OpenFOAM

The Open Source CFD Toolbox

Programmer's Guide

Version 2.0.0
16th June 2011

Copyright © 2004-2011 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-50
3.2 Steady turbulent flow over a backward-facing step . . . P-53
3.2.1 Problem specification . . . P-53
3.2.2 Mesh generation . . . P-54
3.2.3 Boundary conditions and initial fields . . . P-57
3.2.4 Case control . . . P-58
3.2.5 Running the case and post-processing . . . P-58
3.3 Supersonic flow over a forward-facing step . . . P-59
3.3.1 Problem specification . . . P-59
3.3.2 Mesh generation . . . P-60
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-62
3.4.1 Problem specification . . . P-63

3.4.2 Mesh Generation P-64
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-68
3.5.1 Problem specification P-68
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.

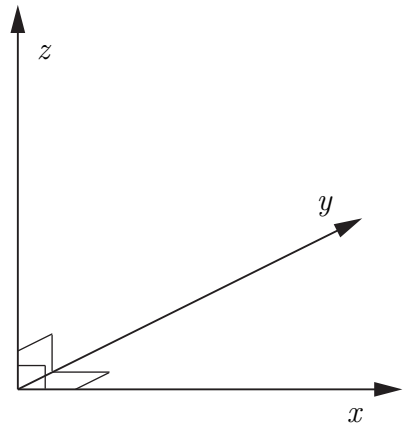


Figure 1.1: Right handed axes

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 $\mathbf{b} = \mathbf{T} \bullet \mathbf{a}$. 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

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

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

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

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

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

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 $\mathbf{a}$.

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

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.

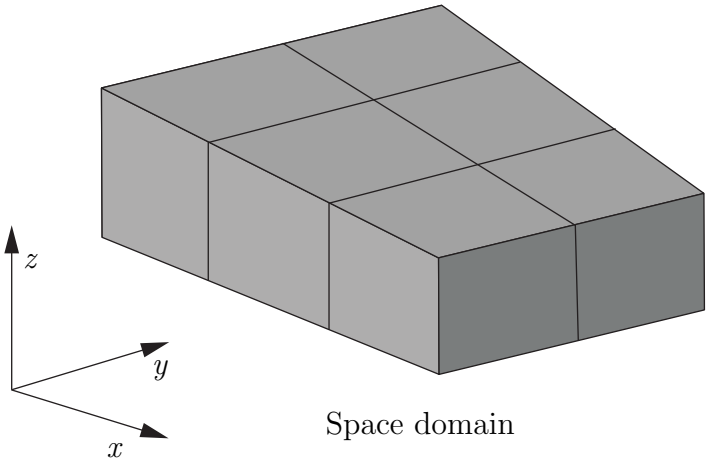


Figure 2.1: Discretisation of the solution domain

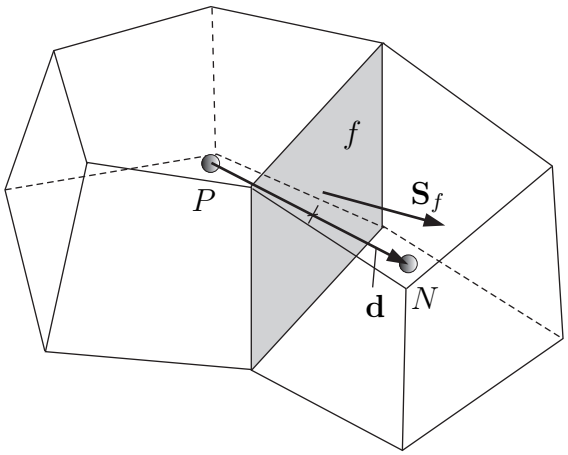


Figure 2.2: Parameters in finite volume discretisation

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.*.

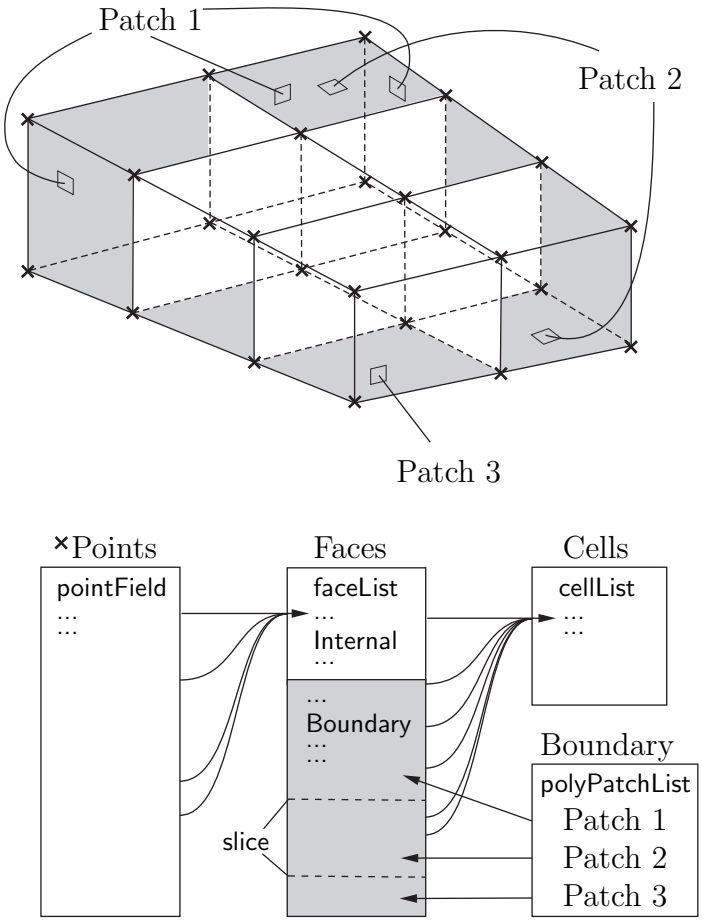


Figure 2.3: Schematic of the basic mesh description used in OpenFOAM

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].

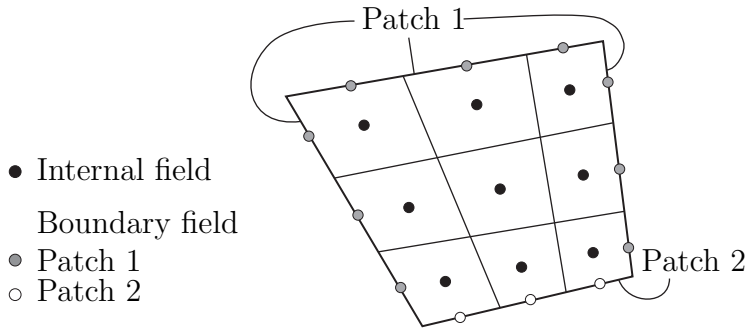
2.4 Equation discretisation

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

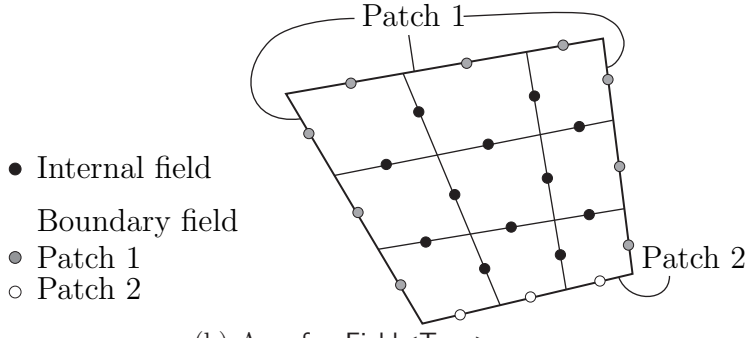
$$[A]\,[x] = [b] \tag{2.12}$$

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.

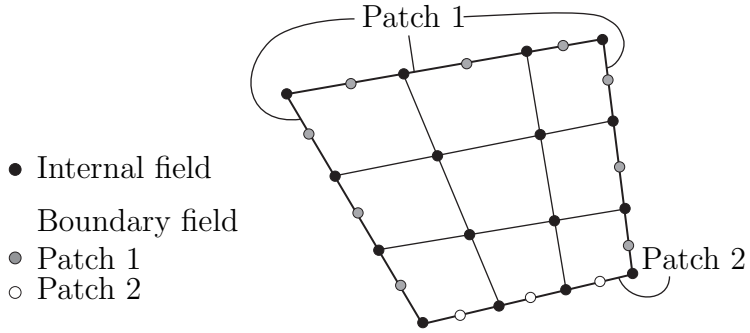
[1]The diagram is not an exact description of the class hierarchy, rather a representation of the general structure leading from some primitive classes to geometric<Type>Field.



(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)

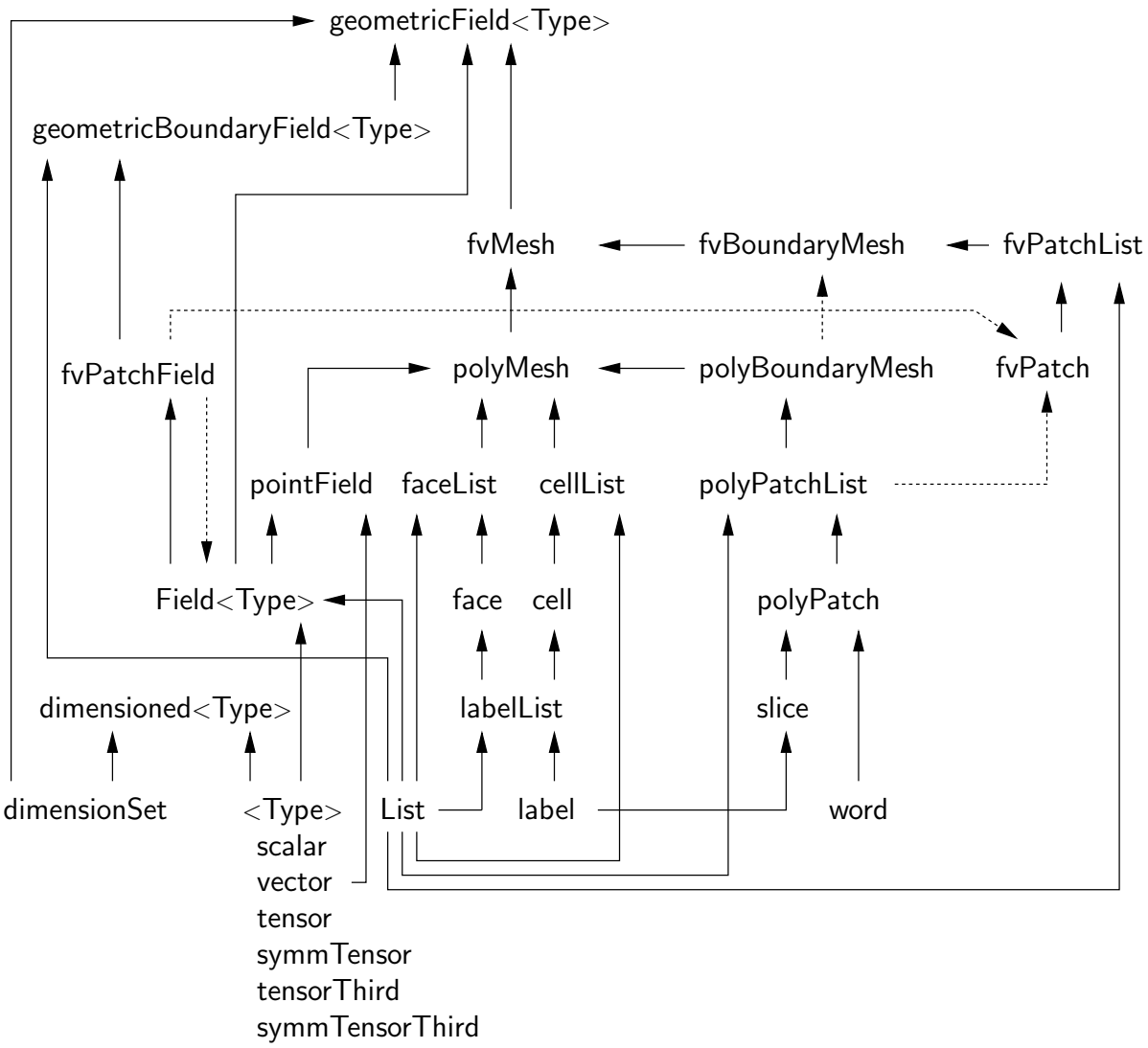


Figure 2.5: Basic class structure leading to geometricField<Type>

$[A]$ 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.

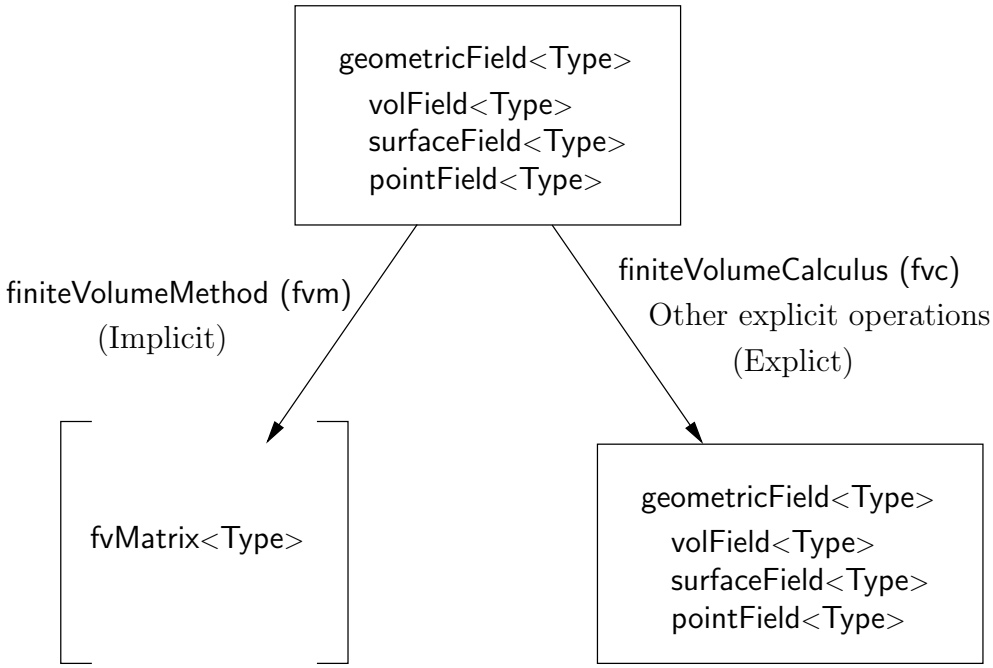


Figure 2.6: A geometricField<Type> and its operators

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.

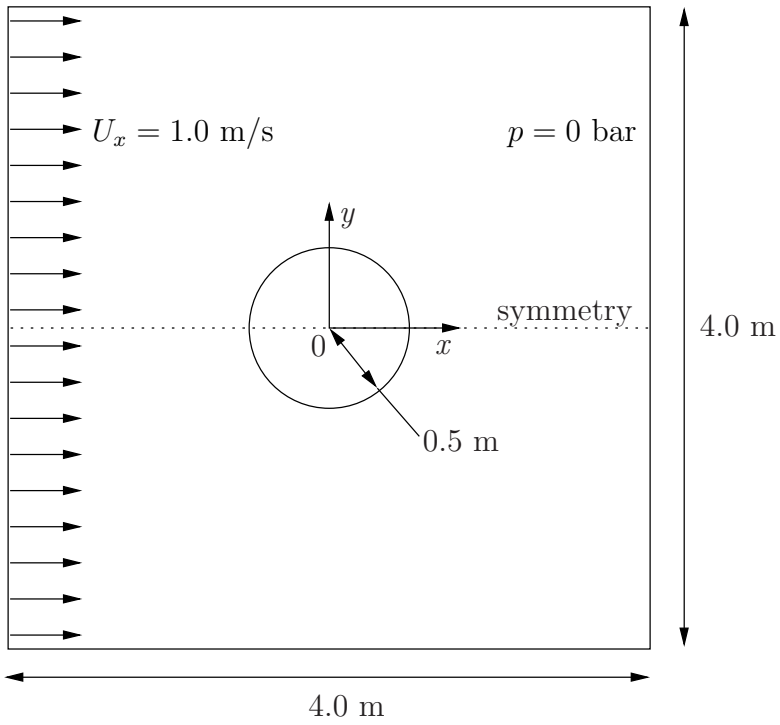


Figure 3.1: Geometry of flow round a cylinder

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

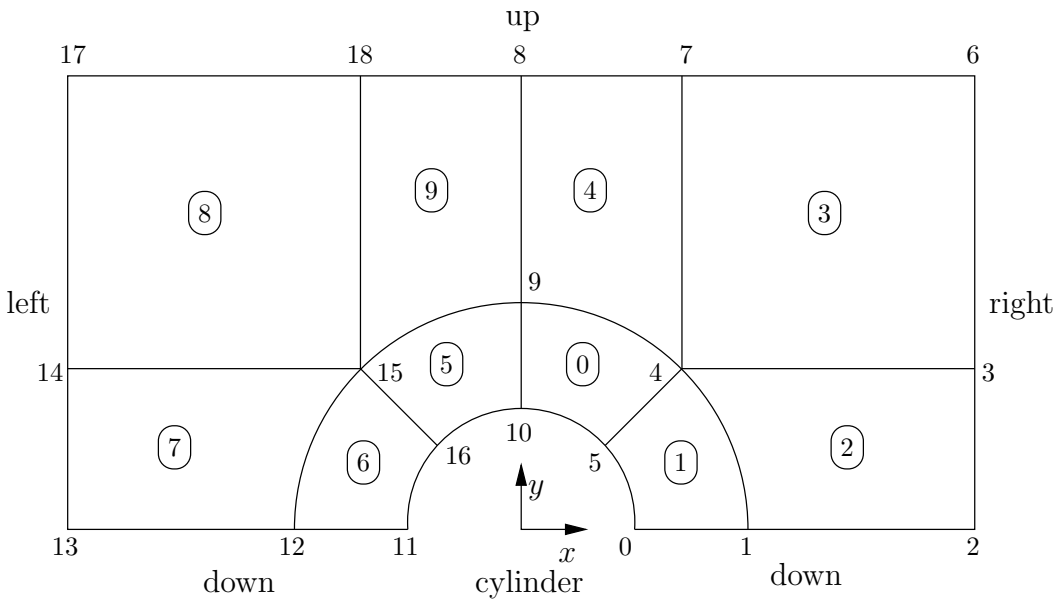


Figure 3.2: Blocks in cylinder geometry

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:

```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  2.0.0                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
```

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

convertToMeters 1;

vertices #codeStream
{
    codeInclude
    #{
        #include "pointField.H"
    #};

    code
    #{
        pointField points(19);
        points[0]  = point(0.5, 0, -0.5);
        points[1]  = point(1, 0, -0.5);
        points[2]  = point(2, 0, -0.5);
        points[3]  = point(2, 0.707107, -0.5);
        points[4]  = point(0.707107, 0.707107, -0.5);
        points[5]  = point(0.353553, 0.353553, -0.5);
        points[6]  = point(2, 2, -0.5);
        points[7]  = point(0.707107, 2, -0.5);
        points[8]  = point(0, 2, -0.5);
        points[9]  = point(0, 1, -0.5);
        points[10] = point(0, 0.5, -0.5);
        points[11] = point(-0.5, 0, -0.5);
        points[12] = point(-1, 0, -0.5);
        points[13] = point(-2, 0, -0.5);
        points[14] = point(-2, 0.707107, -0.5);
        points[15] = point(-0.707107, 0.707107, -0.5);
        points[16] = point(-0.353553, 0.353553, -0.5);
        points[17] = point(-2, 2, -0.5);
        points[18] = point(-0.707107, 2, -0.5);

        // Duplicate z points
        label sz = points.size();
        points.setSize(2*sz);
        for (label i = 0; i < sz; i++)
        {
            const point& pt = points[i];
            points[i+sz] = point(pt.x(), pt.y(), -pt.z());
        }

        os  << points;
    #};
};


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)
    arc 34 28 (-0.34202 0.939693 0.5)
);

boundary
(
    down
    {
        type symmetryPlane;
        faces
        (
            (0 1 20 19)
            (1 2 21 20)
            (12 11 30 31)
            (13 12 31 32)
        );
    }
    right
    {
        type patch;
        faces
        (
            (2 3 22 21)
            (3 6 25 22)
        );
    }
    up
    {
        type symmetryPlane;
        faces
        (
            (7 8 27 26)
            (6 7 26 25)
            (8 18 37 27)
            (18 17 36 37)
        );
    }
    left
    {
        type patch;
        faces
        (
            (14 13 32 33)
            (17 14 33 36)
        );
    }
    cylinder
    {
        type symmetryPlane;
        faces
        (
            (10 5 24 29)
            (5 0 19 24)
            (16 10 29 35)
            (11 16 35 30)
        );
    }
);

mergePatchPairs
(
);

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

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.

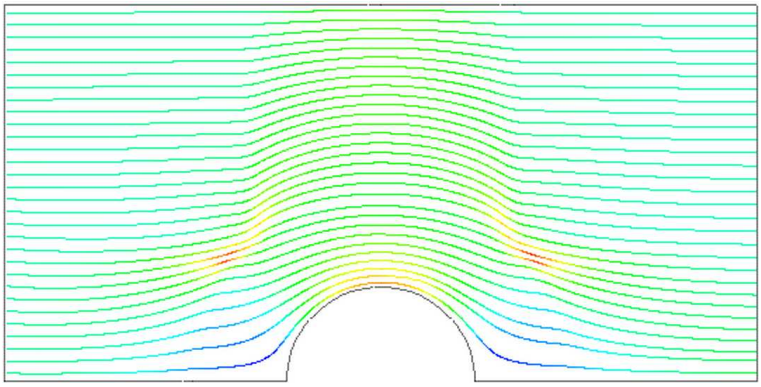
3.1.5 Running the case

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:

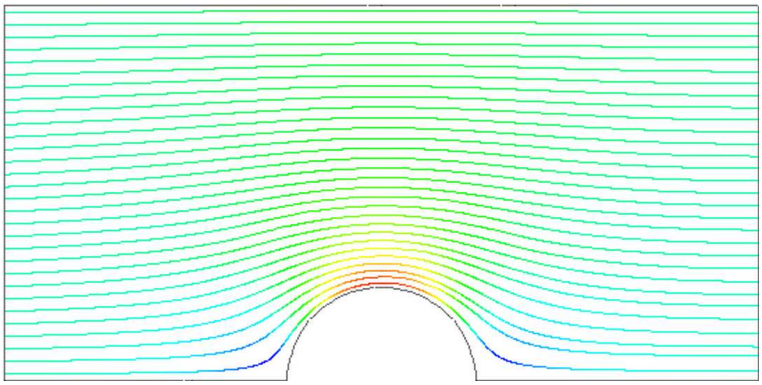
```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     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 off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;

functions
{
    difference
    {
        // Load the library containing the 'coded' functionObject
        functionObjectLibs ("libutilityFunctionObjects.so");
        type coded;
        // Name of on-the-fly generated functionObject
        redirectType error;
        code
```

```
58          #{
59              // Lookup U
60              Info<< "Looking up field U\n" << endl;
61              const volVectorField& U = mesh().lookupObject<volVectorField>("U");
62
63              Info<< "Reading inlet velocity  uInfX\n" << endl;
64
65              dimensionedScalar uInfX
66              (
67                  "uInfx",
68                  dimensionSet(0, 1, -1, 0, 0),
69                  U.boundaryField()[3][0].x()
70              );
71              Info << "U at inlet = " << uInfX.value() << " m/s" << endl;
72
73              dimensionedScalar radius
74              (
75                  "radius",
76                  dimensionSet(0, 1, 0, 0, 0),
77                  mag(U.mesh().boundary()[4].Cf()[0])
78              );
79
80              Info << "Cylinder radius = " << radius.value() << " m" << endl;
81
82              volVectorField UA
83              (
84                  IOobject
85                  (
86                      "UA",
87                      mesh().time().timeName(),
88                      U.mesh(),
89                      IOobject::NO_READ,
90                      IOobject::AUTO_WRITE
91                  ),
92                  U
93              );
94
95              Info<< "\nEvaluating analytical solution" << endl;
96
97              const volVectorField& centres = UA.mesh().C();
98              volScalarField magCentres(mag(centres));
99              volScalarField theta(acos((centres & vector(1,0,0))/magCentres));
100
101             volVectorField cs2theta
102             (
103                 cos(2*theta)*vector(1,0,0)
104               + sin(2*theta)*vector(0,1,0)
105             );
106
107             UA = uInfX*(dimensionedVector(vector(1,0,0))
108               - pow((radius/magCentres),2)*cs2theta);
109
110             // Force writing of UA (since time has not changed)
111             UA.write();
112
113             volScalarField error("error", mag(U-UA)/mag(UA));
114
115             Info<<"Writing relative error in U to " << error.objectPath()
116                 << endl;
117
118             error.write();
119         #};
120     }
121 }
122
123
124 // ************************************************************************* //
```

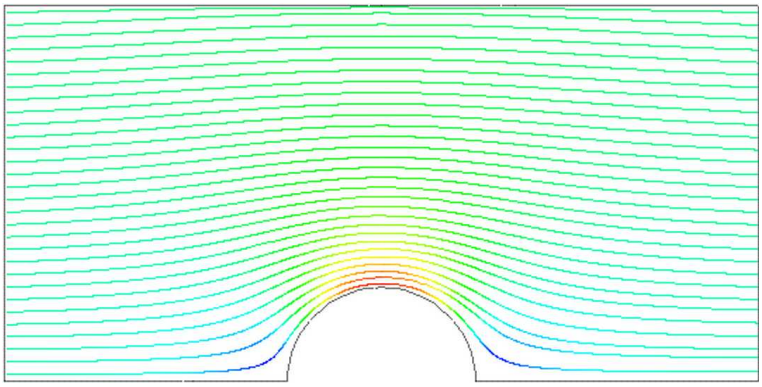
potentialFoam executes an iterative loop around the pressure equation which it solves in order that explicit terms relating to non-orthogonal correction in the Laplacian term may be updated in successive iterations. The number of iterations around the pressure equation is controlled by the `nNonOrthogonalCorrectors` 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



(a) With no non-orthogonal correction



(b) With non-orthogonal correction



(c) Analytical solution

Figure 3.3: Streamlines of potential flow

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).

3.2 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.

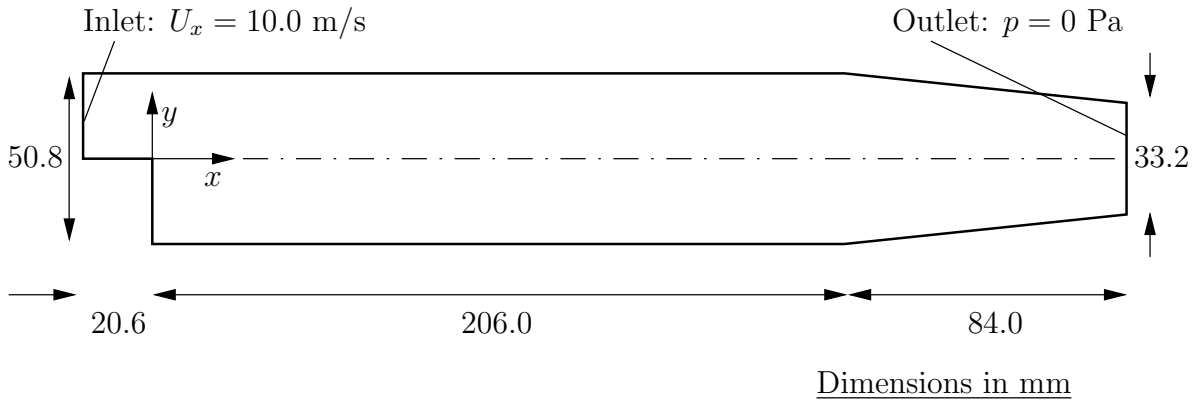


Figure 3.4: Geometry of backward-facing step

Governing equations

- Mass continuity for incompressible flow

$$\nabla \bullet \mathbf{U} = 0 \tag{3.3}$$

- Steady flow momentum equation

$$\nabla \bullet (\mathbf{UU}) + \nabla \bullet \mathbf{R} = -\nabla p \tag{3.4}$$

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:  2.0.0                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
13     object      blockMeshDict;
```

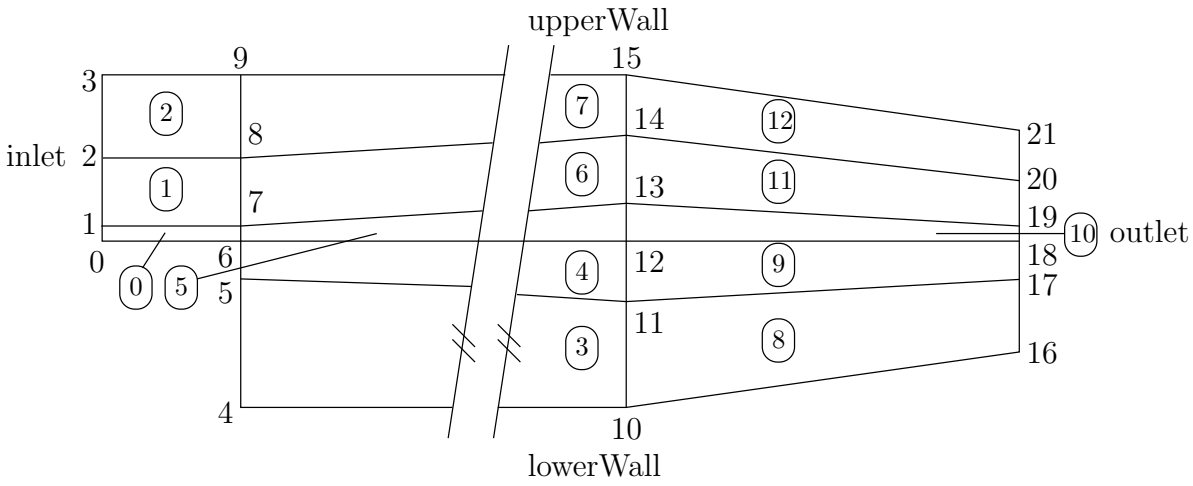


Figure 3.5: Blocks in backward-facing step

```
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  (
```

```
69      hex (0 6 7 1 22 28 29 23) (18 7 1) simpleGrading (0.5 1.8 1)
70      hex (1 7 8 2 23 29 30 24) (18 10 1) simpleGrading (0.5 4 1)
71      hex (2 8 9 3 24 30 31 25) (18 13 1) simpleGrading (0.5 0.25 1)
72      hex (4 10 11 5 26 32 33 27) (180 18 1) simpleGrading (4 1 1)
73      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)
74      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)
75      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)
76      hex (8 14 15 9 30 36 37 31) (180 13 1) simpleGrading (4 0.25 1)
77      hex (10 16 17 11 32 38 39 33) (25 18 1) simpleGrading (2.5 1 1)
78      hex (11 17 18 12 33 39 40 34) (25 9 1) simpleGrading (2.5 1 1)
79      hex (12 18 19 13 34 40 41 35) (25 7 1) simpleGrading (2.5 1 1)
80      hex (13 19 20 14 35 41 42 36) (25 10 1) simpleGrading (2.5 1 1)
81      hex (14 20 21 15 36 42 43 37) (25 13 1) simpleGrading (2.5 0.25 1)
82  );
83
84  edges
85  (
86  );
87
88  boundary
89  (
90      inlet
91      {
92          type patch;
93          faces
94          (
95              (0 22 23 1)
96              (1 23 24 2)
97              (2 24 25 3)
98          );
99      }
100     outlet
101     {
102         type patch;
103         faces
104         (
105             (16 17 39 38)
106             (17 18 40 39)
107             (18 19 41 40)
108             (19 20 42 41)
109             (20 21 43 42)
110         );
111     }
112     upperWall
113     {
114         type wall;
115         faces
116         (
117             (3 25 31 9)
118             (9 31 37 15)
119             (15 37 43 21)
120         );
121     }
122     lowerWall
123     {
124         type wall;
125         faces
126         (
127             (0 6 28 22)
128             (6 5 27 28)
129             (5 4 26 27)
130             (4 10 32 26)
131             (10 16 38 32)
132         );
133     }
134     frontAndBack
135     {
136         type empty;
137         faces
138         (
139             (22 28 29 23)
140             (23 29 30 24)
141             (24 30 31 25)
142             (26 32 33 27)
143             (27 33 34 28)
144             (28 34 35 29)
145             (29 35 36 30)
146             (30 36 37 31)
```

```
147                 (32 38 39 33)
148                 (33 39 40 34)
149                 (34 40 41 35)
150                 (35 41 42 36)
151                 (36 42 43 37)
152                 (0 1 7 6)
153                 (1 2 8 7)
154                 (2 3 9 8)
155                 (4 5 11 10)
156                 (5 6 12 11)
157                 (6 7 13 12)
158                 (7 8 14 13)
159                 (8 9 15 14)
160                 (10 11 17 16)
161                 (11 12 18 17)
162                 (12 13 19 18)
163                 (13 14 20 19)
164                 (14 15 21 20)
165             );
166         }
167     );
168
169     mergePatchPairs
170     (
171     );
172
173     // ************************************************************************* //
```

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

$$U'_x = U'_y = U'_z = \frac{5}{100}10 = 0.5 \text{ m/s} \tag{3.5}$$

and

$$k = \frac{3}{2}(0.5)^2 = 0.375 \text{ m}^2/\text{s}^2 \tag{3.6}$$

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

$$\varepsilon = \frac{C_\mu^{0.75} k^{1.5}}{l} = \frac{0.09^{0.75} 0.375^{1.5}}{0.1 \times 25.4 \times 10^{-3}} = 14.855\,\mathrm{m}^2/\mathrm{s}^3 \tag{3.7}$$

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

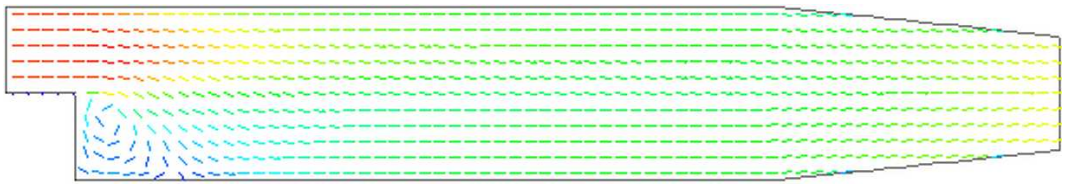
3.2.4 Case control

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.

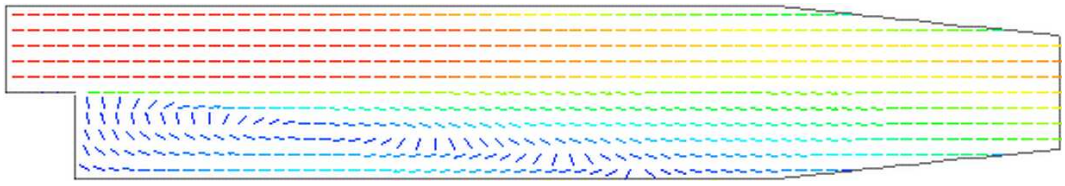
Special attention should be paid to the settings of *fvTolerances*. Although the top level simpleFoam code contains only equations for p and $\mathbf{U}$, the turbulent model solves equations for k, ε and $\mathbf{R}$, and tolerance settings are required for all 5 equations. A `solverTolerance` of 10^{-5} 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.

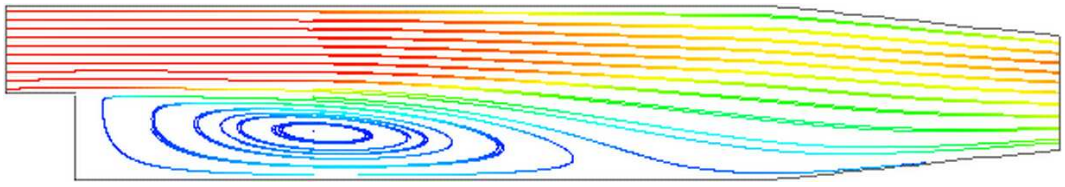
3.2.5 Running the case and post-processing



(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

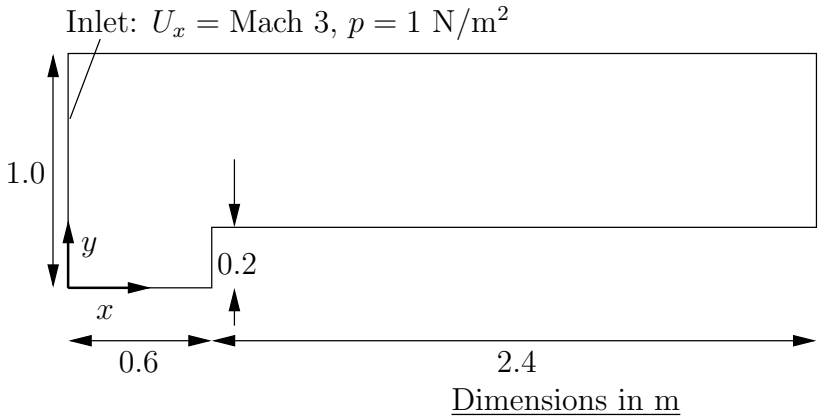


Figure 3.7: Geometry of the forward step geometry

Governing equations

- Mass continuity

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathbf{U}) = 0 \tag{3.8}$$

- Ideal gas

$$p = \rho R T \tag{3.9}$$

- Momentum equation for Newtonian fluid

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \bullet (\rho \mathbf{U}\mathbf{U}) - \nabla \bullet \mu \nabla \mathbf{U} = -\nabla p \tag{3.10}$$

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

$$\frac{\partial \rho e}{\partial t} + \nabla \bullet (\rho \mathbf{U} e) - \nabla \bullet \left(\frac{k}{C_v}\right) \nabla e = p \nabla \bullet \mathbf{U} \tag{3.11}$$

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 $\mu = 18.1 \mu \text{Pa s}$

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

$$\gamma = C_p/C_v = \frac{R}{C_v} + 1 \tag{3.12}$$

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:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices
(
    (0 0 -0.05)
    (0.6 0 -0.05)
    (0 0.2 -0.05)
    (0.6 0.2 -0.05)
    (3 0.2 -0.05)
    (0 1 -0.05)
    (0.6 1 -0.05)
    (3 1 -0.05)
    (0 0 0.05)
    (0.6 0 0.05)
    (0 0.2 0.05)
    (0.6 0.2 0.05)
    (3 0.2 0.05)
    (0 1 0.05)
    (0.6 1 0.05)
    (3 1 0.05)
);

blocks
(
    hex (0 1 3 2 8 9 11 10) (25 10 1) simpleGrading (1 1 1)
    hex (2 3 6 5 10 11 14 13) (25 40 1) simpleGrading (1 1 1)
    hex (3 4 7 6 11 12 15 14) (100 40 1) simpleGrading (1 1 1)
);

edges
(
);

boundary
(
    inlet
    {
        type patch;
        faces
        (
            (0 8 10 2)
            (2 10 13 5)
        );
    }
    outlet
    {
        type patch;
        faces
        (
            (4 7 15 12)
        );
    }
    bottom
    {
        type symmetryPlane;
        faces
        (
            (0 1 9 8)
        );
    }
    top
    {
        type symmetryPlane;
```

```
80          faces
81          (
82              (5 13 14 6)
83              (6 14 15 7)
84          );
85      }
86      obstacle
87      {
88          type patch;
89          faces
90          (
91              (1 3 11 9)
92              (3 4 12 11)
93          );
94      }
95  );
96
97  mergePatchPairs
98  (
99  );
100
101 // ************************************************************************* //
```

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.

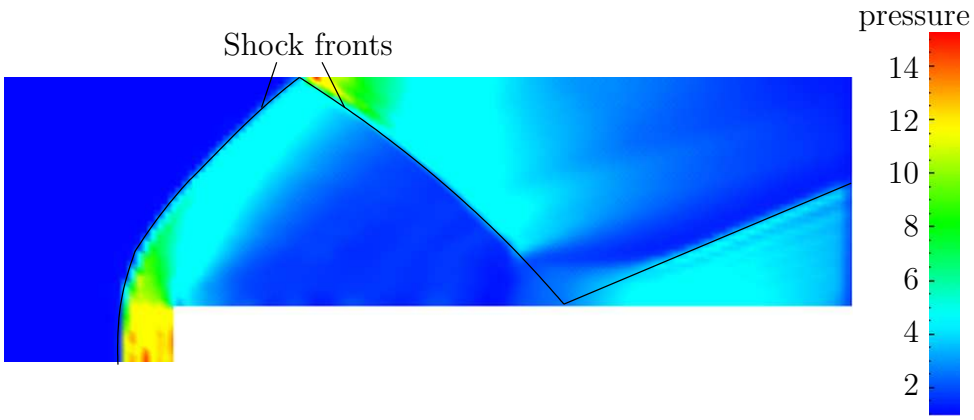


Figure 3.8: Shock fronts in the forward step problem

3.3.4 Exercise

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

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

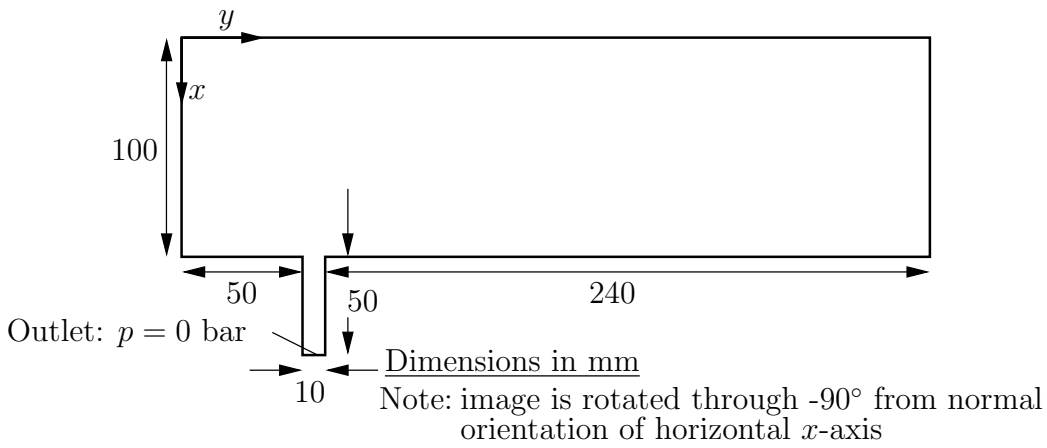


Figure 3.9: Geometry of a tank with outflow pipe

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

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathbf{U}) = 0 \tag{3.13}$$

- The barotropic relationship

$$\frac{\partial \rho}{\partial p} = \frac{\rho}{K} = \psi \tag{3.14}$$

  where K is the bulk modulus

- Equation 3.14 is linearised as

$$\rho \approx \rho_0 + \psi \left(p - p_0 \right) \tag{3.15}$$

  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

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \bullet \left(\rho \mathbf{U} \mathbf{U} \right) - \nabla \bullet \mu \nabla \mathbf{U} = -\nabla p \tag{3.16}$$

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:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     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)
```

```
37      (1 0.6 0.1)
38      (1.5 0.6 0.1)
39      (0 3 0.1)
40      (1 3 0.1)
41  );
42
43  blocks
44  (
45      hex (0 1 3 2 10 11 13 12) (30 20 1) simpleGrading (1 1 1)
46      hex (2 3 6 5 12 13 16 15) (30 5 1) simpleGrading (1 1 1)
47      hex (3 4 7 6 13 14 17 16) (25 5 1) simpleGrading (1 1 1)
48      hex (5 6 9 8 15 16 19 18) (30 95 1) simpleGrading (1 1 1)
49  );
50
51  edges
52  (
53  );
54
55  boundary
56  (
57      outerWall
58      {
59          type wall;
60          faces
61          (
62              (0 1 11 10)
63              (1 3 13 11)
64              (3 4 14 13)
65              (7 6 16 17)
66              (6 9 19 16)
67              (9 8 18 19)
68          );
69      }
70      axis
71      {
72          type symmetryPlane;
73          faces
74          (
75              (0 10 12 2)
76              (2 12 15 5)
77              (5 15 18 8)
78          );
79      }
80      nozzle
81      {
82          type patch;
83          faces
84          (
85              (4 7 17 14)
86          );
87      }
88      back
89      {
90          type empty;
91          faces
92          (
93              (0 2 3 1)
94              (2 5 6 3)
95              (3 6 7 4)
96              (5 8 9 6)
97          );
98      }
99      front
100     {
101         type empty;
102         faces
103         (
104             (10 11 13 12)
105             (12 13 16 15)
106             (13 14 17 16)
107             (15 16 19 18)
108         );
109     }
110 );
111
112 mergePatchPairs
113 (
114 );
115
116 // ************************************************************************* //
```

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:

$$c = \sqrt{\frac{1}{\psi}} = \sqrt{\frac{1}{4.54 \times 10^{-7}}} = 1483.2\text{m/s}. \tag{3.17}$$

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

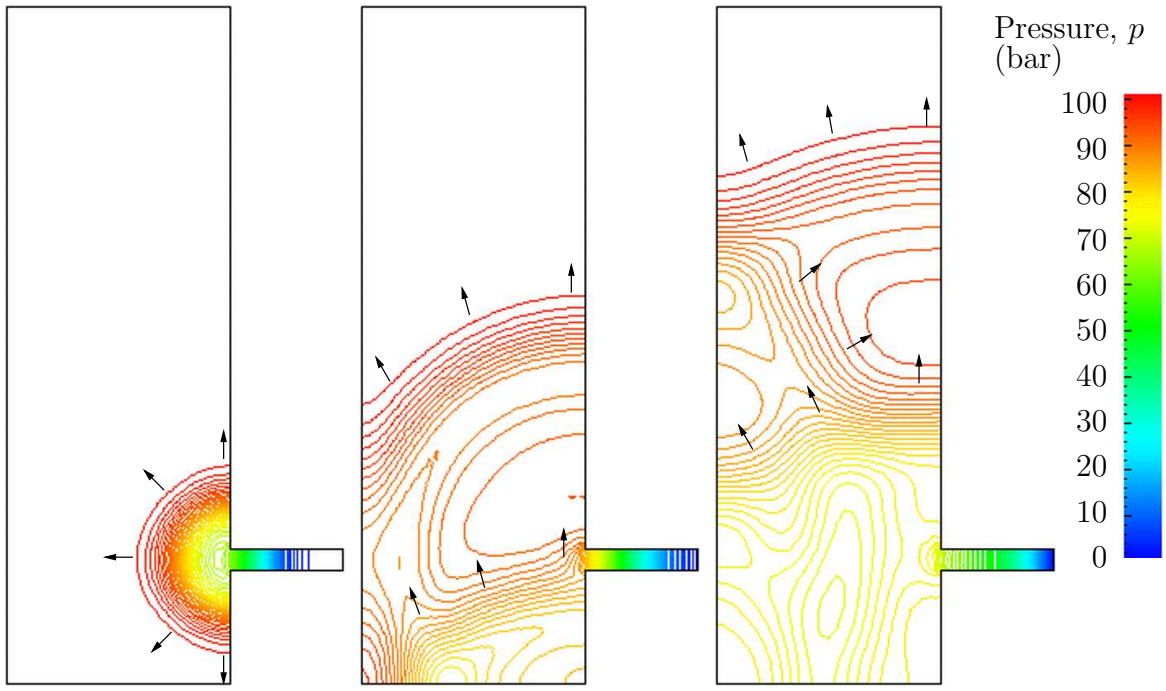
$$Co = \frac{U\,\Delta t}{\Delta x} \tag{3.18}$$

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.

```
/*---------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    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;

```

```
40  writeCompression off;
41
42  timeFormat      general;
43
44  timePrecision   6;
45
46  runTimeModifiable true;
47
48
49  // ************************************************************************* //
```

3.4.4 Running the case

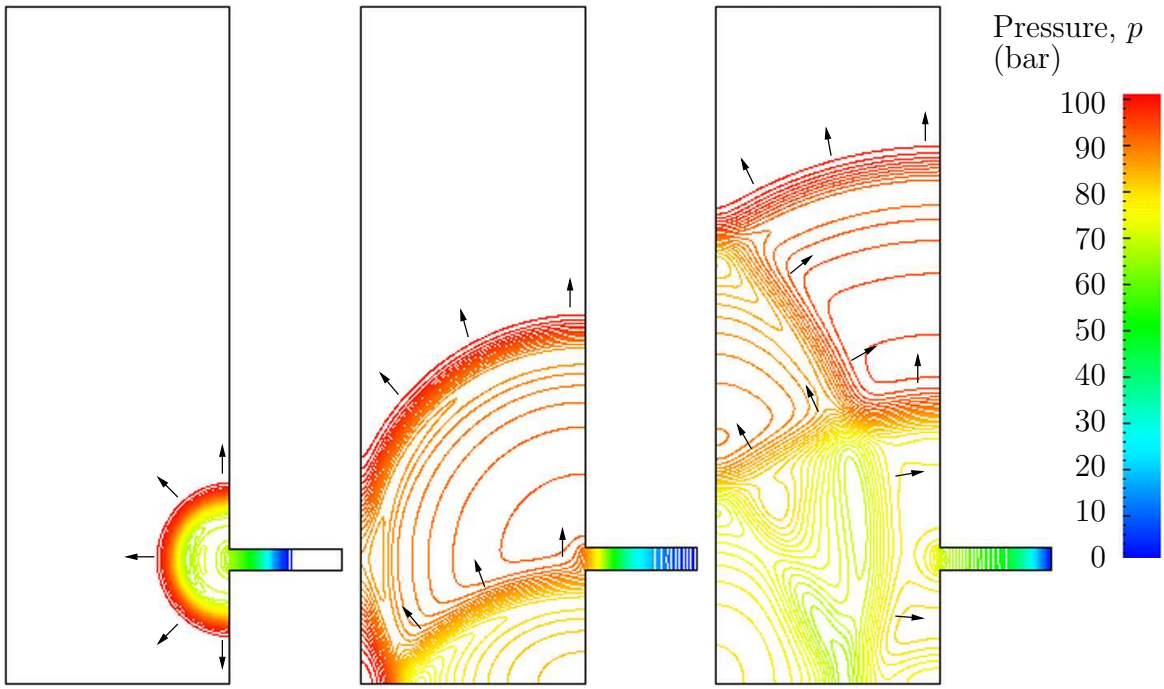


(a) At $t = 50$ µs (b) At $t = 100$ µs (c) At $t = 150$ µs

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.



(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

3.4.5 Improving the solution by refining the 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.

Governing equations

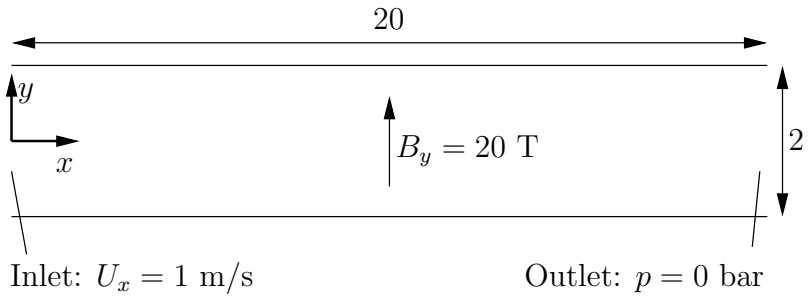


Figure 3.12: Geometry of the Hartmann problem

- Mass continuity for incompressible fluid

$$\nabla \bullet \mathbf{U} = 0 \tag{3.19}$$

- Momentum equation for incompressible fluid

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \bullet (\mathbf{U}\mathbf{U}) + \nabla \bullet (2\mathbf{B}\Gamma_{\mathbf{BU}}\mathbf{B}) + \nabla \bullet (\nu \mathbf{U}) + \nabla (\Gamma_{\mathbf{BU}}\mathbf{B} : \mathbf{B}) = -\nabla p \tag{3.20}$$

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

- Maxwell's equations

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \tag{3.21}$$

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

$$\nabla \bullet \mathbf{B} = 0 \tag{3.22}$$

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} = \mathbf{J} \tag{3.23}$$

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

$$\nabla \bullet \mathbf{J} = 0 \tag{3.24}$$

- Constitutive law

$$\mathbf{B} = \mu \mathbf{H} \tag{3.25}$$

- Ohm's law

$$\mathbf{J} = \sigma (\mathbf{E} + \mathbf{U} \times \mathbf{B}) \tag{3.26}$$

- Combining Equation 3.21, Equation 3.23, Equation 3.26, and taking the curl

$$\frac{\partial \mathbf{B}}{\partial t} + \nabla \bullet (\mathbf{U}\mathbf{B}) - \nabla \bullet (\phi_{\mathbf{B}}\mathbf{U}) - \nabla \bullet (\Gamma_{\mathbf{B}}\mathbf{B}) = 0 \tag{3.27}$$

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:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                      |                                            |
| \\      /  F ield              | OpenFOAM: The Open Source CFD Toolbox      |
|  \\    /   O peration          | Version:  2.0.0                            |
|   \\  /    A nd                | Web:      www.OpenFOAM.com                 |
|    \\/     M anipulation       |                                            |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices
(
    (0 -1 0)
    (20 -1 0)
    (20 1 0)
    (0 1 0)
    (0 -1 0.1)
    (20 -1 0.1)
    (20 1 0.1)
    (0 1 0.1)
);

blocks
(
    hex (0 1 2 3 4 5 6 7) (100 40 1) simpleGrading (1 1 1)
);

edges
```

```
37  (
38  );
39
40  boundary
41  (
42      inlet
43      {
44          type patch;
45          faces
46          (
47              (0 4 7 3)
48          );
49      }
50      outlet
51      {
52          type patch;
53          faces
54          (
55              (2 6 5 1)
56          );
57      }
58      lowerWall
59      {
60          type patch;
61          faces
62          (
63              (1 5 4 0)
64          );
65      }
66      upperWall
67      {
68          type patch;
69          faces
70          (
71              (3 7 6 2)
72          );
73      }
74      frontAndBack
75      {
76          type empty;
77          faces
78          (
79              (0 3 2 1)
80              (4 5 6 7)
81          );
82      }
83  );
84
85  mergePatchPairs
86  (
87  );
88
89  // ************************************************************************* //
```

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

$$M = BL\sqrt{\frac{\sigma}{\rho\nu}} \tag{3.28}$$

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 Ucomponents. In this case, $M = 1$ and the electromagnetic body forces no longer dominate.

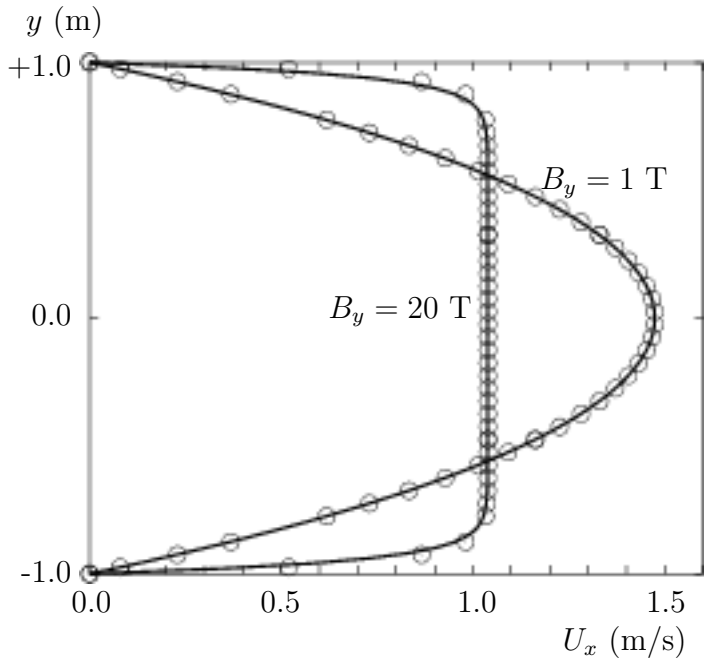


Figure 3.13: Velocity profile in the Hartmann problem for $B_y = 1$ T and $B_y = 20$ T.

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:

$$\frac{U_x(y)}{U_x(0)} = \frac{\cosh M - \cosh M(y/L)}{\cosh M - 1} \tag{3.29}$$

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-108
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-192
<RASModel>Coeffs keyword, U-192
<delta>Coeffs keyword, U-192
0.000000e+00 directory, U-108
1-dimensional mesh, U-135
1D mesh, U-135
2-dimensional mesh, U-135
2D mesh, U-135

Numbers

0 directory, U-108

A

access functions, P-23
addLayersControls keyword, U-151
adiabaticFlameT utility, U-99
adjointShapeOptimizationFoam solver, U-88
adjustableRunTime keyword entry, U-64, U-116
adjustTimeStep keyword, U-63
agglomerator keyword, U-127
algorithms tools, U-100
alphaContactAngle boundary condition, U-61
analytical solution, P-45
Animations window panel, U-174
anisotropicFilter model, U-105
Annotation window panel, U-27, U-174
ansysToFoam utility, U-93
APIfunctions model, U-104
applications, U-71
Apply button, U-170, U-174
applyBoundaryLayer utility, U-93
applyWallFunctionBoundaryConditions utility, U-93
arbitrarily unstructured, P-31
arc keyword entry, U-144
arc keyword, U-143
As keyword, U-190
ascii keyword entry, U-116
attachMesh utility, U-94
Auto Accept button, U-174
autoMesh library, U-101
autoPatch utility, U-94
autoRefineMesh utility, U-95
axes right-handed, U-141 right-handed rectangular Cartesian, P-15, U-20
axi-symmetric cases, U-140, U-149
axi-symmetric mesh, U-135

B

background
process, U-27, U-84
backward
keyword entry, U-124
Backward differencing, P-39
barotropicCompressibilityModels
library, U-103
basicMultiComponentMixture model, U-103, U-188
basicSolidThermo
library, U-104
basicThermophysicalModels
library, U-102
binary
keyword entry, U-116
BirdCarreau model, U-106
blended differencing, P-38
block
expansion ratio, U-145
block keyword, U-143
blocking
keyword entry, U-83
blockMesh
library, U-101
blockMesh solver, P-47
blockMesh utility, U-39, U-93, U-141
blockMesh executable
vertex numbering, U-145
blockMeshDict
dictionary, U-20, U-22, U-38, U-51, U-141, U-150
blocks keyword, U-22, U-33, U-145
boundaries, U-135
boundary, U-135
boundary
dictionary, U-134, U-141
boundary keyword, U-146
boundary condition
alphaContactAngle, U-61
calculated, U-141
cyclic, U-140, U-147
directionMixed, U-141
empty, P-64, P-70, U-20, U-135, U-140
fixedGradient, U-141
fixedValue, U-141
fluxCorrectedVelocity, U-142
inlet, P-70
inletOutlet, U-142
mixed, U-141
movingWallVelocity, U-142
outlet, P-70
outletInlet, U-142
partialSlip, U-142
patch, U-139
pressureDirectedInletVelocity, U-142
pressureInletVelocity, U-142
pressureOutlet, P-64
pressureTransmissive, U-142
processor, U-140
setup, U-22
slip, U-142
supersonicFreeStream, U-142
surfaceNormalFixedValue, U-142
symmetryPlane, P-64, U-140
totalPressure, U-142
turbulentInlet, U-142
wall, U-42
wall, P-64, P-70, U-60, U-139
wallBuoyantPressure, U-142
wedge, U-135, U-140, U-149
zeroGradient, U-141
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-112
boundaryFoam solver, U-88
bounded
keyword entry, U-122, U-123
boxToCell keyword, U-62
boxTurb utility, U-93
breaking of a dam, U-58
bubbleFoam solver, U-89
buoyantBaffleSimpleFoam solver, U-91
buoyantBoussinesqPimpleFoam solver, U-91
buoyantBoussinesqSimpleFoam solver, U-91
buoyantPimpleFoam solver, U-91
buoyantSimpleFoam solver, U-91
buoyantSimpleRadiationFoam solver, U-91
button
Apply, U-170, U-174
Auto Accept, U-174
Choose Preset, U-172
Delete, U-170
Edit Color Map, U-172

Enable Line Series, U-37
Orientation Axes, U-27, U-174
Refresh Times, U-27
Rescale to Data Range, U-27
Reset, U-170
Set Ambient Color, U-173
Update GUI, U-171
Use Parallel Projection, U-27
Use parallel projection, U-173

C

C++ syntax
/*...*/, U-81
//, U-81
include, U-74, U-81
cacheAgglomeration keyword, U-128
calculated
boundary condition, U-141
cAlpha keyword, U-65
cases, U-107
castellatedMesh keyword, U-151
castellatedMeshControls
dictionary, U-153–U-155
castellatedMeshControls keyword, U-151
cavitatingFoam solver, U-89
cavity flow, U-19
CEI_ARCH
environment variable, U-179
CEI_HOME
environment variable, U-179
cell
expansion ratio, U-145
cell class, P-31
cell
keyword entry, U-181
cellLimited
keyword entry, U-122
cellPoint
keyword entry, U-181
cellPointFace
keyword entry, U-181
cells
dictionary, U-141
central differencing, P-38
cfdTools tools, U-100
cfx4ToFoam utility, U-93, U-159
changeDictionary utility, U-93
channelFoam solver, U-88
Charts window panel, U-174
checkMesh utility, U-94, U-161
chemFoam solver, U-90
chemistryModel
library, U-104
chemistryModel model, U-104
chemistrySolver model, U-104
chemkinToFoam utility, U-99
Choose Preset button, U-172
chtMultiRegionFoam solver, U-91
Chung
library, U-103
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-131, U-133
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-111
word, P-26, P-31
class keyword, U-109
clockTime
keyword entry, U-116
cloud keyword, U-182
cmptAv
tensor member function, P-25
Co utility, U-96
coalChemistryFoam solver, U-91
coalCombustion
library, U-101
cofactors
tensor member function, P-25
coldEngineFoam solver, U-90
collapseEdges utility, U-95
Color By menu, U-173

Color Legend window, U-29
Color Legend window panel, U-172
Color Scale window panel, U-172
Colors window panel, U-174
combinePatchFaces utility, U-95
comments, U-81
commsType keyword, U-83
compressed
    keyword entry, U-116
compressibleInterFoam solver, U-90
compressibleLESModels
    library, U-106
compressibleRASModels
    library, U-105
constant directory, U-107, U-187
constLaminarFlameSpeed model, U-103
constTransport model, U-103, U-188
containers tools, U-100
continuum
    mechanics, P-15
control
    of time, U-115
controlDict
    dictionary, P-66, U-24, U-33, U-44, U-53, U-64, U-107, U-166
controlDict file, P-50
convection, *see* divergence, P-38
convergence, U-41
conversion
    library, U-101
convertToMeters keyword, U-143, U-144
coordinate
    system, P-15
coordinate system, U-20
corrected
    keyword entry, U-122, U-123
Courant number, P-42, U-24
Cp keyword, U-189
cpuTime
    keyword entry, U-116
Crank Nicholson
    temporal discretisation, P-43
CrankNicholson
    keyword entry, U-124
createBaffles utility, U-94
createPatch utility, U-94
createTurbulenceFields utility, U-97
cross product, *see* tensor, vector cross product
CrossPowerLaw
    keyword entry, U-62
CrossPowerLaw model, U-106
cubeRootVolDelta model, U-105
cubicCorrected
    keyword entry, U-124
cubicCorrection
    keyword entry, U-121
curl, P-37
curl
    fvc member function, P-37
Current Time Controls menu, U-27, U-171
curve keyword, U-182
Cv keyword, U-189
cyclic
    boundary condition, U-140, U-147
cyclic
    keyword entry, U-140
cylinder
    flow around a, P-45

D

d2dt2
    fvc member function, P-37
    fvm member function, P-37
dam
    breaking of a, U-58
datToFoam utility, U-93
db tools, U-100
ddt
    fvc member function, P-37
    fvm member function, P-37
DeardorffDiffStress model, U-105, U-106
debug keyword, U-151
decomposePar utility, U-84, U-85, U-99
decomposeParDict
    dictionary, U-84
decomposition
    of field, U-84
    of mesh, U-84
decompositionMethods
    library, U-101
decompression of a tank, P-62
defaultFieldValues keyword, U-62
deformedGeom utility, U-94
Delete button, U-170
delta keyword, U-86, U-192
deltaT keyword, U-116
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-126, U-127
DIC
keyword entry, U-127
DICGaussSeidel
keyword entry, U-127
dictionary
LESProperties, U-192
PISO, U-25
blockMeshDict, U-20, U-22, U-38, U-51, U-141, U-150
boundary, U-134, U-141
castellatedMeshControls, U-153–U-155
cells, U-141
controlDict, P-66, U-24, U-33, U-44, U-53, U-64, U-107, U-166
decomposeParDict, U-84
faces, U-133, U-141
fvSchemes, U-64, U-65, U-107, U-118
fvSolution, U-107, U-125
mechanicalProperties, U-53
neighbour, U-134
owner, U-133
points, U-133, U-141
thermalProperties, U-53
thermophysicalProperties, U-187
transportProperties, U-23, U-40, U-44
turbulenceProperties, U-43, U-63, U-191
dieselEngineFoam solver, U-91
dieselFoam solver, U-91
dieselMixture model, U-103, U-188
dieselSpray
library, U-101
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-127
dimension
checking in OpenFOAM, P-26, U-111
dimensional units, U-111
dimensioned<Type> template class, P-26
dimensionedTypes tools, U-100
dimensions keyword, U-23, U-112
dimensionSet class, P-26, P-32, P-33
dimensionSet tools, U-100
directionMixed
boundary condition, U-141
directory
0.000000e+00, U-108
0, U-108
Make, U-75
constant, U-107, U-187
fluentInterface, U-176
polyMesh, U-107, U-133
processorN, U-85
run, U-107
system, P-50, U-107
tutorials, P-45, U-19
discretisation
equation, P-33
Display window panel, U-26, U-27, U-170, U-171
distance
keyword entry, U-155, U-182
distributed model, U-102
distributed keyword, U-86, U-87
distributionModels
library, U-101
div
fvc member function, P-37
fvm member function, P-37
divergence, P-37, P-39
divSchemes keyword, U-118
dnsFoam solver, U-90
doLayers keyword, U-151
double inner product, *see* tensor,double inner product
dsmc
library, U-101
dsmcFieldsCalc utility, U-97
dsmcFoam solver, U-92
dsmcInitialise utility, U-93
dx
keyword entry, U-181
dynamicFvMesh
library, U-101
dynamicMesh
library, U-101
dynLagrangian model, U-105

dynMixedSmagorinsky model, U-105
dynOneEqEddy model, U-105, U-106
dynSmagorinsky model, U-105

E

eConstThermo model, U-103, U-187
edgeGrading keyword, U-145
edgeMesh
  library, U-101
edges keyword, U-143
Edit menu, U-173, U-174
Edit Color Map button, U-172
egrMixture model, U-103, U-188
electrostaticFoam solver, U-92
empty
  boundary condition, P-64, P-70, U-20, U-135, U-140
empty
  keyword entry, U-140
Enable Line Series button, U-37
endTime keyword, U-24, U-115, U-116
engine
  library, U-101
engineCompRatio utility, U-97
engineFoam solver, U-91
engineSwirl utility, U-93
ensight74FoamExec utility, U-179
ENSIGHT7_INPUT
  environment variable, U-179
ENSIGHT7_READER
  environment variable, U-179
ensightFoamReader utility, U-95
enstrophy utility, U-96
environment variable
  CEI_ARCH, U-179
  CEI_HOME, U-179
  ENSIGHT7_INPUT, U-179
  ENSIGHT7_READER, U-179
  FOAM_RUN, U-107
  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_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_USER_DIR, U-78
  WM_PROJECT_VERSION, U-78
  WM_PROJECT, U-78
  wmake, U-78
ePsiThermo model, U-102, U-188
equilibriumCO utility, U-99
equilibriumFlameT utility, U-99
errorReduction keyword, U-159
Euler
  keyword entry, U-124
Euler implicit
  differencing, P-39
  temporal discretisation, P-42
examples
  decompression of a tank, P-62
  flow around a cylinder, P-45
  flow over backward step, P-53
  Hartmann problem, P-68
  supersonic flow over forward step, P-59
execFlowFunctionObjects utility, U-97
expandDictionary utility, U-99
expansionRatio keyword, U-158
explicit
  temporal discretisation, P-42
extrude2DMesh utility, U-93
extrudeMesh utility, U-93
extrudeToRegionMesh utility, U-93

F

face class, P-31
face keyword, U-182
faceAgglomerate utility, U-93
faceAreaPair
  keyword entry, U-127
faceLimited
  keyword entry, U-122
faces
  dictionary, U-133, U-141
FDIC
  keyword entry, U-127
featureAngle keyword, U-158
features keyword, U-153
field
  U, U-25
  p, U-25
  decomposition, U-84
FieldField<Type> template class, P-32
fieldFunctionObjects
  library, U-101
fields, P-29

mapping, U-166
fields tools, U-100
`fields` keyword, U-181
Field<Type> template class, P-29
`fieldValues` keyword, U-62
fieldview9Reader utility, U-95
file
Make/files, U-77
controlDict, P-50
files, U-75
g, U-63
options, U-75
snappyHexMeshDict, U-151
transportProperties, U-62
file format, U-108
fileFormats
library, U-102
`fileModificationChecking` keyword, U-83
`fileModificationSkew` keyword, U-83
files file, U-75
`filteredLinear2`
keyword entry, U-121
`finalLayerRatio` keyword, U-158
financialFoam solver, U-92
finite volume
discretisation, P-27
mesh, P-31
finiteVolume
library, U-100
finiteVolume tools, U-100
finiteVolumeCalculus class, P-36
finiteVolumeMethod class, P-36
fireFoam solver, U-91
`firstTime` keyword, U-115
`fixed`
keyword entry, U-116
fixedGradient
boundary condition, U-141
fixedValue
boundary condition, U-141
flattenMesh utility, U-94
`floatTransfer` keyword, U-83
flow
free surface, U-58
laminar, U-19
steady, turbulent, P-53
supersonic, P-59
turbulent, U-19
flow around a cylinder, P-45
flow over backward step, P-53
flowType utility, U-96
fluent3DMeshToFoam utility, U-93
fluentInterface directory, U-176
fluentMeshToFoam utility, U-93, U-159
fluxCorrectedVelocity
boundary condition, U-142
`fluxRequired` keyword, U-118
OpenFOAM
cases, U-107
FOAM_RUN
environment variable, U-107
foamCalc utility, U-34, U-97
foamCalcFunctions
library, U-101
foamCorrectVrt script/alias, U-164
foamDataToFluent utility, U-96, U-176
foamDebugSwitches utility, U-99
`FoamFile` keyword, U-109
`foamFile`
keyword entry, U-181
foamFormatConvert utility, U-99
foamInfoExec utility, U-99
foamJob script/alias, U-184
foamListTimes utility, U-97
foamLog script/alias, U-184
foamMeshToFluent utility, U-94, U-176
foamToEnsight utility, U-96
foamToEnsightParts utility, U-96
foamToFieldview9 utility, U-96
foamToGMV utility, U-96
foamToStarMesh utility, U-94
foamToSurface utility, U-94
foamToTecplot360 utility, U-96
foamToVTK utility, U-96
foamUpgradeCyclics utility, U-93
foamUpgradeFvSolution utility, U-93
forces
library, U-101
foreground
process, U-27
`format` keyword, U-109
`fourth`
keyword entry, U-122, U-123
`functions` keyword, U-117
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-103
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-100
fvMatrix template class, P-36
fvMesh class, P-31
fvMesh tools, U-100
fvMotionSolvers
library, U-101
fvSchemes
dictionary, U-64, U-65, U-107, U-118
fvSchemes class, P-38
fvSchemes
menu entry, U-54
fvSolution
dictionary, U-107, U-125

G

g file, U-63
gambitToFoam utility, U-94, U-159
GAMG
keyword entry, U-55, U-126, U-127
Gamma
keyword entry, U-121
Gamma differencing, P-38
Gauss
keyword entry, U-122
Gauss's theorem, P-36
GaussSeidel
keyword entry, U-127
General window panel, U-173, U-174
general
keyword entry, U-116
genericFvPatchField
library, U-102
geometric-algebraic multi-grid, U-127
GeometricBoundaryField template class, P-32
geometricField<Type> template class, P-32
geometry keyword, U-151
gGrad
fvc member function, P-37
global tools, U-100
gmshToFoam utility, U-94
gnuplot
keyword entry, U-117, U-181
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-118
graph tools, U-100
graphFormat keyword, U-117
GuldersEGRLaminarFlameSpeed model, U-103
GuldersLaminarFlameSpeed model, U-103

H

hConstThermo model, U-103, U-187
Help menu, U-173
HerschelBulkley model, U-106
Hf keyword, U-189
hhuMixtureThermo model, U-102, U-189
hierarchical
keyword entry, U-85, U-86
highCpCoeffs keyword, U-190
homogenousDynSmagorinsky model, U-105
homogeneousMixture model, U-103, U-188
hPolynomialThermo model, U-103, U-187
hPsiMixtureThermo model, U-102, U-188
hPsiThermo model, U-102, U-188
hRhoMixtureThermo model, U-102, U-189
hRhoThermo model, U-102, U-188
hsPsiMixtureThermo model, U-102, U-188
hsPsiThermo model, U-102, U-188
hsRhoMixtureThermo model, U-102, U-189

I

I
tensor member function, P-25
icoFoam solver, U-19, U-23, U-24, U-27, U-88
icoPolynomial model, U-103, U-187
icoUncoupledKinematicParcelDyMFoam solver, U-91
icoUncoupledKinematicParcelFoam solver, U-91

ideasToFoam utility, U-159
ideasUnvToFoam utility, U-94
identities, *see* tensor, identities
identity, *see* tensor, identity
incompressibleLESModels
    library, U-105
incompressibleRASModels
    library, U-104
incompressibleTransportModels
    library, P-54, U-106
incompressibleTurbulenceModels
    library, P-54
index
    notation, P-16, P-17
Information window panel, U-170
inhomogeneousMixture model, U-103, U-188
inlet
    boundary condition, P-70
inletOutlet
    boundary condition, U-142
inner product, *see* tensor, inner product
inotify
    keyword entry, U-83
inotifyMaster
    keyword entry, U-83
inside
    keyword entry, U-155
insideCells utility, U-94
interDyMFoam solver, U-90
interfaceProperties
    library, U-106
interfaceProperties model, U-106
interFoam solver, U-90
interMixingFoam solver, U-90
internalField keyword, U-23, U-112
interPhaseChangeFoam solver, U-90
interpolation tools, U-100
interpolationScheme keyword, U-181
interpolations tools, U-100
interpolationSchemes keyword, U-118
inv
    tensor member function, P-25
iterations
    maximum, U-126

J

janafThermo model, U-103, U-188
jobControl
    library, U-101
jplot
    keyword entry, U-117, U-181

K

kEpsilon model, U-104, U-105
keyword
    As, U-190
    Cp, U-189
    Cv, U-189
    FoamFile, U-109
    Hf, U-189
    LESModel, U-192
    Pr, U-190
    RASModel, U-192
    Tcommon, U-190
    Thigh, U-190
    Tlow, U-190
    Ts, U-190
    addLayersControls, U-151
    adjustTimeStep, U-63
    agglomerator, U-127
    arc, U-143
    blocks, U-22, U-33, U-145
    block, U-143
    boundaryField, U-23, U-112
    boundary, U-146
    boxToCell, U-62
    cAlpha, U-65
    cacheAgglomeration, U-128
    castellatedMeshControls, U-151
    castellatedMesh, U-151
    class, U-109
    cloud, U-182
    commsType, U-83
    convertToMeters, U-143, U-144
    curve, U-182
    debug, U-151
    defaultFieldValues, U-62
    deltaT, U-116
    delta, U-86, U-192
    dimensions, U-23, U-112
    distributed, U-86, U-87
    divSchemes, U-118
    doLayers, U-151
    edgeGrading, U-145
    edges, U-143
    endTime, U-24, U-115, U-116
    errorReduction, U-159
    expansionRatio, U-158
    face, U-182
    featureAngle, U-158
    features, U-153
    fieldValues, U-62

fields, U-181
fileModificationChecking, U-83
fileModificationSkew, U-83
finalLayerRatio, U-158
firstTime, U-115
floatTransfer, U-83
fluxRequired, U-118
format, U-109
functions, U-117
geometry, U-151
gradSchemes, U-118
graphFormat, U-117
highCpCoeffs, U-190
internalField, U-23, U-112
interpolationSchemes, U-118
interpolationScheme, U-181
laplacianSchemes, U-118
latestTime, U-40
layers, U-158
leastSquares, U-54
levels, U-155
libs, U-82, U-117
locationInMesh, U-153, U-155
location, U-109
lowCpCoeffs, U-190
manualCoeffs, U-86
maxAlphaCo, U-63
maxBoundarySkewness, U-159
maxConcave, U-159
maxCo, U-63
maxDeltaT, U-63
maxFaceThicknessRatio, U-158
maxGlobalCells, U-153
maxInternalSkewness, U-159
maxIter, U-126
maxLocalCells, U-153
maxNonOrtho, U-159
maxThicknessToMedialRatio, U-158
mergeLevels, U-128
mergePatchPairs, U-143
mergeTolerance, U-151
meshQualityControls, U-151
method, U-86
midPointAndFace, U-182
midPoint, U-182
minArea, U-159
minDeterminant, U-159
minFaceWeight, U-159
minFlatness, U-159
minMedianAxisAngle, U-158
minRefinementCells, U-153
minThickness, U-158
minTriangleTwist, U-159
minTwist, U-159
minVolRatio, U-159
minVol, U-159
mode, U-155
molWeight, U-189
mu, U-190
nAlphaSubCycles, U-65
nBufferCellsNoExtrude, U-158
nCellsBetweenLevels, U-153
nFaces, U-134
nFinestSweeps, U-128
nGrow, U-158
nLayerIter, U-158
nMoles, U-189
nPostSweeps, U-128
nPreSweeps, U-128
nRelaxIter, U-156, U-158
nRelaxedIter, U-158
nSmoothNormals, U-158
nSmoothPatch, U-156
nSmoothScale, U-159
nSmoothSurfaceNormals, U-158
nSmoothThickness, U-158
nSolveIter, U-156
neighbourPatch, U-147
numberOfSubdomains, U-86
n, U-86
object, U-109
order, U-86
pRefCell, U-25, U-130
pRefValue, U-25, U-130
p_rhgRefCell, U-130
p_rhgRefValue, U-130
patchMap, U-166
patches, U-143
preconditioner, U-126, U-127
pressure, U-53
printCeoffs, U-192
printCoeffs, U-43
processorWeights, U-85
processorWeights, U-86
purgeWrite, U-116
refGradient, U-141
refinementRegions, U-153, U-155
refinementSurfaces, U-153, U-154
refinementRegions, U-155
regions, U-62

relTol, U-55, U-126
relativeSizes, U-158
relaxed, U-159
resolveFeatureAngle, U-153, U-154
roots, U-86, U-87
runTimeModifiable, U-117
scotchCoeffs, U-86
setFormat, U-181
sets, U-181
simpleGrading, U-145
simulationType, U-43, U-63, U-191
smoother, U-128
snGradSchemes, U-118
snapControls, U-151
snap, U-151
solvers, U-125
solver, U-55, U-125
specie, U-189
spline, U-143
startFace, U-134
startFrom, U-24, U-115
startTime, U-24, U-115
stopAt, U-115
strategy, U-85, U-86
surfaceFormat, U-181
surfaces, U-181
thermoType, U-187
thermodynamics, U-189
timeFormat, U-116
timePrecision, U-117
timeScheme, U-118
tolerance, U-55, U-126, U-156
topoSetSource, U-62
traction, U-53
transport, U-189
turbulence, U-192
type, U-135, U-138
uniform, U-182
valueFraction, U-141
value, U-23, U-141
version, U-109
vertices, U-22, U-143, U-144
writeCompression, U-116
writeControl, U-24, U-64, U-116
writeFormat, U-57, U-116
writeInterval, U-24, U-34, U-116
writePrecision, U-116
<LESModel>Coeffs, U-192
<RASModel>Coeffs, U-192
<delta>Coeffs, U-192

keyword entry
CrankNicholson, U-124
CrossPowerLaw, U-62
DICGaussSeidel, U-127
DIC, U-127
DILU, U-127
Euler, U-124
FDIC, U-127
GAMG, U-55, U-126, U-127
Gamma, U-121
GaussSeidel, U-127
Gauss, U-122
LESModel, U-43, U-191
MGridGen, U-127
MUSCL, U-121
Newtonian, U-62
PBiCG, U-126
PCG, U-126
QUICK, U-124
RASModel, U-43, U-191
SFCD, U-121, U-124
UMIST, U-119
adjustableRunTime, U-64, U-116
arc, U-144
ascii, U-116
backward, U-124
binary, U-116
blocking, U-83
bounded, U-122, U-123
cellLimited, U-122
cellPointFace, U-181
cellPoint, U-181
cell, U-181
clockTime, U-116
compressed, U-116
corrected, U-122, U-123
cpuTime, U-116
cubicCorrected, U-124
cubicCorrection, U-121
cyclic, U-140
diagonal, U-126, U-127
distance, U-155, U-182
dx, U-181
empty, U-140
faceAreaPair, U-127
faceLimited, U-122
filteredLinear2, U-121
fixed, U-116
foamFile, U-181
fourth, U-122, U-123

general, U-116
gnuplot, U-117, U-181
hierarchical, U-85, U-86
inotifyMaster, U-83
inotify, U-83
inside, U-155
jplot, U-117, U-181
laminar, U-43, U-191
latestTime, U-115
leastSquares, U-122
limitedCubic, U-121
limitedLinear, U-121
limited, U-122, U-123
linearUpwind, U-121, U-124
linear, U-121, U-124
line, U-144
localEuler, U-124
manual, U-85, U-86
metis, U-86
midPoint, U-121
nextWrite, U-116
noWriteNow, U-116
nonBlocking, U-83
none, U-119, U-127
null, U-181
outside, U-155
patch, U-140, U-182
polyLine, U-144
polySpline, U-144
processor, U-140
raw, U-117, U-181
runTime, U-34, U-116
scheduled, U-83
scientific, U-116
scotch, U-85, U-86
simpleSpline, U-144
simple, U-85, U-86
skewLinear, U-121, U-124
smoothSolver, U-126
startTime, U-24, U-115
steadyState, U-124
stl, U-181
symmetryPlane, U-140
timeStampMaster, U-83
timeStamp, U-83
timeStep, U-24, U-34, U-116
uncompressed, U-116
uncorrected, U-122, U-123
upwind, U-121, U-124
vanLeer, U-121
vtk, U-181
wall, U-140
wedge, U-140
writeControl, U-116
writeNow, U-115
xmgr, U-117, U-181
xyz, U-182
x, U-182
y, U-182
z, U-182
kivaToFoam utility, U-94
kOmega model, U-104
kOmegaSST model, U-104, U-105
kOmegaSSTSAS model, U-105
Kronecker delta, P-21

L

lagrangian
library, U-101
lagrangianIntermediate
library, U-101
Lambda2 utility, U-96
LamBremhorstKE model, U-104
laminar model, U-104, U-105
laminar
keyword entry, U-43, U-191
laminarFlameSpeedModels
library, U-103
laplaceFilter model, U-105
Laplacian, P-38
laplacian, P-37
laplacian
fvc member function, P-37
fvm member function, P-37
laplacianFoam solver, U-88
laplacianSchemes keyword, U-118
latestTime
keyword entry, U-115
latestTime keyword, U-40
LaunderGibsonRSTM model, U-104, U-105
LaunderSharmaKE model, U-104, U-105
layers keyword, U-158
leastSquares
keyword entry, U-122
leastSquares keyword, U-54
LESdeltas
library, U-105
LESfilters
library, U-105
LESModel
keyword entry, U-43, U-191

`LESModel` keyword, U-192
LESProperties
dictionary, U-192
`levels` keyword, U-155
libraries, U-71
library
Chung, U-103
LESdeltas, U-105
LESfilters, U-105
MGridGenGAMGAgglomeration, U-102
ODE, U-101
OSspecific, U-102
OpenFOAM, U-100
P1, U-103
PV3FoamReader, U-169
PVFoamReader, U-169
SLGThermo, U-104
Wallis, U-103
autoMesh, U-101
barotropicCompressibilityModels, U-103
basicSolidThermo, U-104
basicThermophysicalModels, U-102
blockMesh, U-101
chemistryModel, U-104
coalCombustion, U-101
compressibleLESModels, U-106
compressibleRASModels, U-105
conversion, U-101
decompositionMethods, U-101
dieselSpray, U-101
distributionModels, U-101
dsmc, U-101
dynamicFvMesh, U-101
dynamicMesh, U-101
edgeMesh, U-101
engine, U-101
fieldFunctionObjects, U-101
fileFormats, U-102
finiteVolume, U-100
foamCalcFunctions, U-101
forces, U-101
fvDOM, U-103
fvMotionSolvers, U-101
genericFvPatchField, U-102
incompressibleLESModels, U-105
incompressibleRASModels, U-104
incompressibleTransportModels, P-54, U-106
incompressibleTurbulenceModels, P-54
interfaceProperties, U-106
jobControl, U-101
lagrangianIntermediate, U-101
lagrangian, U-101
laminarFlameSpeedModels, U-103
linear, U-103
liquidMixtureProperties, U-104
liquidProperties, U-104
meshTools, U-101
molecularMeasurements, U-101
molecule, U-101
pairPatchAgglomeration, U-102
postCalc, U-101
potential, U-101
primitive, P-23
radiationModels, U-103
randomProcesses, U-102
reactionThermophysicalModels, U-102
sampling, U-101
solidMixtureProperties, U-104
solidParticle, U-101
solidProperties, U-104
solid, U-104
specie, U-103
surfMesh, U-101
surfaceFilmModels, U-106
systemCall, U-101
thermalPorousZone, U-104
thermophysicalFunctions, U-104
thermophysical, U-187
topoChangerFvMesh, U-101
triSurface, U-101
twoPhaseInterfaceProperties, U-106
utilityFunctionObjects, U-101
viewFactor, U-103
vtkFoam, U-169
vtkPV3Foam, U-169
`libs` keyword, U-82, U-117
lid-driven cavity flow, U-19
LienCubicKE model, U-104
LienCubicKELowRe model, U-104
LienLeschzinerLowRe model, U-104
Lights window panel, U-174
`limited`
keyword entry, U-122, U-123
`limitedCubic`
keyword entry, U-121
`limitedLinear`
keyword entry, U-121
`line`
keyword entry, U-144
`Line Style` menu, U-37

linear
library, U-103
`linear`
keyword entry, U-121, U-124
`linearUpwind`
keyword entry, U-121, U-124
liquid
electrically-conducting, P-68
liquidMixtureProperties
library, U-104
liquidProperties
library, U-104
lists, P-29
List<Type> template class, P-29
`localEuler`
keyword entry, U-124
`location` keyword, U-109
`locationInMesh` keyword, U-153, U-155
locDynOneEqEddy model, U-105
`lowCpCoeffs` keyword, U-190
lowReOneEqEddy model, U-106
LRDDiffStress model, U-105
LRR model, U-104, U-105
`lsGrad`
fvc member function, P-37
LTSInterFoam solver, U-90
LTSReactingParcelFoam solver, U-91

M

Mach utility, U-96
`mag`
tensor member function, P-25
magneticFoam solver, U-92
magnetohydrodynamics, P-68
`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-93, U-166
mapping
fields, U-166
`Marker Style` menu, U-37
matrices tools, U-100
`max`
tensor member function, P-25
`maxAlphaCo` keyword, U-63
`maxBoundarySkewness` keyword, U-159
`maxCo` keyword, U-63
`maxConcave` keyword, U-159
`maxDeltaT` keyword, U-63
maxDeltaxyz model, U-105
`maxFaceThicknessRatio` keyword, U-158
`maxGlobalCells` keyword, U-153
maximum iterations, U-126
`maxInternalSkewness` keyword, U-159
`maxIter` keyword, U-126
`maxLocalCells` keyword, U-153
`maxNonOrtho` keyword, U-159
`maxThicknessToMedialRatio` keyword, U-158
mdEquilibrationFoam solver, U-92
mdFoam solver, U-92
mdInitialise utility, U-93
mechanicalProperties
dictionary, U-53
memory tools, U-100
menu
`Color By`, U-173
`Current Time Controls`, U-27, U-171
`Edit`, U-173, U-174
`Help`, U-173
`Line Style`, U-37
`Marker Style`, U-37
`VCR Controls`, U-27, U-171
`View`, U-173
menu entry
`Plot Over Line`, U-36
`Save Animation`, U-175
`Save Screenshot`, U-175
`Settings`, U-174
`Show Color Legend`, U-29
`Solid Color`, U-173
`Toolbars`, U-173
`View Settings...`, U-26
`View Settings`, U-27, U-173
`Wireframe`, U-173
`fvSchemes`, U-54
`mergeLevels` keyword, U-128
mergeMeshes utility, U-94
mergeOrSplitBaffles utility, U-94
`mergePatchPairs` keyword, U-143
`mergeTolerance` keyword, U-151
mesh
1-dimensional, U-135
1D, U-135
2-dimensional, U-135
2D, U-135

axi-symmetric, U-135
basic, P-31
block structured, U-141
decomposition, U-84
description, U-131
finite volume, P-31
generation, U-141, U-150
grading, U-141, U-145
grading, example of, P-53
non-orthogonal, P-45
refinement, P-62
resolution, U-31
specification, U-131
split-hex, U-150
Stereolithography (STL), U-150
surface, U-150
validity constraints, U-131
Mesh Parts window panel, U-26
meshes tools, U-100
meshQualityControls keyword, U-151
meshTools
library, U-101
message passing interface
openMPI, U-85
method keyword, U-86
metis
keyword entry, U-86
MGridGenGAMGAgglomeration
library, U-102
MGridGen
keyword entry, U-127
mhdFoam solver, P-70, U-92
midPoint
keyword entry, U-121
midPoint keyword, U-182
midPointAndFace keyword, U-182
min
tensor member function, P-25
minArea keyword, U-159
minDeterminant keyword, U-159
minFaceWeight keyword, U-159
minFlatness keyword, U-159
minMedianAxisAngle keyword, U-158
MINMOD differencing, P-38
minRefinementCells keyword, U-153
minThickness keyword, U-158
minTriangleTwist keyword, U-159
minTwist keyword, U-159
minVol keyword, U-159
minVolRatio keyword, U-159
mirrorMesh utility, U-94
mixed
boundary condition, U-141
mixedSmagorinsky model, U-105
mixtureAdiabaticFlameT utility, U-99
mode keyword, U-155
model
APIfunctions, U-104
BirdCarreau, U-106
CrossPowerLaw, U-106
DeardorffDiffStress, U-105, U-106
GuldersEGRLaminarFlameSpeed, U-103
GuldersLaminarFlameSpeed, U-103
HerschelBulkley, U-106
LRDDiffStress, U-105
LRR, U-104, U-105
LamBremhorstKE, U-104
LaunderGibsonRSTM, U-104, U-105
LaunderSharmaKE, U-104, U-105
LienCubicKELowRe, U-104
LienCubicKE, U-104
LienLeschzinerLowRe, U-104
NSRDSfunctions, U-104
Newtonian, U-106
NonlinearKEShih, U-104
PrandtlDelta, U-105
RNGkEpsilon, U-104, U-105
Smagorinsky2, U-105
Smagorinsky, U-105, U-106
SpalartAllmarasDDES, U-105
SpalartAllmarasIDDES, U-105
SpalartAllmaras, U-105, U-106
anisotropicFilter, U-105
basicMultiComponentMixture, U-103, U-188
chemistryModel, U-104
chemistrySolver, U-104
constLaminarFlameSpeed, U-103
constTransport, U-103, U-188
cubeRootVolDelta, U-105
dieselMixture, U-103, U-188
distributed, U-102
dynLagrangian, U-105
dynMixedSmagorinsky, U-105
dynOneEqEddy, U-105, U-106
dynSmagorinsky, U-105
eConstThermo, U-103, U-187
ePsiThermo, U-102, U-188
egrMixture, U-103, U-188
hConstThermo, U-103, U-187
hPolynomialThermo, U-103, U-187

hPsiMixtureThermo, U-102, U-188
hPsiThermo, U-102, U-188
hRhoMixtureThermo, U-102, U-189
hRhoThermo, U-102, U-188
hhuMixtureThermo, U-102, U-189
homogenousDynSmagorinsky, U-105
homogeneousMixture, U-103, U-188
hsPsiMixtureThermo, U-102, U-188
hsPsiThermo, U-102, U-188
hsRhoMixtureThermo, U-102, U-189
icoPolynomial, U-103, U-187
inhomogeneousMixture, U-103, U-188
interfaceProperties, U-106
janafThermo, U-103, U-188
kEpsilon, U-104, U-105
kOmegaSSTSAS, U-105
kOmegaSST, U-104, U-105
kOmega, U-104
laminar, U-104, U-105
laplaceFilter, U-105
locDynOneEqEddy, U-105
lowReOneEqEddy, U-106
maxDeltaxyz, U-105
mixedSmagorinsky, U-105
multiComponentMixture, U-103, U-188
oneEqEddy, U-105, U-106
perfectGas, U-103, U-187
polynomialTransport, U-104, U-188
powerLaw, U-106
ptsotchDecomp, U-102
pureMixture, U-102, U-188
qZeta, U-104
reactingMixture, U-103, U-188
realizableKE, U-104, U-105
reconstruct, U-102
scaleSimilarity, U-105
scotchDecomp, U-102
simpleFilter, U-105
smoothDelta, U-105
specieThermo, U-103, U-188
spectEddyVisc, U-105
sutherlandTransport, U-104, U-188
veryInhomogeneousMixture, U-103, U-188
modifyMesh utility, U-95
molecularMeasurements
library, U-101
molecule
library, U-101
`molWeight` keyword, U-189
moveDynamicMesh utility, U-94
moveEngineMesh utility, U-94
moveMesh utility, U-94
movingWallVelocity
boundary condition, U-142
MPI
openMPI, U-85
MRFInterFoam solver, U-90
MRFMultiphaseInterFoam solver, U-90
MRFSimpleFoam solver, U-88
mshToFoam utility, U-94
`mu` keyword, U-190
multiComponentMixture model, U-103, U-188
multigrid
geometric-algebraic, U-127
multiphaseInterFoam solver, U-90
`MUSCL`
keyword entry, U-121

N

`n` keyword, U-86
nabla
operator, P-27
`nAlphaSubCycles` keyword, U-65
`nBufferCellsNoExtrude` keyword, U-158
`nCellsBetweenLevels` keyword, U-153
neighbour
dictionary, U-134
`neighbourPatch` keyword, U-147
netgenNeutralToFoam utility, U-94
`Newtonian`
keyword entry, U-62
Newtonian model, U-106
`nextWrite`
keyword entry, U-116
`nFaces` keyword, U-134
`nFinestSweeps` keyword, U-128
`nGrow` keyword, U-158
`nLayerIter` keyword, U-158
`nMoles` keyword, U-189
non-orthogonal mesh, P-45
`nonBlocking`
keyword entry, U-83
`none`
keyword entry, U-119, U-127
NonlinearKEShih model, U-104
nonNewtonianIcoFoam solver, U-88
`noWriteNow`
keyword entry, U-116
`nPostSweeps` keyword, U-128
`nPreSweeps` keyword, U-128
`nRelaxedIter` keyword, U-158

nRelaxIter keyword, U-156, U-158
nSmoothNormals keyword, U-158
nSmoothPatch keyword, U-156
nSmoothScale keyword, U-159
nSmoothSurfaceNormals keyword, U-158
nSmoothThickness keyword, U-158
nSolveIter keyword, U-156
NSRDSfunctions model, U-104
null
    keyword entry, U-181
numberOfSubdomains keyword, U-86

O

object keyword, U-109
objToVTK utility, U-94
ODE
    library, U-101
oneEqEddy model, U-105, U-106
Opacity text box, U-173
OpenFOAM
    applications, U-71
    file format, U-108
    libraries, U-71
OpenFOAM
    library, U-100
OpenFOAM file syntax
    //, U-108
openMPI
    message passing interface, U-85
    MPI, U-85
operator
    scalar, P-28
    vector, P-27
Options window, U-174
options file, U-75
order keyword, U-86
Orientation Axes button, U-27, U-174
OSspecific
    library, U-102
outer product, *see* tensor, outer product
outlet
    boundary condition, P-70
outletInlet
    boundary condition, U-142
outside
    keyword entry, U-155
owner
    dictionary, U-133

P

p field, U-25
P1
    library, U-103
p_rhgRefCell keyword, U-130
p_rhgRefValue keyword, U-130
pairPatchAgglomeration
    library, U-102
paraFoam, U-25, U-169
parallel
    running, U-84
partialSlip
    boundary condition, U-142
particleTracks utility, U-97
patch
    boundary condition, U-139
patch
    keyword entry, U-140, U-182
patchAverage utility, U-97
patches keyword, U-143
patchIntegrate utility, U-97
patchMap keyword, U-166
patchSummary utility, U-99
PBiCG
    keyword entry, U-126
PCG
    keyword entry, U-126
pdfPlot utility, U-97
PDRFoam solver, U-91
PDRMesh utility, U-95
Pe utility, U-96
perfectGas model, U-103, U-187
permutation symbol, P-20
pimpleDyMFoam solver, U-88
pimpleFoam solver, U-88
Pipeline Browser window, U-26, U-170
PISO
    dictionary, U-25
pisoFoam solver, U-19, U-89
Plot Over Line
    menu entry, U-36
plot3dToFoam utility, U-94
pointField class, P-31
pointField<Type> template class, P-33
points
    dictionary, U-133, U-141
polyBoundaryMesh class, P-31
polyDualMesh utility, U-94
polyLine
    keyword entry, U-144
polyMesh directory, U-107, U-133
polyMesh class, P-31, U-131, U-133

polynomialTransport model, U-104, U-188
polyPatch class, P-31
polyPatchList class, P-31
polySpline
keyword entry, U-144
porousExplicitSourceReactingParcelFoam solver, U-91
porousInterFoam solver, U-90
porousSimpleFoam solver, U-89
post-processing, U-169
post-processing
paraFoam, U-169
postCalc
library, U-101
postChannel utility, U-97
potential
library, U-101
potentialFoam solver, P-46, U-88
pow
tensor member function, P-25
powerLaw model, U-106
pPrime2 utility, U-96
Pr keyword, U-190
PrandtlDelta model, U-105
preconditioner keyword, U-126, U-127
pRefCell keyword, U-25, U-130
pRefValue keyword, U-25, U-130
pressure keyword, U-53
pressure waves
in liquids, P-63
pressureDirectedInletVelocity
boundary condition, U-142
pressureInletVelocity
boundary condition, U-142
pressureOutlet
boundary condition, P-64
pressureTransmissive
boundary condition, U-142
primitive
library, P-23
primitives tools, U-100
printCeoffs keyword, U-192
printCoeffs keyword, U-43
processorWeights keyword, U-85
probeLocations utility, U-97
process
background, U-27, U-84
foreground, U-27
processor
boundary condition, U-140
processor
keyword entry, U-140
processorN directory, U-85
processorWeights keyword, U-86
Properties window panel, U-27, U-170, U-171
ptot utility, U-97
ptsotchDecomp model, U-102
pureMixture model, U-102, U-188
purgeWrite keyword, U-116
PV3FoamReader
library, U-169
PVFoamReader
library, U-169

Q

Q utility, U-96
QUICK
keyword entry, U-124
qZeta model, U-104

R

R utility, U-97
radiationModels
library, U-103
randomProcesses
library, U-102
RASModel
keyword entry, U-43, U-191
RASModel keyword, U-192
raw
keyword entry, U-117, U-181
reactingFoam solver, U-91
reactingMixture model, U-103, U-188
reactingParcelFilmFoam solver, U-92
reactingParcelFoam solver, U-92
reactionThermophysicalModels
library, U-102
realizableKE model, U-104, U-105
reconstruct model, U-102
reconstructPar utility, U-87, U-99
reconstructParMesh utility, U-99
redistributeMeshPar utility, U-99
refGradient keyword, U-141
refineHexMesh utility, U-95
refinementRegions keyword, U-155
refinementLevel utility, U-95
refinementRegions keyword, U-153, U-155
refinementSurfaces keyword, U-153, U-154
refineMesh utility, U-95
refineWallLayer utility, U-95
Refresh Times button, U-27

regions keyword, U-62
relative tolerance, U-126
relativeSizes keyword, U-158
relaxed keyword, U-159
relTol keyword, U-55, U-126
removeFaces utility, U-95
Render View window, U-174
Render View window panel, U-174
renumberMesh utility, U-95
Rescale to Data Range button, U-27
Reset button, U-170
resolveFeatureAngle keyword, U-153, U-154
restart, U-40
Reynolds number, U-19, U-23
rhoPorousMRFLTSPimpleFoam solver, U-89
rhoPorousMRFPimpleFoam solver, U-89
rhoPorousMRFSimpleFoam solver, U-89
rhoCentralDyMFoam solver, U-89
rhoCentralFoam solver, U-89
rhoPimpleFoam solver, U-89
rhoReactingFoam solver, U-91
rhoSimpleFoam solver, U-89
rhoSimplecFoam solver, U-89
rmdepall script/alias, U-79
RNGkEpsilon model, U-104, U-105
roots keyword, U-86, U-87
rotateMesh utility, U-95
run
    parallel, U-84
run directory, U-107
runTime
    keyword entry, U-34, U-116
runTimeModifiable keyword, U-117

S

sammToFoam utility, U-94
sample utility, U-97, U-180
sampling
    library, U-101
Save Animation
    menu entry, U-175
Save Screenshot
    menu entry, U-175
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-163
scaleSimilarity model, U-105
scheduled
    keyword entry, U-83
scientific
    keyword entry, U-116
scotch
    keyword entry, U-85, U-86
scotchCoeffs keyword, U-86
scotchDecomp model, U-102
script/alias
    foamCorrectVrt, U-164
    foamJob, U-184
    foamLog, U-184
    make, U-73
    rmdepall, U-79
    wclean, U-78
    wmake, U-73
second time derivative, P-37
Seed window, U-175
selectCells utility, U-95
Set Ambient Color button, U-173
setFields utility, U-61, U-62, U-93
setFormat keyword, U-181
sets keyword, U-181
setSet utility, U-95
setsToZones utility, U-95
Settings
    menu entry, U-174
settlingFoam solver, U-90
SFCD
    keyword entry, U-121, U-124
shallowWaterFoam solver, U-89
shape, U-145
Show Color Legend
    menu entry, U-29
SI units, U-112
simple
    keyword entry, U-85, U-86
simpleFilter model, U-105
simpleFoam solver, P-54, U-89
simpleGrading keyword, U-145
simpleSpline
    keyword entry, U-144
simulationType keyword, U-43, U-63, U-191
singleCellMesh utility, U-95
skew
    tensor member function, P-25
skewLinear
    keyword entry, U-121, U-124
SLGThermo

library, U-104
slice class, P-31
slip
boundary condition, U-142
Smagorinsky model, U-105, U-106
Smagorinsky2 model, U-105
smapToFoam utility, U-96
smoothDelta model, U-105
`smoother` keyword, U-128
`smoothSolver`
keyword entry, U-126
`snap` keyword, U-151
`snapControls` keyword, U-151
snappyHexMesh utility
background mesh, U-152
cell removal, U-154
cell splitting, U-153
mesh layers, U-156
meshing process, U-151
snapping to surfaces, U-156
snappyHexMesh utility, U-93, U-150
snappyHexMeshDict file, U-151
`snGrad`
fvc member function, P-37
`snGradCorrection`
fvc member function, P-37
`snGradSchemes` keyword, U-118
solid
library, U-104
`Solid Color`
menu entry, U-173
solidDisplacementFoam solver, U-92
solidDisplacementFoam solver, U-53
solidEquilibriumDisplacementFoam solver, U-92
solidMixtureProperties
library, U-104
solidParticle
library, U-101
solidProperties
library, U-104
solver
LTSInterFoam, U-90
LTSReactingParcelFoam, U-91
MRFInterFoam, U-90
MRFMultiphaseInterFoam, U-90
MRFSimpleFoam, U-88
PDRFoam, U-91
SRFSimpleFoam, U-89
XiFoam, U-91
adjointShapeOptimizationFoam, U-88
blockMesh, P-47
boundaryFoam, U-88
bubbleFoam, U-89
buoyantBaffleSimpleFoam, U-91
buoyantBoussinesqPimpleFoam, U-91
buoyantBoussinesqSimpleFoam, U-91
buoyantPimpleFoam, U-91
buoyantSimpleFoam, U-91
buoyantSimpleRadiationFoam, U-91
cavitatingFoam, U-89
channelFoam, U-88
chemFoam, U-90
chtMultiRegionFoam, U-91
coalChemistryFoam, U-91
coldEngineFoam, U-90
compressibleInterFoam, U-90
dieselEngineFoam, U-91
dieselFoam, U-91
dnsFoam, U-90
dsmcFoam, U-92
electrostaticFoam, U-92
engineFoam, U-91
financialFoam, U-92
fireFoam, U-91
icoFoam, U-19, U-23, U-24, U-27, U-88
icoUncoupledKinematicParcelDyMFoam, U-91
icoUncoupledKinematicParcelFoam, U-91
interDyMFoam, U-90
interFoam, U-90
interMixingFoam, U-90
interPhaseChangeFoam, U-90
laplacianFoam, U-88
magneticFoam, U-92
mdEquilibrationFoam, U-92
mdFoam, U-92
mhdFoam, P-70, U-92
multiphaseInterFoam, U-90
nonNewtonianIcoFoam, U-88
pimpleDyMFoam, U-88
pimpleFoam, U-88
pisoFoam, U-19, U-89
porousExplicitSourceReactingParcelFoam, U-91
porousInterFoam, U-90
porousSimpleFoam, U-89
potentialFoam, P-46, U-88
reactingFoam, U-91
reactingParcelFilmFoam, U-92
reactingParcelFoam, U-92

rhoCentralDyMFoam, U-89
rhoCentralFoam, U-89
rhoPimpleFoam, U-89
rhoReactingFoam, U-91
rhoSimpleFoam, U-89
rhoSimplecFoam, U-89
rhoPorousMRFLTSPimpleFoam, U-89
rhoPorousMRFPimpleFoam, U-89
rhoPorousMRFSimpleFoam, U-89
scalarTransportFoam, U-88
settlingFoam, U-90
shallowWaterFoam, U-89
simpleFoam, P-54, U-89
solidDisplacementFoam, U-92
solidDisplacementFoam, U-53
solidEquilibriumDisplacementFoam, U-92
sonicDyMFoam, U-89
sonicFoam, P-60, U-89
sonicLiquidFoam, P-64, U-89
twoLiquidMixingFoam, U-90
twoPhaseEulerFoam, U-90
uncoupledKinematicParcelFoam, U-92
windSimpleFoam, U-89
`solver` keyword, U-55, U-125
solver relative tolerance, U-126
solver tolerance, U-126
`solvers` keyword, U-125
sonicDyMFoam solver, U-89
sonicFoam solver, P-60, U-89
sonicLiquidFoam solver, P-64, U-89
source, P-37
SpalartAllmaras model, U-105, U-106
SpalartAllmarasDDES model, U-105
SpalartAllmarasIDDES model, U-105
specie
library, U-103
`specie` keyword, U-189
specieThermo model, U-103, U-188
spectEddyVisc model, U-105
`spline` keyword, U-143
splitCells utility, U-95
splitMesh utility, U-95
splitMeshRegions utility, U-95
`sqr`
tensor member function, P-25
`sqrGradGrad`
fvc member function, P-37
SRFSimpleFoam solver, U-89
star3ToFoam utility, U-94
star4ToFoam utility, U-94
`startFace` keyword, U-134
`startFrom` keyword, U-24, U-115
starToFoam utility, U-159
`startTime`
keyword entry, U-24, U-115
`startTime` keyword, U-24, U-115
steady flow
turbulent, P-53
steadyParticleTracks utility, U-97
`steadyState`
keyword entry, U-124
Stereolithography (STL), U-150
stitchMesh utility, U-95
`stl`
keyword entry, U-181
`stopAt` keyword, U-115
`strategy` keyword, U-85, U-86
streamFunction utility, U-96
stress analysis of plate with hole, U-48
stressComponents utility, U-96
Style window panel, U-26, U-173
`Su`
fvm member function, P-37
subsetMesh utility, U-95
summation convention, P-17
SUPERBEE differencing, P-38
supersonic flow, P-59
supersonic flow over forward step, P-59
supersonicFreeStream
boundary condition, U-142
surface mesh, U-150
surfaceAdd utility, U-98
surfaceAutoPatch utility, U-98
surfaceCheck utility, U-98
surfaceClean utility, U-98
surfaceCoarsen utility, U-98
surfaceConvert utility, U-98
surfaceFeatureConvert utility, U-98
surfaceFeatureExtract utility, U-98, U-154
surfaceField<Type> template class, P-33
surfaceFilmModels
library, U-106
surfaceFind utility, U-98
`surfaceFormat` keyword, U-181
surfaceInertia utility, U-98
surfaceMesh tools, U-100
surfaceMeshConvert utility, U-98
surfaceMeshConvertTesting utility, U-98
surfaceMeshExport utility, U-98
surfaceMeshImport utility, U-98

surfaceMeshInfo utility, U-98
surfaceMeshTriangulate utility, U-98
surfaceNormalFixedValue
boundary condition, U-142
surfaceOrient utility, U-98
surfacePointMerge utility, U-98
surfaceRedistributePar utility, U-98
surfaceRefineRedGreen utility, U-98
surfaces keyword, U-181
surfaceSmooth utility, U-98
surfaceSplitByPatch utility, U-98
surfaceSplitNonManifolds utility, U-99
surfaceSubset utility, U-99
surfaceToPatch utility, U-99
surfaceTransformPoints utility, U-99
surfMesh
library, U-101
SuSp
fvm member function, P-37
sutherlandTransport model, U-104, U-188
symm
tensor member function, P-25
symmetryPlane
boundary condition, P-64, U-140
symmetryPlane
keyword entry, U-140
symmTensorField class, P-29
symmTensorThirdField class, P-29
system directory, P-50, U-107
systemCall
library, U-101

T

T()
tensor member function, P-25
Tcommon keyword, U-190
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
nth 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-94
text box
Opacity, U-173
thermalPorousZone
library, U-104
thermalProperties
dictionary, U-53
thermodynamics keyword, U-189
thermophysical
library, U-187
thermophysicalFunctions
library, U-104
thermophysicalProperties
dictionary, U-187
thermoType keyword, U-187
Thigh keyword, U-190
time
control, U-115
time derivative, P-37
first, P-39
second, P-37, P-39
time step, U-24
timeFormat keyword, U-116
timePrecision keyword, U-117
timeScheme keyword, U-118
timeStamp
keyword entry, U-83
timeStampMaster
keyword entry, U-83
timeStep
keyword entry, U-24, U-34, U-116
Tlow keyword, U-190
tolerance
solver, U-126
solver relative, U-126
tolerance keyword, U-55, U-126, U-156
Toolbars
menu entry, U-173
tools
algorithms, U-100
cfdTools, U-100
containers, U-100
db, U-100
dimensionSet, U-100
dimensionedTypes, U-100
fields, U-100
finiteVolume, U-100
fvMatrices, U-100
fvMesh, U-100
global, U-100
graph, U-100
interpolations, U-100
interpolation, U-100
matrices, U-100
memory, U-100
meshes, U-100
primitives, U-100
surfaceMesh, U-100
volMesh, U-100
topoChangerFvMesh
library, U-101
topoSet utility, U-95
topoSetSource keyword, U-62
totalPressure
boundary condition, U-142
tr
tensor member function, P-25
trace, *see* tensor, trace
traction keyword, U-53
transform
tensor member function, P-25
transformPoints utility, U-95
transport keyword, U-189

transportProperties
dictionary, U-23, U-40, U-44
transportProperties file, U-62
triple inner product, P-19
triSurface
library, U-101
`Ts` keyword, U-190
turbulence
dissipation, U-42
kinetic energy, U-42
length scale, U-43
`turbulence` keyword, U-192
turbulence model
RAS, U-42
turbulenceProperties
dictionary, U-43, U-63, U-191
turbulent flow
steady, P-53
turbulentInlet
boundary condition, U-142
tutorials
breaking of a dam, U-58
lid-driven cavity flow, U-19
stress analysis of plate with hole, U-48
tutorials directory, P-45, U-19
twoLiquidMixingFoam solver, U-90
twoPhaseEulerFoam solver, U-90
twoPhaseInterfaceProperties
library, U-106
`type` keyword, U-135, U-138

U

`U` field, U-25
Ucomponents utility, P-71
`UMIST`
keyword entry, U-119
`uncompressed`
keyword entry, U-116
`uncorrected`
keyword entry, U-122, U-123
uncoupledKinematicParcelFoam solver, U-92
`uniform` keyword, U-182
units
base, U-112
of measurement, P-26, U-111
S.I. base, P-26
SI, U-112
Système International, U-112
United States Customary System, U-112
USCS, U-112
Update GUI button, U-171
uprime utility, U-96
`upwind`
keyword entry, U-121, U-124
upwind differencing, P-38, U-64
USCS units, U-112
Use Parallel Projection button, U-27
Use parallel projection button, U-173
utility
Co, U-96
Lambda2, U-96
Mach, U-96
PDRMesh, U-95
Pe, U-96
Q, U-96
R, U-97
Ucomponents, P-71
adiabaticFlameT, U-99
ansysToFoam, U-93
applyBoundaryLayer, U-93
applyWallFunctionBoundaryConditions, U-93
attachMesh, U-94
autoPatch, U-94
autoRefineMesh, U-95
blockMesh, U-39, U-93, U-141
boxTurb, U-93
cfx4ToFoam, U-93, U-159
changeDictionary, U-93
checkMesh, U-94, U-161
chemkinToFoam, U-99
collapseEdges, U-95
combinePatchFaces, U-95
createBaffles, U-94
createPatch, U-94
createTurbulenceFields, U-97
datToFoam, U-93
decomposePar, U-84, U-85, U-99
deformedGeom, U-94
dsmcFieldsCalc, U-97
dsmcInitialise, U-93
engineCompRatio, U-97
engineSwirl, U-93
ensight74FoamExec, U-179
ensightFoamReader, U-95
enstrophy, U-96
equilibriumCO, U-99
equilibriumFlameT, U-99
execFlowFunctionObjects, U-97
expandDictionary, U-99
extrude2DMesh, U-93
extrudeMesh, U-93

extrudeToRegionMesh, U-93
faceAgglomerate, U-93
fieldview9Reader, U-95
flattenMesh, U-94
flowType, U-96
fluent3DMeshToFoam, U-93
fluentMeshToFoam, U-93, U-159
foamCalc, U-34, U-97
foamDataToFluent, U-96, U-176
foamDebugSwitches, U-99
foamFormatConvert, U-99
foamInfoExec, U-99
foamListTimes, U-97
foamMeshToFluent, U-94, U-176
foamToEnsightParts, U-96
foamToEnsight, U-96
foamToFieldview9, U-96
foamToGMV, U-96
foamToStarMesh, U-94
foamToSurface, U-94
foamToTecplot360, U-96
foamToVTK, U-96
foamUpgradeCyclics, U-93
foamUpgradeFvSolution, U-93
gambitToFoam, U-94, U-159
gmshToFoam, U-94
ideasToFoam, U-159
ideasUnvToFoam, U-94
insideCells, U-94
kivaToFoam, U-94
mapFields, U-33, U-40, U-44, U-57, U-93, U-166
mdInitialise, U-93
mergeMeshes, U-94
mergeOrSplitBaffles, U-94
mirrorMesh, U-94
mixtureAdiabaticFlameT, U-99
modifyMesh, U-95
moveDynamicMesh, U-94
moveEngineMesh, U-94
moveMesh, U-94
mshToFoam, U-94
netgenNeutralToFoam, U-94
objToVTK, U-94
pPrime2, U-96
particleTracks, U-97
patchAverage, U-97
patchIntegrate, U-97
patchSummary, U-99
pdfPlot, U-97
plot3dToFoam, U-94
polyDualMesh, U-94
postChannel, U-97
probeLocations, U-97
ptot, U-97
reconstructParMesh, U-99
reconstructPar, U-87, U-99
redistributeMeshPar, U-99
refineHexMesh, U-95
refineMesh, U-95
refineWallLayer, U-95
refinementLevel, U-95
removeFaces, U-95
renumberMesh, U-95
rotateMesh, U-95
sammToFoam, U-94
sample, U-97, U-180
scalePoints, U-163
selectCells, U-95
setFields, U-61, U-62, U-93
setSet, U-95
setsToZones, U-95
singleCellMesh, U-95
smapToFoam, U-96
snappyHexMesh, U-93, U-150
splitCells, U-95
splitMeshRegions, U-95
splitMesh, U-95
star3ToFoam, U-94
star4ToFoam, U-94
starToFoam, U-159
steadyParticleTracks, U-97
stitchMesh, U-95
streamFunction, U-96
stressComponents, U-96
subsetMesh, U-95
surfaceAdd, U-98
surfaceAutoPatch, U-98
surfaceCheck, U-98
surfaceClean, U-98
surfaceCoarsen, U-98
surfaceConvert, U-98
surfaceFeatureConvert, U-98
surfaceFeatureExtract, U-98, U-154
surfaceFind, U-98
surfaceInertia, U-98
surfaceMeshConvertTesting, U-98
surfaceMeshConvert, U-98
surfaceMeshExport, U-98
surfaceMeshImport, U-98

surfaceMeshInfo, U-98
surfaceMeshTriangulate, U-98
surfaceOrient, U-98
surfacePointMerge, U-98
surfaceRedistributePar, U-98
surfaceRefineRedGreen, U-98
surfaceSmooth, U-98
surfaceSplitByPatch, U-98
surfaceSplitNonManifolds, U-99
surfaceSubset, U-99
surfaceToPatch, U-99
surfaceTransformPoints, U-99
tetgenToFoam, U-94
topoSet, U-95
transformPoints, U-95
uprime, U-96
viewFactorGen, U-93
vorticity, U-96
wallFunctionTable, U-93
wallGradU, U-96
wallHeatFlux, U-96
wallShearStress, U-96
wdot, U-97
writeCellCentres, U-97
writeMeshObj, U-94
yPlusLES, U-97
yPlusRAS, U-97
zipUpMesh, U-95
utilityFunctionObjects
library, U-101

V

value keyword, U-23, U-141
valueFraction keyword, U-141
van Leer differencing, P-38
vanLeer
keyword entry, U-121
VCR Controls menu, U-27, U-171
vector, P-16
operator, P-27
unit, P-20
vector class, P-24, U-111
vector product, *see* tensor, vector cross product
vectorField class, P-29
version keyword, U-109
vertices keyword, U-22, U-143, U-144
veryInhomogeneousMixture model, U-103, U-188
View menu, U-173
View Settings
menu entry, U-27, U-173
View Settings (Render View) window, U-173
View Settings...
menu entry, U-26
viewFactor
library, U-103
viewFactorGen utility, U-93
viscosity
kinematic, U-23, U-44
volField<Type> template class, P-33
volMesh tools, U-100
vorticity utility, U-96
vtk
keyword entry, U-181
vtkFoam
library, U-169
vtkPV3Foam
library, U-169

W

wall
boundary condition, P-64, P-70, U-60, U-139
wall
keyword entry, U-140
wallBuoyantPressure
boundary condition, U-142
wallFunctionTable utility, U-93
wallGradU utility, U-96
wallHeatFlux utility, U-96
Wallis
library, U-103
wallShearStress utility, U-96
wclean script/alias, U-78
wdot utility, U-97
wedge
boundary condition, U-135, U-140, U-149
wedge
keyword entry, U-140
window
Color Legend, U-29
Options, U-174
Pipeline Browser, U-26, U-170
Render View, U-174
Seed, U-175
View Settings (Render View), U-173
window panel
Animations, U-174
Annotation, U-27, U-174
Charts, U-174
Color Legend, U-172
Color Scale, U-172
Colors, U-174
Display, U-26, U-27, U-170, U-171

General, U-173, U-174
Information, U-170
Lights, U-174
Mesh Parts, U-26
Properties, U-27, U-170, U-171
Render View, U-174
Style, U-26, U-173
windSimpleFoam solver, U-89
`Wireframe`
menu entry, U-173
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_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_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-97
`writeCompression` keyword, U-116
`writeControl`
keyword entry, U-116
`writeControl` keyword, U-24, U-64, U-116
`writeFormat` keyword, U-57, U-116
`writeInterval` keyword, U-24, U-34, U-116
writeMeshObj utility, U-94
`writeNow`
keyword entry, U-115
`writePrecision` keyword, U-116

X

`x`
keyword entry, U-182
XiFoam solver, U-91
`xmgr`
keyword entry, U-117, U-181
`xyz`
keyword entry, U-182

Y

`y`
keyword entry, U-182
yPlusLES utility, U-97
yPlusRAS utility, U-97

Z

`z`
keyword entry, U-182
zeroGradient
boundary condition, U-141
zipUpMesh utility, U-95