    convertToMeters, U-142, U-143
    curve, U-180
    debug, U-150
    defaultFieldValues, U-61
    deltaT, U-113
    delta, U-86, U-190
    dimensions, U-23, U-111
    distributed, U-86, U-87
    divSchemes, U-115
    doLayers, U-150
    edgeGrading, U-145
    edges, U-143
    endTime, U-24, U-113
    errorReduction, U-158
    expansionRatio, U-157
    face, U-180
    featureAngle, U-157
    features, U-151, U-152
    fieldValues, U-61
    fields, U-179
    finalLayerRatio, U-157
    firstTime, U-113
    fluxRequired, U-115
    format, U-107
    functions, U-114
    geometry, U-150
    gradSchemes, U-115
    graphFormat, U-114
    internalField, U-23, U-111
    interpolationSchemes, U-115
    interpolationScheme, U-179
    laplacianSchemes, U-115
    latestTime, U-40
    layers, U-157
    leastSquares, U-54
    levels, U-154
    libs, U-82, U-114
    locationInMesh, U-151, U-153
    location, U-107
    manualCoeffs, U-86
    maxAlphaCo, U-63
    maxBoundarySkewness, U-158
    maxConcave, U-158
    maxCo, U-63
    maxDeltaT, U-63
    maxFaceThicknessRatio, U-157
    maxGlobalCells, U-151
    maxInternalSkewness, U-158
    maxLocalCells, U-151
    maxNonOrtho, U-158
    maxThicknessToMedialRatio, U-157
    mergeLevels, U-125
    mergePatchPairs, U-143
    mergeTolerance, U-150
    meshQualityControls, U-150
    method, U-86
    metisCoeffs, U-86
    midPointAndFace, U-180
    midPoint, U-180
    minArea, U-158
    minDeterminant, U-158
    minFaceWeight, U-158
    minFlatness, U-158
    minMedianAxisAngle, U-157
    minRefinementCells, U-151
    minThickness, U-157

minTriangleTwist, U-158
minTwist, U-158
minVolRatio, U-158
minVol, U-158
mode, U-153
nAlphaSubCycles, U-65
nBufferCellsNoExtrude, U-157
nCellsBetweenLevels, U-151
nFaces, U-132
nFinestSweeps, U-125
nGrow, U-157
nLayerIter, U-157
nPostSweeps, U-125
nPreSweeps, U-125
nRelaxIter, U-155, U-157
nRelaxedIter, U-157
nSmoothNormals, U-157
nSmoothPatch, U-155
nSmoothScale, U-158
nSmoothSurfaceNormals, U-157
nSmoothThickness, U-157
nSolveIter, U-155
numberOfSubdomains, U-86
n, U-86
object, U-107
order, U-86
pRefCell, U-25, U-127
pRefValue, U-25, U-127
p_rhgRefCell, U-127
p_rhgRefValue, U-127
patchMap, U-165
patches, U-143, U-145
preconditioner, U-123, U-124
pressure, U-52
printCeoffs, U-189
printCoeffs, U-43
processorWeights, U-85
processorWeights, U-86
purgeWrite, U-113
refGradient, U-140
refinementRegions, U-151, U-154
refinementSurfaces, U-151, U-152
refinementRegions, U-153
regions, U-61
relTol, U-54, U-123, U-124
relativeSizes, U-157
relaxed, U-158
resolveFeatureAngle, U-151, U-152
roots, U-86, U-87
runTimeModifiable, U-114
scotchCoeffs, U-86
setFormat, U-179
sets, U-179
simpleGrading, U-144
simulationType, U-43, U-62, U-189
smoother, U-125
snGradSchemes, U-115
snapControls, U-150
snap, U-150
solvers, U-123
solver, U-54, U-123
spline, U-143
startFace, U-132
startFrom, U-24, U-113
startTime, U-24, U-113
stopAt, U-113
strategy, U-85, U-86
surfaceFormat, U-179
surfaces, U-179
thermoType, U-185
timeFormat, U-114
timePrecision, U-114
timeScheme, U-115
tolerance, U-54, U-123, U-124, U-155
topoSetSource, U-61
traction, U-52
turbulence, U-189
type, U-133, U-136
uniform, U-180
valueFraction, U-140
value, U-23, U-140
version, U-107
vertices, U-22, U-143
writeCompression, U-114
writeControl, U-24, U-63, U-113
writeFormat, U-57, U-113
writeInterval, U-24, U-33, U-113
writePrecision, U-113
<LESModel>Coeffs, U-190
<RASModel>Coeffs, U-189
<delta>Coeffs, U-190

keyword entry
CrankNicholson, U-121
CrossPowerLaw, U-61
DICGaussSeidel, U-124
DIC, U-124
DILU, U-124
Euler, U-121
FDIC, U-124
GAMG, U-54, U-123, U-124

Gamma, U-118
GaussSeidel, U-124
Gauss, U-119
LESModel, U-43, U-189
MGridGen, U-125
MUSCL, U-118
Newtonian, U-61
PBiCG, U-123
PCG, U-123
QUICK, U-118, U-121
RASModel, U-43, U-189
SFCD, U-118, U-121
UMIST, U-117
adjustableRunTime, U-63, U-113
arc, U-144
ascii, U-113
backward, U-121
binary, U-113
bounded, U-119, U-120
cellLimited, U-119
cellPointFace, U-179
cellPoint, U-179
cell, U-179
clockTime, U-113
compressed, U-114
corrected, U-119, U-120
cpuTime, U-113
cubicCorrected, U-121
cubicCorrection, U-118
cyclic, U-138
diagonal, U-124
distance, U-154, U-180
dx, U-179
empty, U-138
faceAreaPair, U-125
faceLimited, U-119
fixed, U-114
foamFile, U-179
fourth, U-119, U-120
general, U-114
gnuplot, U-114, U-179
hierarchical, U-85, U-86
inside, U-153
jplot, U-114, U-179
laminar, U-43, U-189
latestTime, U-113
leastSquares, U-119
limitedCubic, U-118
limitedLinear, U-118
limited, U-119, U-120
linearUpwind, U-118, U-121
linear, U-118, U-121
line, U-144
manual, U-85, U-86
metis, U-85, U-86
midPoint, U-118
nextWrite, U-113
noWriteNow, U-113
none, U-116, U-124
null, U-179
outside, U-153
patch, U-138, U-180
polyLine, U-144
polySpline, U-144
processor, U-138
raw, U-114, U-179
runTime, U-33, U-113
scientific, U-114
scotch, U-85, U-86
simpleSpline, U-144
simple, U-85, U-86
skewLinear, U-118, U-121
smoothSolver, U-123
startTime, U-24, U-113
steadyState, U-121
stl, U-179
symmetryPlane, U-138
timeStep, U-24, U-33, U-113
uncompressed, U-114
uncorrected, U-119, U-120
upwind, U-118, U-121
vanLeer, U-118
vtk, U-179
wall, U-138
wedge, U-138
writeControl, U-113
writeNow, U-113
xmgr, U-114, U-179
xyz, U-180
x, U-180
y, U-180
z, U-180
kivaToFoam utility, U-93
kOmega model, U-102
kOmegaSST model, U-102, U-103
Kronecker delta, P-21

L

lagrangian
library, U-99
lagrangianIntermediate

library, U-99
Lambda2 utility, U-95
LamBremhorstKE model, U-102
laminar model, U-102
laminar
keyword entry, U-43, U-189
laminarFlameSpeedModels
library, U-100
laplaceFilter model, U-102
Laplacian, P-38
laplacian, P-37
laplacian
fvc member function, P-37
fvm member function, P-37
laplacianFoam solver, U-88
laplacianSchemes keyword, U-115
latestTime
keyword entry, U-113
latestTime keyword, U-40
LaunderGibsonRSTM model, U-102
LaunderSharmaKE model, U-102
layers keyword, U-157
leastSquares
keyword entry, U-119
leastSquares keyword, U-54
LESdeltas
library, U-102
LESfilters
library, U-102
LESModel
keyword entry, U-43, U-189
LESModel keyword, U-190
LESProperties
dictionary, U-189
levels keyword, U-154
libraries, U-71
library
Chung, U-101
LESdeltas, U-102
LESfilters, U-102
MGridGenGAMGAgglomeration, U-99
ODE, U-99
OSspecific, U-99
OpenFOAM, U-98
P1, U-100
PV3FoamReader, U-167
PVFoamReader, U-167
Wallis, U-101
autoMesh, U-99
barotropicCompressibilityModels, U-101
basicThermophysicalModels, U-99
chemistryModel, U-101
coalCombustion, U-99
compressibleLESModels, U-103
compressibleRASModels, U-102
conversion, U-99
decompositionMethods, U-99
dieselSpray, U-99
dsmc, U-99
dynamicFvMesh, U-99
dynamicMesh, U-99
edgeMesh, U-99
engine, U-99
errorEstimation, U-99
fieldFunctionObjects, U-98
finiteVolume, U-98
foamCalcFunctions, U-98
forces, U-98
fvDOM, U-100
fvMotionSolver, U-99
incompressibleLESModels, U-103
incompressibleRASModels, U-102
incompressibleTransportModels, P-55, U-103
incompressibleTurbulenceModels, P-55
lagrangianIntermediate, U-99
lagrangian, U-99
laminarFlameSpeedModels, U-100
linear, U-101
liquidMixture, U-102
liquids, U-102
meshTools, U-99
molecularMeasurements, U-99
molecule, U-99
pdf, U-101
postCalc, U-98
potential, U-99
primitive, P-23
radiation, U-100
randomProcesses, U-99
reactionThermophysicalModels, U-100
sampling, U-98
solidMixture, U-102
solidParticle, U-99
solids, U-102
specie, U-101
surfMesh, U-99
systemCall, U-98
thermophysicalFunctions, U-101
thermophysical, U-185
topoChangerFvMesh, U-99

triSurface, U-99
utilityFunctionObjects, U-98
vtkFoam, U-167
vtkPV3Foam, U-167
libs keyword, U-82, U-114
lid-driven cavity flow, U-19
LienCubicKE model, U-102
LienCubicKELowRe model, U-102
LienLeschzinerLowRe model, U-102
Lights window panel, U-171
limited
keyword entry, U-119, U-120
limitedCubic
keyword entry, U-118
limitedLinear
keyword entry, U-118
line
keyword entry, U-144
Line Style menu, U-36
linear
library, U-101
linear
keyword entry, U-118, U-121
linearUpwind
keyword entry, U-118, U-121
liquid
electrically-conducting, P-69
liquidMixture
library, U-102
liquids
library, U-102
lists, P-29
List<Type> template class, P-29
location keyword, U-107
locationInMesh keyword, U-151, U-153
locDynOneEqEddy model, U-103
lowReOneEqEddy model, U-103
LRDDiffStress model, U-103
LRR model, U-102
lsGrad
fvc member function, P-37

M

Mach utility, U-95
mag
tensor member function, P-25
magnetohydrodynamics, P-69
magSqr
tensor member function, P-25
Make directory, U-75
make script/alias, U-73
Make/files file, U-77
manual
keyword entry, U-85, U-86
manualCoeffs keyword, U-86
mapFields utility, U-33, U-40, U-44, U-57, U-92, U-164
mapping
fields, U-164
Marker Style menu, U-37
matrices tools, U-98
max
tensor member function, P-25
maxAlphaCo keyword, U-63
maxBoundarySkewness keyword, U-158
maxCo keyword, U-63
maxConcave keyword, U-158
maxDeltaT keyword, U-63
maxFaceThicknessRatio keyword, U-157
maxGlobalCells keyword, U-151
maxInternalSkewness keyword, U-158
maxLocalCells keyword, U-151
maxNonOrtho keyword, U-158
maxThicknessToMedialRatio keyword, U-157
mdEquilibrationFoam solver, U-91
mdFoam solver, U-91
mdInitialise utility, U-92
mechanicalProperties
dictionary, U-52
memory tools, U-98
menu
Color By, U-171
Current Time Controls, U-27, U-169
Edit, U-171, U-172
Help, U-171
Line Style, U-36
Marker Style, U-37
Plot Type, U-36
VCR Controls, U-27, U-169
View, U-171
menu entry
Plot Over Line, U-36
Save Animation, U-173
Save Screenshot, U-173
Settings, U-172
Show Color Legend, U-27
Solid Color, U-171
Toolbars, U-171
View Settings..., U-26
View Settings, U-26, U-171
Wireframe, U-171

fvSchemes, U-54
mergeLevels keyword, U-125
mergeMeshes utility, U-93
mergeOrSplitBaffles utility, U-93
mergePatchPairs keyword, U-143
mergeTolerance keyword, U-150
mesh
1-dimensional, U-133
1D, U-133
2-dimensional, U-133
2D, U-133
axi-symmetric, U-133
basic, P-31
block structured, U-139
decomposition, U-84
description, U-129
finite volume, P-31
generation, U-139, U-148
grading, U-139, U-144
grading, example of, P-54
non-orthogonal, P-45
refinement, P-63
resolution, U-32
specification, U-129
split-hex, U-149
Stereolithography (STL), U-149
surface, U-149
validity constraints, U-129
meshes tools, U-98
meshQualityControls keyword, U-150
meshTools
library, U-99
message passing interface
openMPI, U-85
method keyword, U-86
metis
keyword entry, U-85, U-86
metisCoeffs keyword, U-86
MGridGenGAMGAgglomeration
library, U-99
MGridGen
keyword entry, U-125
mhdFoam solver, P-70, U-91
midPoint
keyword entry, U-118
midPoint keyword, U-180
midPointAndFace keyword, U-180
min
tensor member function, P-25
minArea keyword, U-158
minDeterminant keyword, U-158
minFaceWeight keyword, U-158
minFlatness keyword, U-158
minMedianAxisAngle keyword, U-157
MINMOD differencing, P-38
minRefinementCells keyword, U-151
minThickness keyword, U-157
minTriangleTwist keyword, U-158
minTwist keyword, U-158
minVol keyword, U-158
minVolRatio keyword, U-158
mirrorMesh utility, U-93
mixed
boundary condition, U-140
mixedSmagorinsky model, U-103
mixtureAdiabaticFlameT utility, U-97
mode keyword, U-153
model
APIfunctions, U-101
BirdCarreau, U-103
CrossPowerLaw, U-103
DeardorffDiffStress, U-103
GuldersEGRLaminarFlameSpeed, U-101
GuldersLaminarFlameSpeed, U-100
HerschelBulkley, U-103
LRDDiffStress, U-103
LRR, U-102
LamBremhorstKE, U-102
LaunderGibsonRSTM, U-102
LaunderSharmaKE, U-102
LienCubicKELowRe, U-102
LienCubicKE, U-102
LienLeschzinerLowRe, U-102
NSRDSfunctions, U-101
Newtonian, U-103
NonlinearKEShih, U-102
PrandtlDelta, U-102
RNGkEpsilon, U-102
RosinRammler, U-101
Smagorinsky2, U-103
Smagorinsky, U-103
SpalartAllmarasDDES, U-103
SpalartAllmarasIDDES, U-103
SpalartAllmaras, U-102, U-103
anisotropicFilter, U-102
basicMultiComponentMixture, U-100, U-186
chemistryModel, U-101
chemistrySolver, U-101
constLaminarFlameSpeed, U-100
constTransport, U-101, U-186

cubeRootVolDelta, U-102
dieselMixture, U-100, U-186
dynMixedSmagorinsky, U-103
dynOneEqEddy, U-103
dynSmagorinsky, U-103
eConstThermo, U-101, U-185
ePsiThermo, U-100, U-186
egrMixture, U-100, U-186
exponential, U-101
general, U-101
hConstThermo, U-101, U-185
hPolynomialThermo, U-101, U-185
hPsiMixtureThermo, U-100, U-186
hPsiThermo, U-99, U-186
hRhoMixtureThermo, U-100, U-187
hRhoThermo, U-100, U-186
hhuMixtureThermo, U-100, U-187
homogeneousMixture, U-100, U-186
hsPsiMixtureThermo, U-100, U-186
hsPsiThermo, U-99, U-186
hsRhoMixtureThermo, U-100, U-187
icoPolynomial, U-101, U-185
inhomogeneousMixture, U-100, U-186
interfaceProperties, U-103
janafThermo, U-101, U-186
kEpsilon, U-102
kOmegaSST, U-102, U-103
kOmega, U-102
laminar, U-102
laplaceFilter, U-102
locDynOneEqEddy, U-103
lowReOneEqEddy, U-103
mixedSmagorinsky, U-103
multiComponentMixture, U-100, U-186
normal, U-101
oneEqEddy, U-103
perfectGas, U-101, U-185
polynomialTransport, U-101, U-186
powerLaw, U-103
pureMixture, U-100, U-186
qZeta, U-102
reactingMixture, U-100, U-186
realizableKE, U-102
scaleSimilarity, U-103
simpleFilter, U-102
smoothDelta, U-103
specieThermo, U-101, U-186
spectEddyVisc, U-103
sutherlandTransport, U-101, U-186
uniform, U-101
veryInhomogeneousMixture, U-100, U-186
modifyMesh utility, U-94
molecularMeasurements
library, U-99
molecule
library, U-99
momentScalarError utility, U-97
moveDynamicMesh utility, U-93
moveEngineMesh utility, U-93
moveMesh utility, U-93
movingWallVelocity
boundary condition, U-141
MPI
openMPI, U-85
mshToFoam utility, U-93
multiComponentMixture model, U-100, U-186
multigrid
geometric-algebraic, U-125
multiphaseInterFoam solver, U-90
MUSCL
keyword entry, U-118

N

n keyword, U-86
nabla
operator, P-27
nAlphaSubCycles keyword, U-65
nBufferCellsNoExtrude keyword, U-157
nCellsBetweenLevels keyword, U-151
neighbour
dictionary, U-132
netgenNeutralToFoam utility, U-93
Newtonian
keyword entry, U-61
Newtonian model, U-103
nextWrite
keyword entry, U-113
nFaces keyword, U-132
nFinestSweeps keyword, U-125
nGrow keyword, U-157
nLayerIter keyword, U-157
non-orthogonal mesh, P-45
none
keyword entry, U-116, U-124
NonlinearKEShih model, U-102
nonNewtonianIcoFoam solver, U-88
normal model, U-101
noWriteNow
keyword entry, U-113
nPostSweeps keyword, U-125
nPreSweeps keyword, U-125

nRelaxedIter keyword, U-157
nRelaxIter keyword, U-155, U-157
nSmoothNormals keyword, U-157
nSmoothPatch keyword, U-155
nSmoothScale keyword, U-158
nSmoothSurfaceNormals keyword, U-157
nSmoothThickness keyword, U-157
nSolveIter keyword, U-155
NSRDSfunctions model, U-101
null
    keyword entry, U-179
numberOfSubdomains keyword, U-86

O

object keyword, U-107
objToVTK utility, U-93
ODE
    library, U-99
oneEqEddy model, U-103
Opacity text box, U-171
OpenFOAM
    applications, U-71
    file format, U-106
    libraries, U-71
OpenFOAM
    library, U-98
OpenFOAM file syntax
    //, U-106
openMPI
    message passing interface, U-85
    MPI, U-85
operator
    scalar, P-28
    vector, P-27
Options window, U-172
options file, U-75
order keyword, U-86
Orientation Axes button, U-26, U-172
OSspecific
    library, U-99
outer product, *see* tensor, outer product
outlet
    boundary condition, P-70
outletInlet
    boundary condition, U-141
outside
    keyword entry, U-153
owner
    dictionary, U-131

P

p field, U-24
P1
    library, U-100
p_rhgRefCell keyword, U-127
p_rhgRefValue keyword, U-127
paraFoam, U-25, U-167
parallel
    running, U-84
partialSlip
    boundary condition, U-141
particleTracks utility, U-96
patch
    boundary condition, U-137
patch
    keyword entry, U-138, U-180
patchAverage utility, U-96
patches keyword, U-143, U-145
patchIntegrate utility, U-96
patchMap keyword, U-165
patchSummary utility, U-97
PBiCG
    keyword entry, U-123
PCG
    keyword entry, U-123
pdf
    library, U-101
pdfPlot utility, U-96
PDRFoam solver, U-90
Pe utility, U-95
perfectGas model, U-101, U-185
permutation symbol, P-20
pimpleDyMFoam solver, U-88
pimpleFoam solver, U-89
Pipeline Browser window, U-25, U-168
PISO
    dictionary, U-25
pisoFoam solver, U-19, U-89
Plot Over Line
    menu entry, U-36
Plot Type menu, U-36
plot3dToFoam utility, U-93
pointField class, P-31
pointField<Type> template class, P-33
points
    dictionary, U-131, U-139
polyBoundaryMesh class, P-31
polyDualMesh utility, U-93
polyLine
    keyword entry, U-144

polyMesh directory, U-105, U-131
polyMesh class, P-31, U-129, U-131
polynomialTransport model, U-101, U-186
polyPatch class, P-31
polyPatchList class, P-31
polySpline
keyword entry, U-144
porousExplicitSourceReactingParcelFoam solver, U-91
porousSimpleFoam solver, U-89
post-processing, U-167
post-processing
paraFoam, U-167
postCalc
library, U-98
postChannel utility, U-96
potential
library, U-99
potentialFoam solver, P-46, U-88
pow
tensor member function, P-25
powerLaw model, U-103
pPrime2 utility, U-95
PrandtlDelta model, U-102
preconditioner keyword, U-123, U-124
pRefCell keyword, U-25, U-127
pRefValue keyword, U-25, U-127
pressure keyword, U-52
pressure waves
in liquids, P-63
pressureDirectedInletVelocity
boundary condition, U-141
pressureInletVelocity
boundary condition, U-141
pressureOutlet
boundary condition, P-64
pressureTransmissive
boundary condition, U-141
primitive
library, P-23
primitives tools, U-98
printCeoffs keyword, U-189
printCoeffs keyword, U-43
processorWeights keyword, U-85
probeLocations utility, U-96
process
background, U-27, U-83
foreground, U-27
processor
boundary condition, U-138
processor
keyword entry, U-138
processorN directory, U-85
processorWeights keyword, U-86
Properties window panel, U-27, U-168, U-169
ptot utility, U-96
pureMixture model, U-100, U-186
purgeWrite keyword, U-113
PV3FoamReader
library, U-167
PV3FoamReader utility, U-94
PVFoamReader
library, U-167
PVFoamReader utility, U-94

Q

Q utility, U-95
QUICK
keyword entry, U-118, U-121
qZeta model, U-102

R

R utility, U-96
radiation
library, U-100
randomProcesses
library, U-99
RASModel
keyword entry, U-43, U-189
RASModel keyword, U-189
raw
keyword entry, U-114, U-179
reactingFoam solver, U-90
reactingMixture model, U-100, U-186
reactingParcelFoam solver, U-91
reactionThermophysicalModels
library, U-100
realizableKE model, U-102
reconstructPar utility, U-88, U-96
reconstructParMesh utility, U-96
redistributeMeshPar utility, U-96
refGradient keyword, U-140
refineHexMesh utility, U-94
refinementRegions keyword, U-153
refinementLevel utility, U-94
refinementRegions keyword, U-151, U-154
refinementSurfaces keyword, U-151, U-152
refineMesh utility, U-93
refineWallLayer utility, U-94
Region Status window panel, U-26
regions keyword, U-61

relative tolerance, U-124
relativeSizes keyword, U-157
relaxed keyword, U-158
relTol keyword, U-54, U-123, U-124
removeFaces utility, U-94
Render View window, U-172
Render View window panel, U-172
Render View Options window, U-171
renumberMesh utility, U-94
Rescale to Data Range button, U-27
Reset button, U-168
resolveFeatureAngle keyword, U-151, U-152
restart, U-40
Reynolds number, U-19, U-23
rhoPorousMRFPimpleFoam solver, U-89
rhoCentralFoam solver, U-89
rhoPisoFoam solver, U-89
rhoPimpleFoam solver, U-89
rhoPorousSimpleFoam solver, U-89
rhoReactingFoam solver, U-90
rhoSimpleFoam solver, U-89
rmdepall script/alias, U-79
RNGkEpsilon model, U-102
roots keyword, U-86, U-87
RosinRammler model, U-101
rotateMesh utility, U-94
run
    parallel, U-84
run directory, U-105
runTime
    keyword entry, U-33, U-113
runTimeModifiable keyword, U-114

S

sammToFoam utility, U-93
sample utility, U-96, U-178
sampling
    library, U-98
Save Animation
    menu entry, U-173
Save Screenshot
    menu entry, U-173
scalar, P-16
    operator, P-28
scalar class, P-24
scalarField class, P-29
scalarTransportFoam solver, U-88
scale
    tensor member function, P-25
scalePoints utility, U-161
scaleSimilarity model, U-103
scientific
    keyword entry, U-114
scotch
    keyword entry, U-85, U-86
scotchCoeffs keyword, U-86
script/alias
    foamCorrectVrt, U-163
    foamJob, U-181
    foamLog, U-182
    make, U-73
    rmdepall, U-79
    wclean, U-78
    wmake, U-73
second time derivative, P-37
Seed window, U-173
selectCells utility, U-94
Set Solid Color button, U-171
setFields utility, U-61, U-92
setFormat keyword, U-179
sets keyword, U-179
setSet utility, U-94
setsToZones utility, U-94
Settings
    menu entry, U-172
settlingFoam solver, U-90
SFCD
    keyword entry, U-118, U-121
shallowWaterFoam solver, U-89
shape, U-144
Show Color Legend
    menu entry, U-27
SI units, U-110
simple
    keyword entry, U-85, U-86
simpleFilter model, U-102
simpleFoam solver, P-55, U-89
simpleGrading keyword, U-144
simpleSpline
    keyword entry, U-144
simulationType keyword, U-43, U-62, U-189
skew
    tensor member function, P-25
skewLinear
    keyword entry, U-118, U-121
slice class, P-31
slip
    boundary condition, U-141
Smagorinsky model, U-103
Smagorinsky2 model, U-103
smapToFoam utility, U-95

smoothDelta model, U-103
`smoother` keyword, U-125
`smoothSolver`
    keyword entry, U-123
`snap` keyword, U-150
`snapControls` keyword, U-150
snappyHexMesh utility
    background mesh, U-150
    cell removal, U-153
    cell splitting, U-151
    mesh layers, U-155
    meshing process, U-149
    snapping to surfaces, U-154
snappyHexMesh utility, U-92, U-148
snappyHexMeshDict file, U-149
`snGrad`
    fvc member function, P-37
`snGradCorrection`
    fvc member function, P-37
`snGradSchemes` keyword, U-115
`Solid Color`
    menu entry, U-171
solidDisplacementFoam solver, U-91
solidDisplacementFoam solver, U-52
solidEquilibriumDisplacementFoam solver, U-91
solidMixture
    library, U-102
solidParticle
    library, U-99
solids
    library, U-102
solver
    PDRFoam, U-90
    XiFoam, U-90
    blockMesh, P-47
    boundaryFoam, U-88
    bubbleFoam, U-89
    buoyantBoussinesqSimpleFoam, U-90
    buoyantBoussinesqPimpleFoam, U-90
    buoyantPimpleFoam, U-90
    buoyantSimpleFoam, U-90
    buoyantSimpleRadiationFoam, U-90
    cavitatingFoam, U-89
    channelFoam, U-88
    chtMultiRegionFoam, U-90
    chtMultiRegionSimpleFoam, U-90
    coalChemistryFoam, U-91
    coldEngineFoam, U-90
    compressibleInterFoam, U-89
    dieselEngineFoam, U-90
    dieselFoam, U-90
    dnsFoam, U-90
    dsmcFoam, U-91
    electrostaticFoam, U-91
    engineFoam, U-90
    financialFoam, U-91
    fireFoam, U-90
    icoFoam, U-19, U-23, U-24, U-27, U-88
    interFoam, U-89
    interMixingFoam, U-89
    interPhaseChangeFoam, U-90
    laplacianFoam, U-88
    mdEquilibrationFoam, U-91
    mdFoam, U-91
    mhdFoam, P-70, U-91
    multiphaseInterFoam, U-90
    nonNewtonianIcoFoam, U-88
    pimpleDyMFoam, U-88
    pimpleFoam, U-89
    pisoFoam, U-19, U-89
    porousExplicitSourceReactingParcelFoam, U-91
    porousSimpleFoam, U-89
    potentialFoam, P-46, U-88
    reactingFoam, U-90
    reactingParcelFoam, U-91
    rhoCentralFoam, U-89
    rhoPisoFoam, U-89
    rhoPimpleFoam, U-89
    rhoPorousSimpleFoam, U-89
    rhoReactingFoam, U-90
    rhoSimpleFoam, U-89
    rhoPorousMRFPimpleFoam, U-89
    scalarTransportFoam, U-88
    settlingFoam, U-90
    shallowWaterFoam, U-89
    simpleFoam, P-55, U-89
    solidDisplacementFoam, U-91
    solidDisplacementFoam, U-52
    solidEquilibriumDisplacementFoam, U-91
    sonicDyMFoam, U-89
    sonicFoam, P-61, U-89
    sonicLiquidFoam, P-64, U-89
    twoLiquidMixingFoam, U-90
    twoPhaseEulerFoam, U-90
    uncoupledKinematicParcelFoam, U-91
`solver` keyword, U-54, U-123
solver relative tolerance, U-124
solver tolerance, U-124
`solvers` keyword, U-123

sonicDyMFoam solver, U-89
sonicFoam solver, P-61, U-89
sonicLiquidFoam solver, P-64, U-89
source, P-37
SpalartAllmaras model, U-102, U-103
SpalartAllmarasDDES model, U-103
SpalartAllmarasIDDES model, U-103
specie
    library, U-101
specieThermo model, U-101, U-186
spectEddyVisc model, U-103
spline keyword, U-143
splitCells utility, U-94
splitMesh utility, U-94
splitMeshRegions utility, U-94
sqr
    tensor member function, P-25
sqrGradGrad
    fvc member function, P-37
star4ToFoam utility, U-93
startFace keyword, U-132
startFrom keyword, U-24, U-113
starToFoam utility, U-93, U-156
startTime
    keyword entry, U-24, U-113
startTime keyword, U-24, U-113
steady flow
    turbulent, P-54
steadyState
    keyword entry, U-121
Stereolithography (STL), U-149
stitchMesh utility, U-94
stl
    keyword entry, U-179
stopAt keyword, U-113
strategy keyword, U-85, U-86
streamFunction utility, U-95
stress analysis of plate with hole, U-47
stressComponents utility, U-95
Style window panel, U-26, U-171
Su
    fvm member function, P-37
subsetMesh utility, U-94
summation convention, P-17
SUPERBEE differencing, P-38
supersonic flow, P-60
supersonic flow over forward step, P-59
supersonicFreeStream
    boundary condition, U-141
surface mesh, U-149
surfaceField<Type> template class, P-33
surfaceFormat keyword, U-179
surfaceMesh tools, U-98
surfaceNormalFixedValue
    boundary condition, U-141
surfaces keyword, U-179
surfMesh
    library, U-99
SuSp
    fvm member function, P-37
sutherlandTransport model, U-101, U-186
symm
    tensor member function, P-25
symmetryPlane
    boundary condition, P-64, U-138
symmetryPlane
    keyword entry, U-138
symmTensorField class, P-29
symmTensorThirdField class, P-29
system directory, P-49, U-105
systemCall
    library, U-98

T

T()
    tensor member function, P-25
template class
    GeometricBoundaryField, P-32
    fvMatrix, P-36
    dimensioned<Type>, P-26
    FieldField<Type>, P-32
    Field<Type>, P-29
    geometricField<Type>, P-32
    List<Type>, P-29
    pointField<Type>, P-33
    surfaceField<Type>, P-33
    volField<Type>, P-33
temporal discretisation, P-42
    Crank Nicholson, P-43
    Euler implicit, P-42
    explicit, P-42
    in OpenFOAM, P-43
tensor, P-15
    addition, P-18
    algebraic operations, P-18
    algebraic operations in OpenFOAM, P-24
    antisymmetric, *see* tensor, skew
    calculus, P-27
    classes in OpenFOAM, P-23
    cofactors, P-22
    component average, P-20

component maximum, P-20
component minimum, P-20
determinant, P-22
deviatoric, P-22
diagonal, P-22
dimension, P-16
double inner product, P-19
geometric transformation, P-21
Hodge dual, P-23
hydrostatic, P-22
identities, P-21
identity, P-21
inner product, P-18
inverse, P-23
magnitude, P-20
magnitude squared, P-20
mathematics, P-15
notation, P-17
*n*th power, P-20
outer product, P-19
rank, P-16
rank 3, P-17
scalar division, P-18
scalar multiplication, P-18
scale function, P-20
second rank, P-16
skew, P-22
square of, P-20
subtraction, P-18
symmetric, P-22
symmetric rank 2, P-16
symmetric rank 3, P-17
trace, P-22
transformation, P-21
transpose, P-16, P-22
triple inner product, P-19
vector cross product, P-20
tensor class, P-24
tensor member function
`*`, P-25
`+`, P-25
`-`, P-25
`/`, P-25
`&`, P-25
`&&`, P-25
`^`, P-25
`cmptAv`, P-25
`cofactors`, P-25
`det`, P-25
`dev`, P-25
`diag`, P-25
`I`, P-25
`inv`, P-25
`mag`, P-25
`magSqr`, P-25
`max`, P-25
`min`, P-25
`pow`, P-25
`scale`, P-25
`skew`, P-25
`sqr`, P-25
`symm`, P-25
`T()`, P-25
`tr`, P-25
`transform`, P-25
tensorField class, P-29
tensorThirdField class, P-29
tetgenToFoam utility, U-93
text box
Opacity, U-171
thermalProperties
dictionary, U-52, U-53
thermophysical
library, U-185
thermophysicalFunctions
library, U-101
thermophysicalProperties
dictionary, U-185
`thermoType` keyword, U-185
time
control, U-112
time derivative, P-37
first, P-39
second, P-37, P-39
time step, U-24
`timeFormat` keyword, U-114
`timePrecision` keyword, U-114
`timeScheme` keyword, U-115
`timeStep`
keyword entry, U-24, U-33, U-113
tolerance
solver, U-124
solver relative, U-124
`tolerance` keyword, U-54, U-123, U-124, U-155
`Toolbars`
menu entry, U-171
tools
algorithms, U-98
cfdTools, U-98
containers, U-98

db, U-98
dimensionSet, U-98
dimensionedTypes, U-98
fields, U-98
finiteVolume, U-98
fvMatrices, U-98
fvMesh, U-98
global, U-98
graph, U-98
interpolations, U-98
interpolation, U-98
matrices, U-98
memory, U-98
meshes, U-98
primitives, U-98
surfaceMesh, U-98
, U-98
topoChangerFvMesh
library, U-99
`topoSetSource` keyword, U-61
totalPressure
boundary condition, U-141
`tr`
tensor member function, P-25
trace, *see* tensor, trace
`traction` keyword, U-52
`transform`
tensor member function, P-25
transformPoints utility, U-94
transportProperties
dictionary, U-23, U-40, U-43
transportProperties file, U-61
triple inner product, P-19
triSurface
library, U-99
turbulence
dissipation, U-42
kinetic energy, U-42
length scale, U-42
`turbulence` keyword, U-189
turbulence model
RAS, U-41
turbulenceProperties
dictionary, U-43, U-62, U-189
turbulent flow
steady, P-54
turbulentInlet
boundary condition, U-141
tutorials
breaking of a dam, U-58
lid-driven cavity flow, U-19
stress analysis of plate with hole, U-47
tutorials directory, P-45, U-19
twoLiquidMixingFoam solver, U-90
twoPhaseEulerFoam solver, U-90
`type` keyword, U-133, U-136

U

`U` field, U-24
Ucomponents utility, P-71
`UMIST`
keyword entry, U-117
`uncompressed`
keyword entry, U-114
`uncorrected`
keyword entry, U-119, U-120
uncoupledKinematicParcelFoam solver, U-91
uniform model, U-101
`uniform` keyword, U-180
units
base, U-110
of measurement, P-26, U-109
S.I. base, P-26
SI, U-110
Système International, U-110
United States Customary System, U-110
USCS, U-110
Update GUI button, U-27, U-169
uprime utility, U-95
`upwind`
keyword entry, U-118, U-121
upwind differencing, P-38, U-64
USCS units, U-110
Use Parallel Projection button, U-26
Use parallel projection button, U-171
utility
Co, U-95
IFCLookUpTableGen, U-97
Lambda2, U-95
Mach, U-95
PV3FoamReader, U-94
PVFoamReader, U-94
Pe, U-95
Q, U-95
R, U-96
Ucomponents, P-71
adiabaticFlameT, U-97
ansysToFoam, U-92
applyBoundaryLayer, U-92
applyWallFunctionBoundaryConditions, U-92
attachMesh, U-93

autoPatch, U-93
autoRefineMesh, U-94
blockMesh, U-39, U-92, U-139
boxTurb, U-92
cfx4ToFoam, U-92, U-156
changeDictionary, U-92
checkMesh, U-93, U-159
chemkinToFoam, U-97
collapseEdges, U-94
combinePatchFaces, U-94
createBaffles, U-93
createPatch, U-93
createTurbulenceFields, U-96
decomposePar, U-84, U-85, U-96
deformedGeom, U-93
dsmcFieldsCalc, U-96
dsmcInitialise, U-92
engineCompRatio, U-96
engineSwirl, U-92
ensight74FoamExec, U-176
ensightFoamReader, U-94
enstrophy, U-95
equilibriumCO, U-97
equilibriumFlameT, U-97
estimateScalarError, U-97
execFlowFunctionObjects, U-96
expandDictionary, U-97
extrude2DMesh, U-92
extrudeMesh, U-92
fieldview9Reader, U-94
flattenMesh, U-93
flowType, U-95
fluent3DMeshToFoam, U-92
fluentMeshToFoam, U-92, U-156
foamCalc, U-34
foamDataToFluent, U-94, U-174
foamDebugSwitches, U-97
foamFormatConvert, U-97
foamInfoExec, U-97
foamMeshToFluent, U-92, U-174
foamToEnsightParts, U-94
foamToEnsight, U-94
foamToFieldview9, U-95
foamToGMV, U-95
foamToStarMesh, U-92
foamToTecplot360, U-95
foamToVTK, U-95
foamUpgradeFvSolution, U-92
gambitToFoam, U-93, U-156
gmshToFoam, U-93
icoErrorEstimate, U-97
icoMomentError, U-97
ideasToFoam, U-156
ideasUnvToFoam, U-93
insideCells, U-93
kivaToFoam, U-93
mapFields, U-33, U-40, U-44, U-57, U-92, U-164
mdInitialise, U-92
mergeMeshes, U-93
mergeOrSplitBaffles, U-93
mirrorMesh, U-93
mixtureAdiabaticFlameT, U-97
modifyMesh, U-94
momentScalarError, U-97
moveDynamicMesh, U-93
moveEngineMesh, U-93
moveMesh, U-93
mshToFoam, U-93
netgenNeutralToFoam, U-93
objToVTK, U-93
pPrime2, U-95
particleTracks, U-96
patchAverage, U-96
patchIntegrate, U-96
patchSummary, U-97
pdfPlot, U-96
plot3dToFoam, U-93
polyDualMesh, U-93
postChannel, U-96
probeLocations, U-96
ptot, U-96
reconstructParMesh, U-96
reconstructPar, U-88, U-96
redistributeMeshPar, U-96
refineHexMesh, U-94
refineMesh, U-93
refineWallLayer, U-94
refinementLevel, U-94
removeFaces, U-94
renumberMesh, U-94
rotateMesh, U-94
sammToFoam, U-93
sample, U-96, U-178
scalePoints, U-161
selectCells, U-94
setFields, U-61, U-92
setSet, U-94
setsToZones, U-94
smapToFoam, U-95

snappyHexMesh, U-92, U-148
splitCells, U-94
splitMeshRegions, U-94
splitMesh, U-94
star4ToFoam, U-93
starToFoam, U-93, U-156
stitchMesh, U-94
streamFunction, U-95
stressComponents, U-95
subsetMesh, U-94
tetgenToFoam, U-93
transformPoints, U-94
uprime, U-95
vorticity, U-95
wallGradU, U-95
wallHeatFlux, U-95
wallShearStress, U-95
wdot, U-96
writeCellCentres, U-96
writeMeshObj, U-93
yPlusLES, U-95
yPlusRAS, U-95
zipUpMesh, U-94
cellSet, U-93
faceSet, U-93
pointSet, U-93
utilityFunctionObjects
library, U-98

V

value keyword, U-23, U-140
valueFraction keyword, U-140
van Leer differencing, P-38
vanLeer
keyword entry, U-118
VCR Controls menu, U-27, U-169
vector, P-16
operator, P-27
unit, P-20
vector class, P-24, U-109
vector product, *see* tensor, vector cross product
vectorField class, P-29
version keyword, U-107
vertices keyword, U-22, U-143
veryInhomogeneousMixture model, U-100, U-186
View menu, U-171
View Settings
menu entry, U-26, U-171
View Settings...
menu entry, U-26
viscosity
kinematic, U-23, U-43
volField<Type> template class, P-33
vorticity utility, U-95
vtk
keyword entry, U-179
vtkFoam
library, U-167
vtkPV3Foam
library, U-167

W

wall
boundary condition, P-64, P-70, U-60, U-137
wall
keyword entry, U-138
wallBuoyantPressure
boundary condition, U-141
wallGradU utility, U-95
wallHeatFlux utility, U-95
Wallis
library, U-101
wallShearStress utility, U-95
wclean script/alias, U-78
wdot utility, U-96
wedge
boundary condition, U-133, U-138, U-148
wedge
keyword entry, U-138
window
Chart Options, U-36
Color Legend, U-29
Options, U-172
Pipeline Browser, U-25, U-168
Render View Options, U-171
Render View, U-172
Seed, U-173
window panel
Annotation, U-26, U-172
Color Legend, U-170
Color Scale, U-170
Display, U-26, U-27, U-168, U-169
General, U-171, U-172
Information, U-168
Lights, U-171
Properties, U-27, U-168, U-169
Region Status, U-26
Render View, U-172
Style, U-26, U-171
Wireframe
menu entry, U-171
WM_ARCH

environment variable, U-78
WM_ARCH_OPTION
environment variable, U-78
WM_COMPILE_OPTION
environment variable, U-78
WM_COMPILER
environment variable, U-78
WM_COMPILER_BIN
environment variable, U-78
WM_COMPILER_DIR
environment variable, U-78
WM_COMPILER_LIB
environment variable, U-78
WM_DIR
environment variable, U-78
WM_JAVAC_OPTION
environment variable, U-78
WM_LINK_LANGUAGE
environment variable, U-78
WM_MPLIB
environment variable, U-78
WM_OPTIONS
environment variable, U-78
WM_PRECISION_OPTION
environment variable, U-78
WM_PROJECT
environment variable, U-78
WM_PROJECT_DIR
environment variable, U-78
WM_PROJECT_INST_DIR
environment variable, U-78
WM_PROJECT_LANGUAGE
environment variable, U-78
WM_PROJECT_USER_DIR
environment variable, U-78
WM_PROJECT_VERSION
environment variable, U-78
wmake
platforms, U-75
wmake script/alias, U-73
word class, P-26, P-31
writeCellCentres utility, U-96
writeCompression keyword, U-114
writeControl
keyword entry, U-113
writeControl keyword, U-24, U-63, U-113
writeFormat keyword, U-57, U-113
writeInterval keyword, U-24, U-33, U-113
writeMeshObj utility, U-93
writeNow
keyword entry, U-113
writePrecision keyword, U-113

X

x
keyword entry, U-180
XiFoam solver, U-90
xmgr
keyword entry, U-114, U-179
xyz
keyword entry, U-180

Y

y
keyword entry, U-180
yPlusLES utility, U-95
yPlusRAS utility, U-95

Z

z
keyword entry, U-180
zeroGradient
boundary condition, U-140
zipUpMesh utility, U-94