OpenFOAM

*The Open Source CFD Toolbox*

# Programmer's Guide

Version 2.4.0
21st May 2015

Copyright © 2011-2015 OpenFOAM Foundation Ltd.
Author: Christopher J. Greenshields, CFD Direct Ltd.

This work is licensed under a
**Creative Commons Attribution-NonCommercial-NoDerivs 3.0 Unported License**.

Typeset in LaTeX.

### License

THE WORK (AS DEFINED BELOW) IS PROVIDED UNDER THE TERMS OF THIS CREATIVE COMMONS PUBLIC LICENSE ("CCPL" OR "LICENSE"). THE WORK IS PROTECTED BY COPYRIGHT AND/OR OTHER APPLICABLE LAW. ANY USE OF THE WORK OTHER THAN AS AUTHORIZED UNDER THIS LICENSE OR COPYRIGHT LAW IS PROHIBITED.

BY EXERCISING ANY RIGHTS TO THE WORK PROVIDED HERE, YOU ACCEPT AND AGREE TO BE BOUND BY THE TERMS OF THIS LICENSE. TO THE EXTENT THIS LICENSE MAY BE CONSIDERED TO BE A CONTRACT, THE LICENSOR GRANTS YOU THE RIGHTS CONTAINED HERE IN CONSIDERATION OF YOUR ACCEPTANCE OF SUCH TERMS AND CONDITIONS.

### 1. Definitions

a. "Adaptation" means a work based upon the Work, or upon the Work and other pre-existing works, such as a translation, adaptation, derivative work, arrangement of music or other alterations of a literary or artistic work, or phonogram or performance and includes cinematographic adaptations or any other form in which the Work may be recast, transformed, or adapted including in any form recognizably derived from the original, except that a work that constitutes a Collection will not be considered an Adaptation for the purpose of this License. For the avoidance of doubt, where the Work is a musical work, performance or phonogram, the synchronization of the Work in timed-relation with a moving image ("synching") will be considered an Adaptation for the purpose of this License.

b. "Collection" means a collection of literary or artistic works, such as encyclopedias and anthologies, or performances, phonograms or broadcasts, or other works or subject matter other than works listed in Section 1(f) below, which, by reason of the selection and arrangement of their contents, constitute intellectual creations, in which the Work is included in its entirety in unmodified form along with one or more other contributions, each constituting separate and independent works in themselves, which together are assembled into a collective whole. A work that constitutes a Collection will not be considered an Adaptation (as defined above) for the purposes of this License.

c. "Distribute" means to make available to the public the original and copies of the Work through sale or other transfer of ownership.

d. "Licensor" means the individual, individuals, entity or entities that offer(s) the Work under the terms of this License.

e. "Original Author" means, in the case of a literary or artistic work, the individual, individuals, entity or entities who created the Work or if no individual or entity can be identified, the

publisher; and in addition (i) in the case of a performance the actors, singers, musicians, dancers, and other persons who act, sing, deliver, declaim, play in, interpret or otherwise perform literary or artistic works or expressions of folklore; (ii) in the case of a phonogram the producer being the person or legal entity who first fixes the sounds of a performance or other sounds; and, (iii) in the case of broadcasts, the organization that transmits the broadcast.

f. "Work" means the literary and/or artistic work offered under the terms of this License including without limitation any production in the literary, scientific and artistic domain, whatever may be the mode or form of its expression including digital form, such as a book, pamphlet and other writing; a lecture, address, sermon or other work of the same nature; a dramatic or dramatico-musical work; a choreographic work or entertainment in dumb show; a musical composition with or without words; a cinematographic work to which are assimilated works expressed by a process analogous to cinematography; a work of drawing, painting, architecture, sculpture, engraving or lithography; a photographic work to which are assimilated works expressed by a process analogous to photography; a work of applied art; an illustration, map, plan, sketch or three-dimensional work relative to geography, topography, architecture or science; a performance; a broadcast; a phonogram; a compilation of data to the extent it is protected as a copyrightable work; or a work performed by a variety or circus performer to the extent it is not otherwise considered a literary or artistic work.

g. "You" means an individual or entity exercising rights under this License who has not previously violated the terms of this License with respect to the Work, or who has received express permission from the Licensor to exercise rights under this License despite a previous violation.

h. "Publicly Perform" means to perform public recitations of the Work and to communicate to the public those public recitations, by any means or process, including by wire or wireless means or public digital performances; to make available to the public Works in such a way that members of the public may access these Works from a place and at a place individually chosen by them; to perform the Work to the public by any means or process and the communication to the public of the performances of the Work, including by public digital performance; to broadcast and rebroadcast the Work by any means including signs, sounds or images.

i. "Reproduce" means to make copies of the Work by any means including without limitation by sound or visual recordings and the right of fixation and reproducing fixations of the Work, including storage of a protected performance or phonogram in digital form or other electronic medium.

## 2. Fair Dealing Rights.

Nothing in this License is intended to reduce, limit, or restrict any uses free from copyright or rights arising from limitations or exceptions that are provided for in connection with the copyright protection under copyright law or other applicable laws.

## 3. License Grant.

Subject to the terms and conditions of this License, Licensor hereby grants You a worldwide, royalty-free, non-exclusive, perpetual (for the duration of the applicable copyright) license to exercise the rights in the Work as stated below:

a. to Reproduce the Work, to incorporate the Work into one or more Collections, and to Reproduce the Work as incorporated in the Collections;

b. and, to Distribute and Publicly Perform the Work including as incorporated in Collections.

The above rights may be exercised in all media and formats whether now known or hereafter devised. The above rights include the right to make such modifications as are technically necessary to exercise the rights in other media and formats, but otherwise you have no rights to make Adaptations. Subject to 8(f), all rights not expressly granted by Licensor are hereby reserved, including but not limited to the rights set forth in Section 4(d).

## 4. Restrictions.

The license granted in Section 3 above is expressly made subject to and limited by the following restrictions:

a. You may Distribute or Publicly Perform the Work only under the terms of this License. You must include a copy of, or the Uniform Resource Identifier (URI) for, this License with every copy of the Work You Distribute or Publicly Perform. You may not offer or impose any terms on the Work that restrict the terms of this License or the ability of the recipient of the Work to exercise the rights granted to that recipient under the terms of the License. You may not sublicense the Work. You must keep intact all notices that refer to this License and to the disclaimer of warranties with every copy of the Work You Distribute or Publicly Perform. When You Distribute or Publicly Perform the Work, You may not impose any effective technological measures on the Work that restrict the ability of a recipient of the Work from You to exercise the rights granted to that recipient under the terms of the License. This Section 4(a) applies to the Work as incorporated in a Collection, but this does not require the Collection apart from the Work itself to be made subject to the terms of this License. If You create a Collection, upon notice from any Licensor You must, to the extent practicable, remove from the Collection any credit as required by Section 4(c), as requested.

b. You may not exercise any of the rights granted to You in Section 3 above in any manner that is primarily intended for or directed toward commercial advantage or private monetary compensation. The exchange of the Work for other copyrighted works by means of digital file-sharing or otherwise shall not be considered to be intended for or directed toward commercial advantage or private monetary compensation, provided there is no payment of any monetary compensation in connection with the exchange of copyrighted works.

c. If You Distribute, or Publicly Perform the Work or Collections, You must, unless a request has been made pursuant to Section 4(a), keep intact all copyright notices for the Work and provide, reasonable to the medium or means You are utilizing: (i) the name of the Original Author (or pseudonym, if applicable) if supplied, and/or if the Original Author and/or Licensor designate another party or parties (e.g., a sponsor institute, publishing entity, journal) for attribution ("Attribution Parties") in Licensor's copyright notice, terms of service or by other reasonable means, the name of such party or parties; (ii) the title of the Work if supplied; (iii) to the extent reasonably practicable, the URI, if any, that Licensor specifies to be associated with the Work, unless such URI does not refer to the copyright notice or licensing information for the Work. The credit required by this Section 4(c) may be implemented in any reasonable manner; provided, however, that in the case of a Collection, at a minimum such credit will appear, if a credit for all contributing authors of Collection appears, then as part of these credits and in a manner at least as prominent as the credits for the other contributing authors. For the avoidance of doubt, You may only use the credit required by this Section for the purpose of attribution in the manner set out above and, by exercising Your rights under this License, You may not implicitly or explicitly assert or imply any connection with, sponsorship or endorsement by the Original Author, Licensor and/or

Attribution Parties, as appropriate, of You or Your use of the Work, without the separate, express prior written permission of the Original Author, Licensor and/or Attribution Parties.

d. For the avoidance of doubt:

   i. **Non-waivable Compulsory License Schemes**. In those jurisdictions in which the right to collect royalties through any statutory or compulsory licensing scheme cannot be waived, the Licensor reserves the exclusive right to collect such royalties for any exercise by You of the rights granted under this License;

   ii. **Waivable Compulsory License Schemes**. In those jurisdictions in which the right to collect royalties through any statutory or compulsory licensing scheme can be waived, the Licensor reserves the exclusive right to collect such royalties for any exercise by You of the rights granted under this License if Your exercise of such rights is for a purpose or use which is otherwise than noncommercial as permitted under Section 4(b) and otherwise waives the right to collect royalties through any statutory or compulsory licensing scheme; and,

   iii. **Voluntary License Schemes**. The Licensor reserves the right to collect royalties, whether individually or, in the event that the Licensor is a member of a collecting society that administers voluntary licensing schemes, via that society, from any exercise by You of the rights granted under this License that is for a purpose or use which is otherwise than noncommercial as permitted under Section 4(b).

e. Except as otherwise agreed in writing by the Licensor or as may be otherwise permitted by applicable law, if You Reproduce, Distribute or Publicly Perform the Work either by itself or as part of any Collections, You must not distort, mutilate, modify or take other derogatory action in relation to the Work which would be prejudicial to the Original Author's honor or reputation.

### 5. Representations, Warranties and Disclaimer

UNLESS OTHERWISE MUTUALLY AGREED BY THE PARTIES IN WRITING, LICENSOR OFFERS THE WORK AS-IS AND MAKES NO REPRESENTATIONS OR WARRANTIES OF ANY KIND CONCERNING THE WORK, EXPRESS, IMPLIED, STATUTORY OR OTHERWISE, INCLUDING, WITHOUT LIMITATION, WARRANTIES OF TITLE, MERCHANTIBILITY, FITNESS FOR A PARTICULAR PURPOSE, NONINFRINGEMENT, OR THE ABSENCE OF LATENT OR OTHER DEFECTS, ACCURACY, OR THE PRESENCE OF ABSENCE OF ERRORS, WHETHER OR NOT DISCOVERABLE. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OF IMPLIED WARRANTIES, SO SUCH EXCLUSION MAY NOT APPLY TO YOU.

### 6. Limitation on Liability.

EXCEPT TO THE EXTENT REQUIRED BY APPLICABLE LAW, IN NO EVENT WILL LICENSOR BE LIABLE TO YOU ON ANY LEGAL THEORY FOR ANY SPECIAL, INCIDENTAL, CONSEQUENTIAL, PUNITIVE OR EXEMPLARY DAMAGES ARISING OUT OF THIS LICENSE OR THE USE OF THE WORK, EVEN IF LICENSOR HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

### 7. Termination

a. This License and the rights granted hereunder will terminate automatically upon any breach by You of the terms of this License. Individuals or entities who have received Collections

from You under this License, however, will not have their licenses terminated provided such individuals or entities remain in full compliance with those licenses. Sections 1, 2, 5, 6, 7, and 8 will survive any termination of this License.

b. Subject to the above terms and conditions, the license granted here is perpetual (for the duration of the applicable copyright in the Work). Notwithstanding the above, Licensor reserves the right to release the Work under different license terms or to stop distributing the Work at any time; provided, however that any such election will not serve to withdraw this License (or any other license that has been, or is required to be, granted under the terms of this License), and this License will continue in full force and effect unless terminated as stated above.

## 8. Miscellaneous

a. Each time You Distribute or Publicly Perform the Work or a Collection, the Licensor offers to the recipient a license to the Work on the same terms and conditions as the license granted to You under this License.

b. If any provision of this License is invalid or unenforceable under applicable law, it shall not affect the validity or enforceability of the remainder of the terms of this License, and without further action by the parties to this agreement, such provision shall be reformed to the minimum extent necessary to make such provision valid and enforceable.

c. No term or provision of this License shall be deemed waived and no breach consented to unless such waiver or consent shall be in writing and signed by the party to be charged with such waiver or consent.

d. This License constitutes the entire agreement between the parties with respect to the Work licensed here. There are no understandings, agreements or representations with respect to the Work not specified here. Licensor shall not be bound by any additional provisions that may appear in any communication from You.

e. This License may not be modified without the mutual written agreement of the Licensor and You. The rights granted under, and the subject matter referenced, in this License were drafted utilizing the terminology of the Berne Convention for the Protection of Literary and Artistic Works (as amended on September 28, 1979), the Rome Convention of 1961, the WIPO Copyright Treaty of 1996, the WIPO Performances and Phonograms Treaty of 1996 and the Universal Copyright Convention (as revised on July 24, 1971). These rights and subject matter take effect in the relevant jurisdiction in which the License terms are sought to be enforced according to the corresponding provisions of the implementation of those treaty provisions in the applicable national law. If the standard suite of rights granted under applicable copyright law includes additional rights not granted under this License, such additional rights are deemed to be included in the License; this License is not intended to restrict the license of any rights under applicable law.

**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 ESI Group
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**
1. Definitions . . . P-2
2. Fair Dealing Rights. . . . P-3
3. License Grant. . . . P-3
4. Restrictions. . . . P-4
5. Representations, Warranties and Disclaimer . . . P-5
6. Limitation on Liability. . . . P-5
7. Termination . . . P-5
8. Miscellaneous . . . P-6

**Trademarks** **P-7**

**Contents** **P-9**

**1 Tensor mathematics** **P-13**
1.1 Coordinate system . . . P-13
1.2 Tensors . . . P-13
1.2.1 Tensor notation . . . P-15
1.3 Algebraic tensor operations . . . P-16
1.3.1 The inner product . . . P-16
1.3.2 The double inner product of two tensors . . . P-17
1.3.3 The triple inner product of two third rank tensors . . . P-17
1.3.4 The outer product . . . P-17
1.3.5 The cross product of two vectors . . . P-18
1.3.6 Other general tensor operations . . . P-18
1.3.7 Geometric transformation and the identity tensor . . . P-19
1.3.8 Useful tensor identities . . . P-19
1.3.9 Operations exclusive to tensors of rank 2 . . . P-20
1.3.10 Operations exclusive to scalars . . . P-21
1.4 OpenFOAM tensor classes . . . P-21
1.4.1 Algebraic tensor operations in OpenFOAM . . . P-22
1.5 Dimensional units . . . P-24

**2 Discretisation procedures** **P-25**
2.1 Differential operators . . . P-25
2.1.1 Gradient . . . P-25
2.1.2 Divergence . . . P-26
2.1.3 Curl . . . P-26

2.1.4 Laplacian . . . . . P-26
2.1.5 Temporal derivative . . . . . P-26
2.2 Overview of discretisation . . . . . P-27
2.2.1 OpenFOAM lists and fields . . . . . P-27
2.3 Discretisation of the solution domain . . . . . P-28
2.3.1 Defining a mesh in OpenFOAM . . . . . P-29
2.3.2 Defining a geometricField in OpenFOAM . . . . . P-30
2.4 Equation discretisation . . . . . P-31
2.4.1 The Laplacian term . . . . . P-36
2.4.2 The convection term . . . . . P-36
2.4.3 First time derivative . . . . . P-37
2.4.4 Second time derivative . . . . . P-37
2.4.5 Divergence . . . . . P-37
2.4.6 Gradient . . . . . P-38
2.4.7 Grad-grad squared . . . . . P-39
2.4.8 Curl . . . . . P-39
2.4.9 Source terms . . . . . P-39
2.4.10 Other explicit discretisation schemes . . . . . P-39
2.5 Temporal discretisation . . . . . P-40
2.5.1 Treatment of temporal discretisation in OpenFOAM . . . . . P-41
2.6 Boundary Conditions . . . . . P-41
2.6.1 Physical boundary conditions . . . . . P-42

**3 Examples of the use of OpenFOAM P-43**
3.1 Flow around a cylinder . . . . . P-43
3.1.1 Problem specification . . . . . P-44
3.1.2 Note on potentialFoam . . . . . P-45
3.1.3 Mesh generation . . . . . P-45
3.1.4 Boundary conditions and initial fields . . . . . P-47
3.1.5 Running the case . . . . . P-48
3.2 Steady turbulent flow over a backward-facing step . . . . . P-50
3.2.1 Problem specification . . . . . P-50
3.2.2 Mesh generation . . . . . P-53
3.2.3 Boundary conditions and initial fields . . . . . P-56
3.2.4 Case control . . . . . P-57
3.2.5 Running the case and post-processing . . . . . P-57
3.3 Supersonic flow over a forward-facing step . . . . . P-58
3.3.1 Problem specification . . . . . P-58
3.3.2 Mesh generation . . . . . P-59
3.3.3 Running the case . . . . . P-61
3.3.4 Exercise . . . . . P-61
3.4 Decompression of a tank internally pressurised with water . . . . . P-61
3.4.1 Problem specification . . . . . P-62
3.4.2 Mesh Generation . . . . . P-63
3.4.3 Preparing the Run . . . . . P-65
3.4.4 Running the case . . . . . P-66
3.4.5 Improving the solution by refining the mesh . . . . . P-67
3.5 Magnetohydrodynamic flow of a liquid . . . . . P-67

3.5.1 Problem specification . . . . . P-67
3.5.2 Mesh generation . . . . . P-69
3.5.3 Running the case . . . . . P-70

**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.

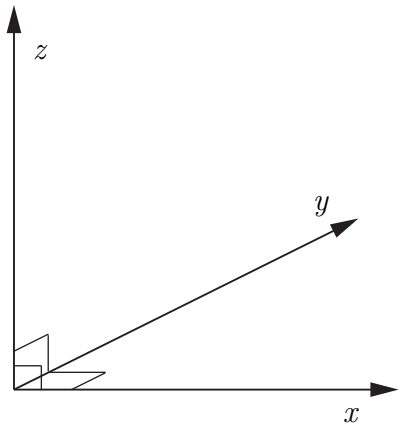



Figure 1.1: Right handed axes

## 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 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 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 $\mu$.

**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 $s = \mathbf{a} \bullet \mathbf{b}$ 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 $\mathbf{b} = \mathbf{T} \bullet \mathbf{a}$, 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 $\mathbf{b} = \mathbf{a} \bullet \mathbf{T} = \mathbf{T}^{\mathrm{T}} \bullet \mathbf{a}$ 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 $\mathbf{P} = \mathbf{T} \bullet \mathbf{S}$ whose components are evaluated as:

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

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

- 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 $r$th 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 $\delta_{ij}$ 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 $2 \times 2$ *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 | | $\mathbf{T}^{\mathrm{T}}$ | `T.T()` |
| Diagonal | | $\mathrm{diag}\,\mathbf{T}$ | `diag(T)` |
| Trace | | $\mathrm{tr}\,\mathbf{T}$ | `tr(T)` |
| Deviatoric component | | $\mathrm{dev}\,\mathbf{T}$ | `dev(T)` |
| Symmetric component | | $\mathrm{symm}\,\mathbf{T}$ | `symm(T)` |
| Skew-symmetric component | | $\mathrm{skew}\,\mathbf{T}$ | `skew(T)` |
| Determinant | | $\det\mathbf{T}$ | `det(T)` |
| Cofactors | | $\mathrm{cof}\,\mathbf{T}$ | `cof(T)` |
| Inverse | | $\mathrm{inv}\,\mathbf{T}$ | `inv(T)` |
| Hodge dual | | $*\,\mathbf{T}$ | `*T` |
| **Operations exclusive to scalars** | | | |
| Sign (boolean) | | $\mathrm{sgn}(s)$ | `sign(s)` |
| Positive (boolean) | | $s >= 0$ | `pos(s)` |
| Negative (boolean) | | $s < 0$ | `neg(s)` |
| Limit | $n$ scalar | $\mathrm{limit}(s, n)$ | `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 | | $\mathrm{asin}\, s$ | `asin(s)` |
| Arc cosine | | $\mathrm{acos}\, s$ | `acos(s)` |
| Arc tangent | | $\mathrm{atan}\, s$ | `atan(s)` |
| Hyperbolic sine | | $\sinh s$ | `sinh(s)` |
| Hyperbolic cosine | | $\cosh s$ | `cosh(s)` |
| Hyperbolic tangent | | $\tanh s$ | `tanh(s)` |
| Hyperbolic arc sine | | $\mathrm{asinh}\, s$ | `asinh(s)` |
| Hyperbolic arc cosine | | $\mathrm{acosh}\, s$ | `acosh(s)` |
| Hyperbolic arc tangent | | $\mathrm{atanh}\, s$ | `atanh(s)` |
| Error function | | $\mathrm{erf}\, s$ | `erf(s)` |
| Complement error function | | $\mathrm{erfc}\, s$ | `erfc(s)` |
| Logarithm gamma function | | $\ln \Gamma s$ | `lgamma(s)` |
| Type 1 Bessel function of order 0 | | $\mathrm{J}_0\, s$ | `j0(s)` |
| Type 1 Bessel function of order 1 | | $\mathrm{J}_1\, s$ | `j1(s)` |
| Type 2 Bessel function of order 0 | | $\mathrm{Y}_0\, s$ | `y0(s)` |
| Type 2 Bessel function of order 1 | | $\mathrm{Y}_1\, s$ | `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* $\nabla$, represented in index notation as $\partial_i$:

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

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 $s$, $\nabla s$ 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)

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

### 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 $\phi$ 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 $\phi$ 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 $\phi$. The second term on the right is known as the convective rate of change of $\phi$.

## 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 $\Delta t$ that may change during a numerical simulation, perhaps depending on some condition calculated during the simulation.

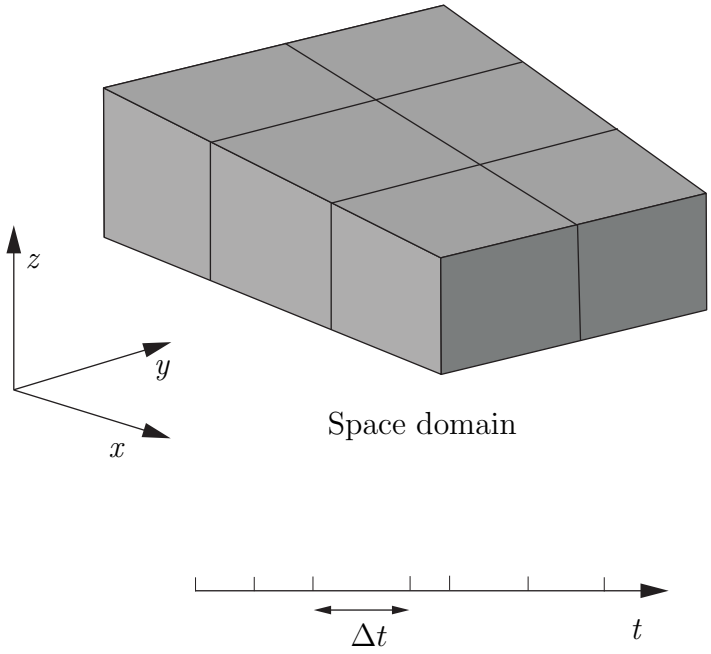



Figure 2.1: Discretisation of the solution domain

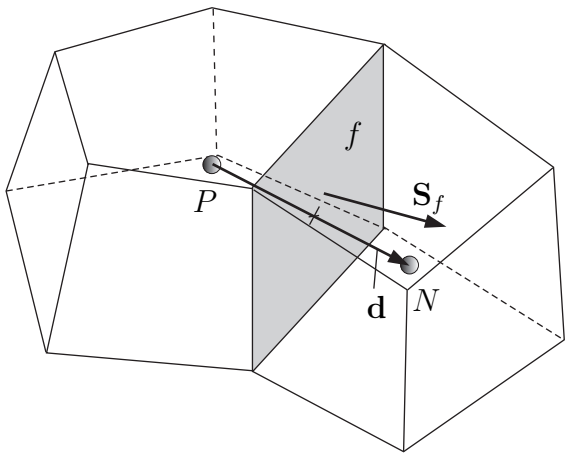



Figure 2.2: Parameters in finite volume discretisation

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.*.

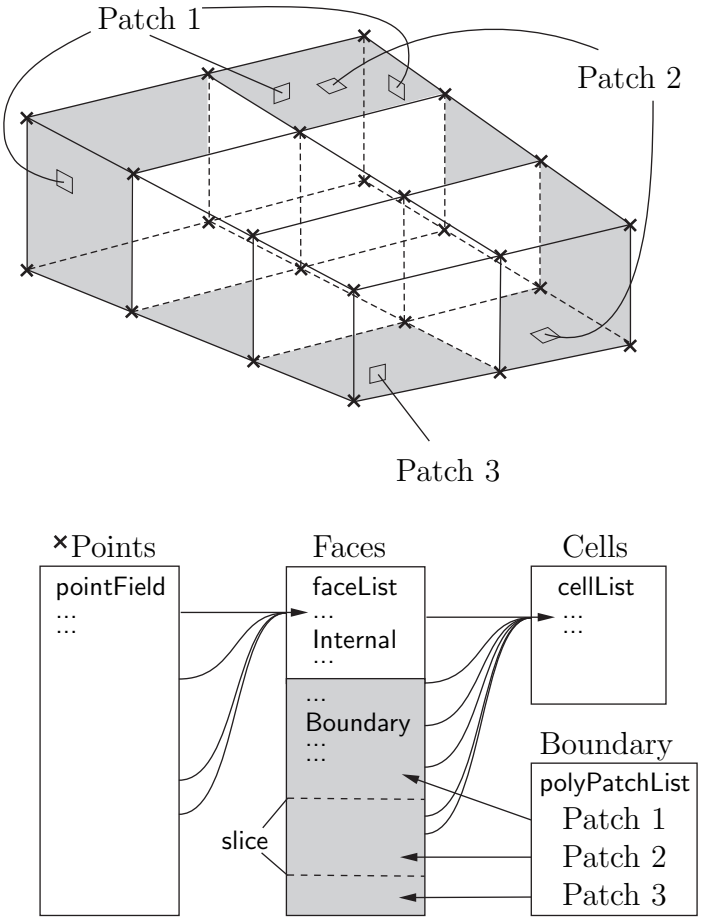



Figure 2.3: Schematic of the basic mesh description used in OpenFOAM

### 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 ** | $\phi_g$ | `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].

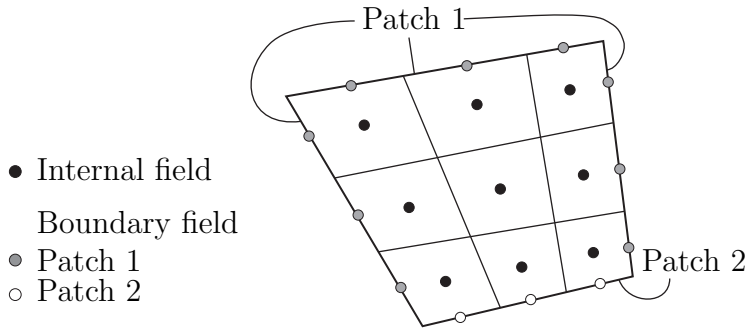
## 2.4 Equation discretisation

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

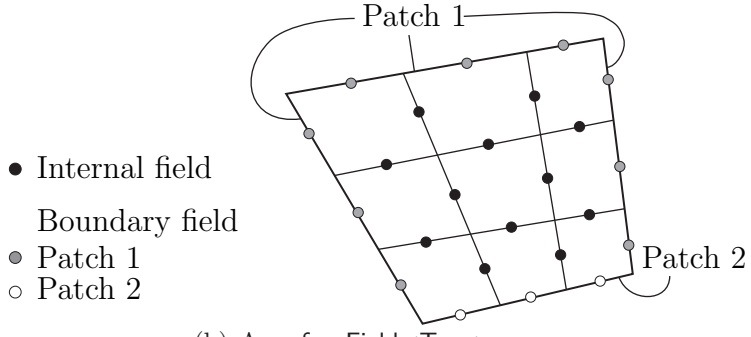
$$[A]\,[x] = [b] \tag{2.12}$$

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.

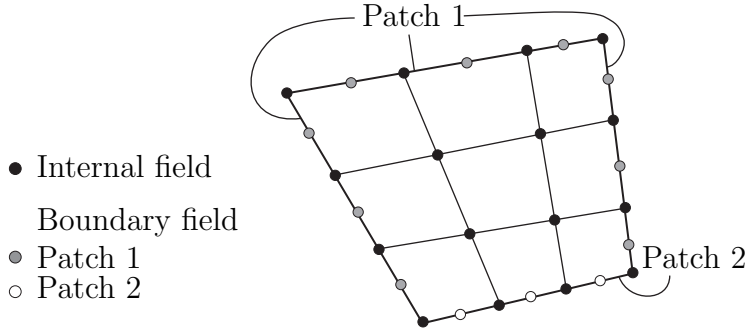
[1]The diagram is not an exact description of the class hierarchy, rather a representation of the general structure leading from some primitive classes to geometric<Type>Field.




(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)

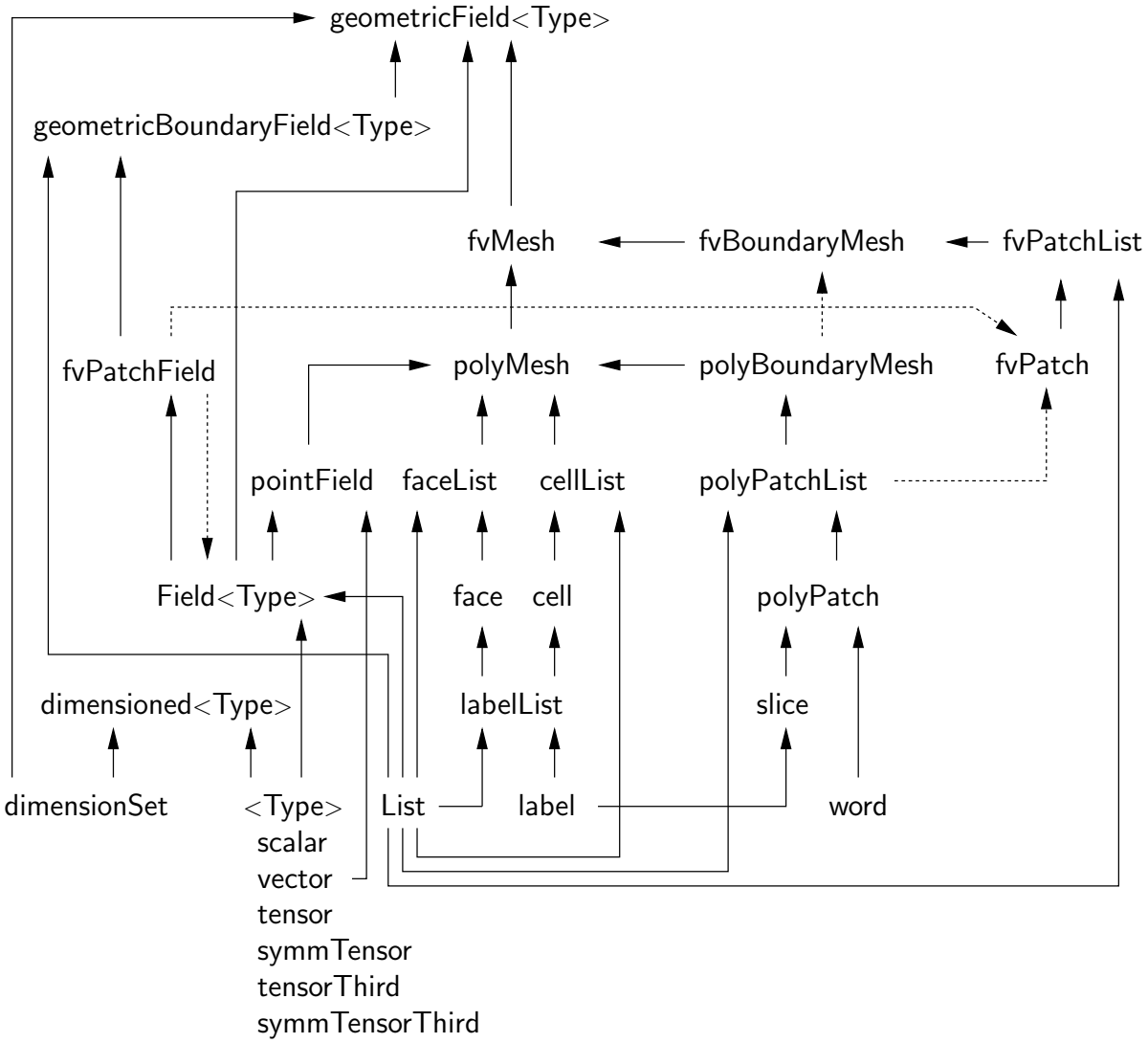



Figure 2.5: Basic class structure leading to geometricField<Type>

$[A]$ 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.* $\nabla^2$, $\nabla \bullet$ 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.

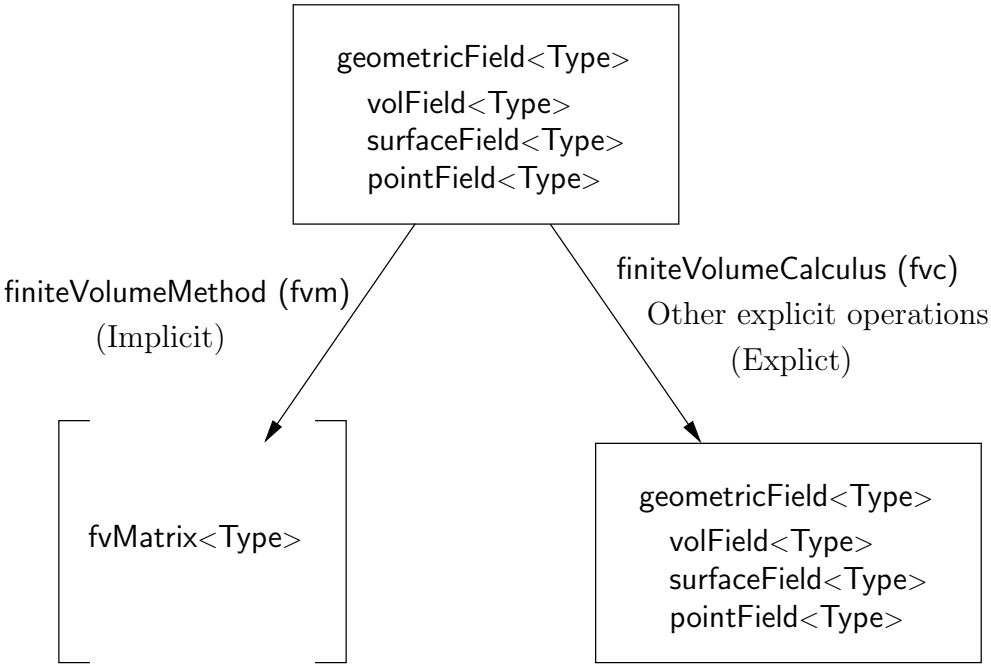



Figure 2.6: A geometricField<Type> and its operators

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, $\phi$ 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 $\phi_f$ 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 $\phi_f$ 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 $\gamma$ 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 $\phi_f$ 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 `gradScheme` keyword in an input file

**Gauss integration** is invoked using the `fvc::grad` function with `gradScheme 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`.

### 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 $\phi$ 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 $\phi^n$.

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

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 $\phi^o$.

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

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 Nicolson** uses the trapezoid rule to discretise the spatial terms, thereby taking a mean of current values $\phi^n$ and old values $\phi^o$.

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

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 Nicolson 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 $\phi_b$

- We can simply substitute $\phi_b$ in cases where the discretisation requires the value on a boundary face $\phi_f$, *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 $\phi_f$ we must interpolate the cell centre value to the boundary by

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

- $\phi_b$ 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.

# 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.

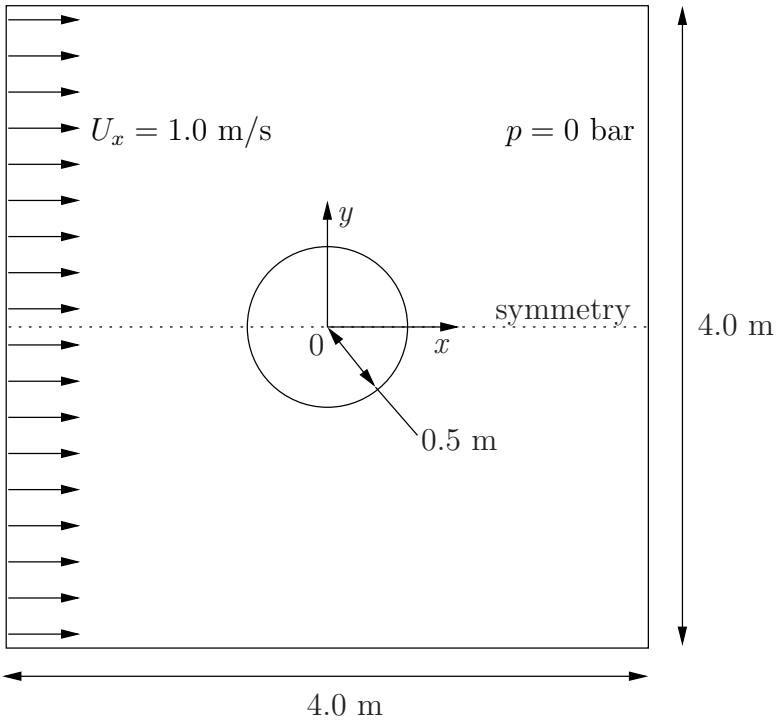



Figure 3.1: Geometry of flow round a cylinder

***Governing equations***

- Mass continuity for an incompressible fluid

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

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

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

***Boundary conditions***

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

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

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

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

### 3.1.2 Note on potentialFoam

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

### 3.1.3 Mesh generation

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

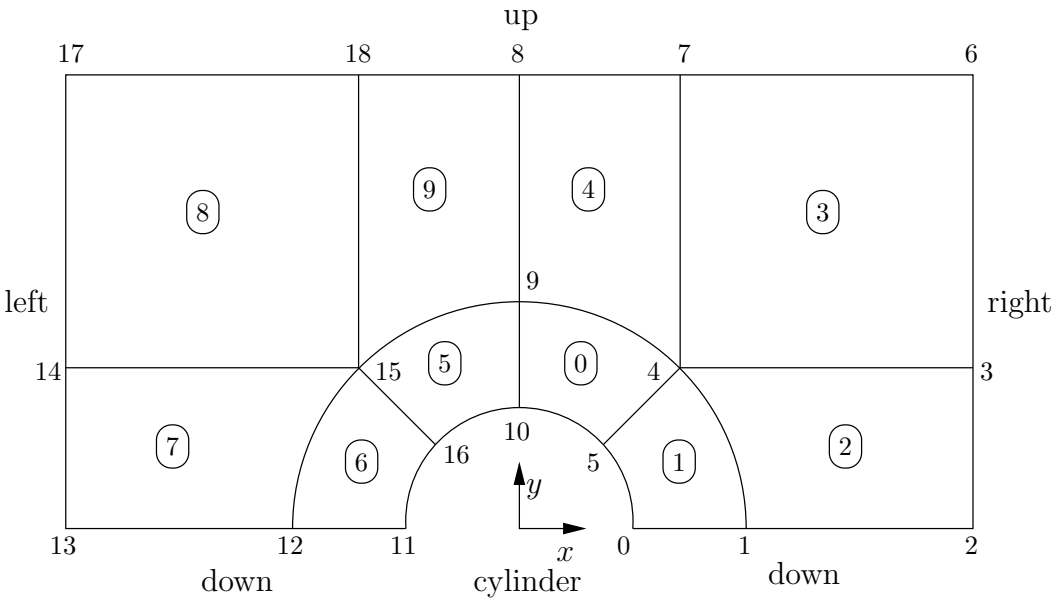



Figure 3.2: Blocks in cylinder geometry

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

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                      |                                            |
| \\      /  F ield              | OpenFOAM: The Open Source CFD Toolbox      |
|  \\    /   O peration          | Version:  2.4.0                            |
|   \\  /    A nd                | Web:      www.OpenFOAM.org                 |
|    \\/     M anipulation       |                                            |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    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)
```

```
86      arc 23 28 (0.34202 0.939693 0.5)
87      arc 11 16 (-0.469846 0.17101 -0.5)
88      arc 16 10 (-0.17101 0.469846 -0.5)
89      arc 12 15 (-0.939693 0.34202 -0.5)
90      arc 15 9 (-0.34202 0.939693 -0.5)
91      arc 30 35 (-0.469846 0.17101 0.5)
92      arc 35 29 (-0.17101 0.469846 0.5)
93      arc 31 34 (-0.939693 0.34202 0.5)
94      arc 34 28 (-0.34202 0.939693 0.5)
95  );
96
97  boundary
98  (
99      down
100     {
101         type symmetryPlane;
102         faces
103         (
104             (0 1 20 19)
105             (1 2 21 20)
106             (12 11 30 31)
107             (13 12 31 32)
108         );
109     }
110     right
111     {
112         type patch;
113         faces
114         (
115             (2 3 22 21)
116             (3 6 25 22)
117         );
118     }
119     up
120     {
121         type symmetryPlane;
122         faces
123         (
124             (7 8 27 26)
125             (6 7 26 25)
126             (8 18 37 27)
127             (18 17 36 37)
128         );
129     }
130     left
131     {
132         type patch;
133         faces
134         (
135             (14 13 32 33)
136             (17 14 33 36)
137         );
138     }
139     cylinder
140     {
141         type symmetry;
142         faces
143         (
144             (10 5 24 29)
145             (5 0 19 24)
146             (16 10 29 35)
147             (11 16 35 30)
148         );
149     }
150 );
151
152 mergePatchPairs
153 (
154 );
155
156 // ************************************************************************* //
```

### 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:  2.4.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     potentialFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         1;

deltaT          1;

writeControl    timeStep;

writeInterval   1;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression 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              scalar ULeft = 0.0;
66              label leftI = mesh().boundaryMesh().findPatchID("left");
67              const fvPatchVectorField& fvp = U.boundaryField()[leftI];
68              if (fvp.size())
69              {
70                  ULeft = fvp[0].x();
71              }
72              reduce(ULeft, maxOp<scalar>());
73
74              dimensionedScalar uInfX
75              (
76                  "uInfx",
77                  dimensionSet(0, 1, -1, 0, 0),
78                  ULeft
79              );
80
81              Info << "U at inlet = " << uInfX.value() << " m/s" << endl;
82
83
84              scalar magCylinder = 0.0;
85              label cylI = mesh().boundaryMesh().findPatchID("cylinder");
86              const fvPatchVectorField& cylFvp = mesh().C().boundaryField()[cylI];
87              if (cylFvp.size())
88              {
89                  magCylinder = mag(cylFvp[0]);
90              }
91              reduce(magCylinder, maxOp<scalar>());
92
93              dimensionedScalar radius
94              (
95                  "radius",
96                  dimensionSet(0, 1, 0, 0, 0),
97                  magCylinder
98              );
99
100             Info << "Cylinder radius = " << radius.value() << " m" << endl;
101
102             volVectorField UA
103             (
104                 IOobject
105                 (
106                     "UA",
107                     mesh().time().timeName(),
108                     U.mesh(),
109                     IOobject::NO_READ,
110                     IOobject::AUTO_WRITE
111                 ),
112                 U
113             );
114
115             Info<< "\nEvaluating analytical solution" << endl;
116
117             const volVectorField& centres = UA.mesh().C();
118             volScalarField magCentres(mag(centres));
119             volScalarField theta(acos((centres & vector(1,0,0))/magCentres));
120
121             volVectorField cs2theta
122             (
123                 cos(2*theta)*vector(1,0,0)
124               + sin(2*theta)*vector(0,1,0)
125             );
126
127             UA = uInfX*(dimensionedVector(vector(1,0,0))
128               - pow((radius/magCentres),2)*cs2theta);
129
130             // Force writing of UA (since time has not changed)
131             UA.write();
132
133             volScalarField error("error", mag(U-UA)/mag(UA));
134
135             Info<<"Writing relative error in U to " << error.objectPath()
```

```
136                     << endl;
137
138                 error.write();
139             #};
140         }
141     }
142
143
144     // ************************************************************************* //
```

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

## 3.2 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.

***Governing equations***

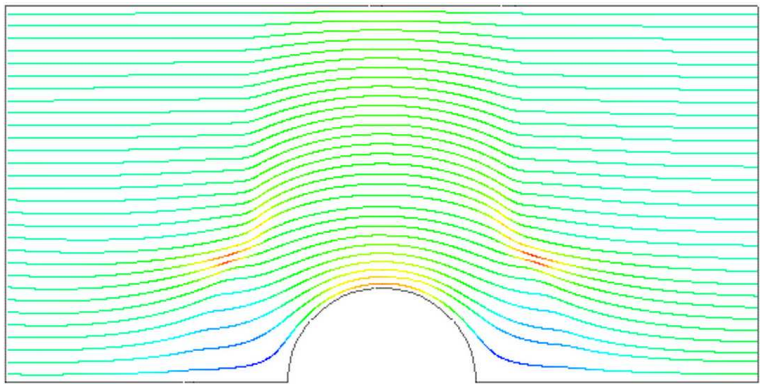
- Mass continuity for incompressible flow

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

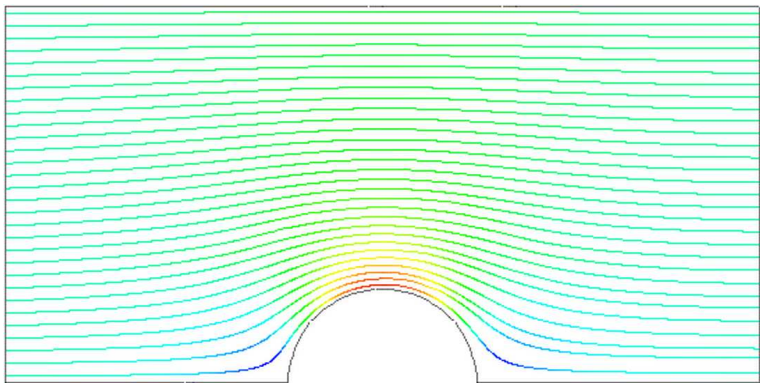
- Steady flow momentum equation

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

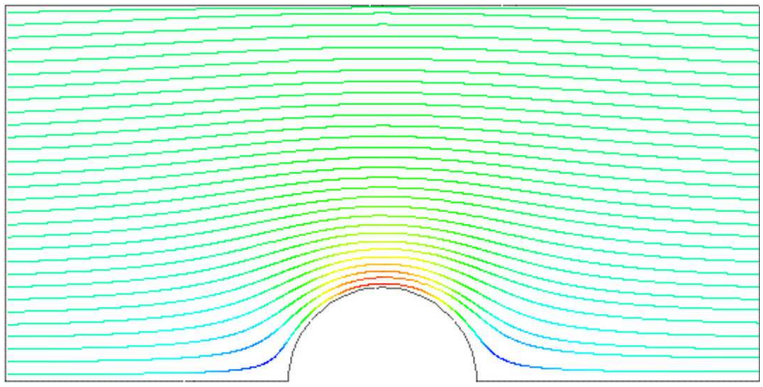
where $p$ is kinematic pressure and (in slightly over-simplistic terms) $\mathbf{R} = \nu_{eff}\nabla\mathbf{U}$ is the viscous stress term with an effective kinematic viscosity $\nu_{eff}$, calculated from selected transport and turbulence models.



(a) With no non-orthogonal correction



(b) With non-orthogonal correction



(c) Analytical solution

Figure 3.3: Streamlines of potential flow

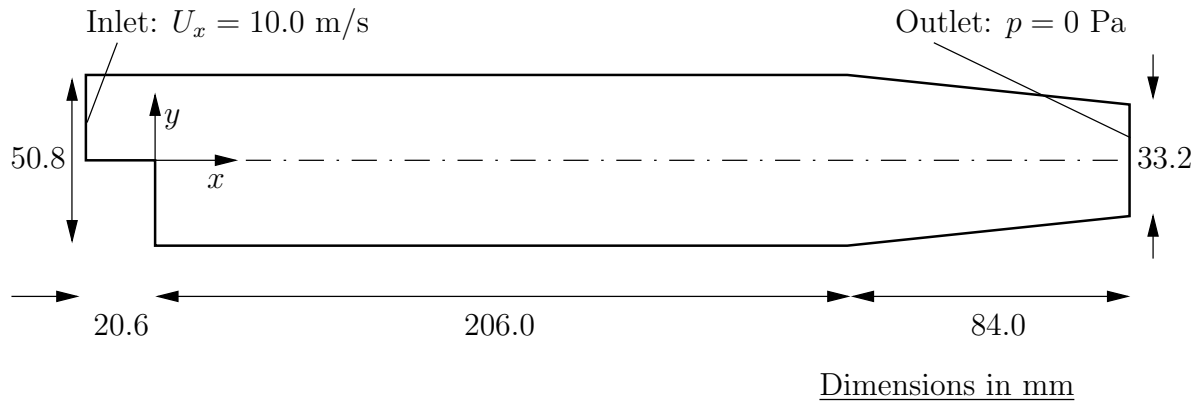



Figure 3.4: Geometry of backward-facing step

***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 $\nu = \mu/\rho = 18.1 \times 10^{-6}/1.293 = 14.0$ $\mu$m$^2$/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:  2.4.0                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
13     object      blockMeshDict;
```

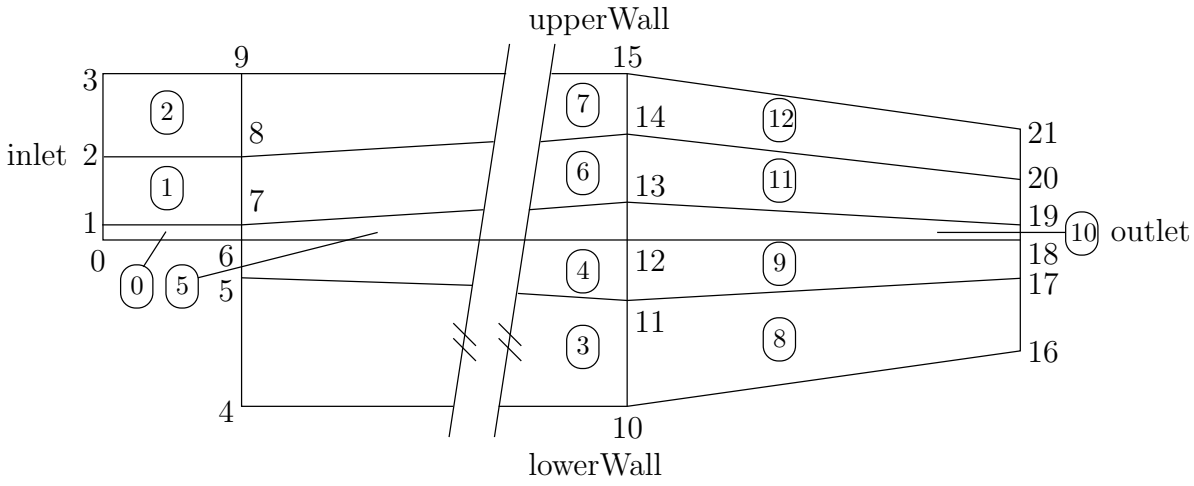



Figure 3.5: Blocks in backward-facing step

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

```
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 $\varepsilon$. 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 $\varepsilon$. **U** is given in the problem specification, but the values of $k$ and $\epsilon$ 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.

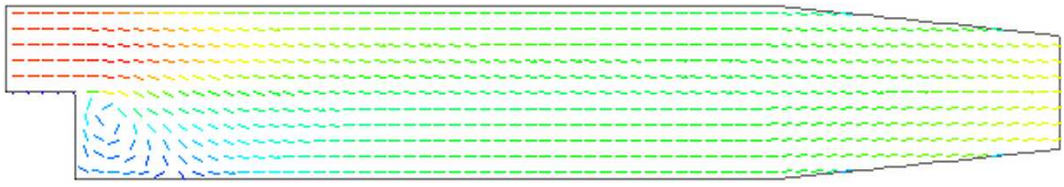
### 3.2.4 Case control

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.

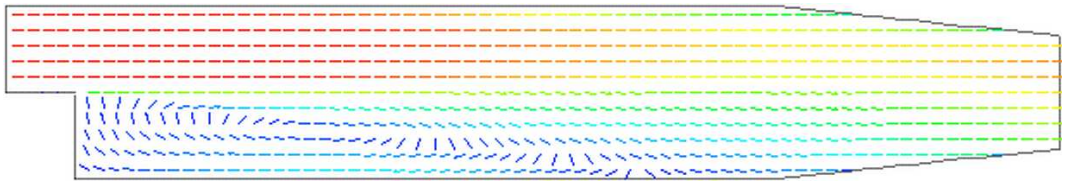
Special attention should be paid to the settings of *fvTolerances*. Although the top level simpleFoam code contains only equations for $p$ and $\mathbf{U}$, the turbulent model solves equations for $k$, $\varepsilon$ and $\mathbf{R}$, and tolerance settings are required for all 5 equations. A `solverTolerance`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

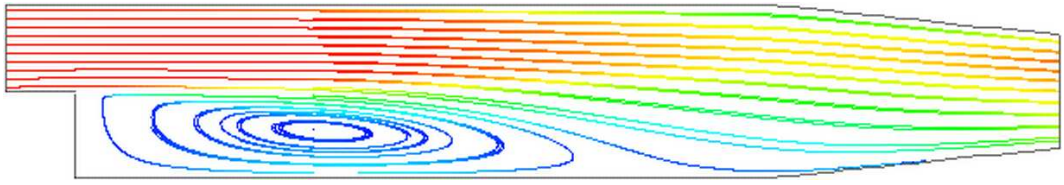
### 3.2.5 Running the case and post-processing



(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

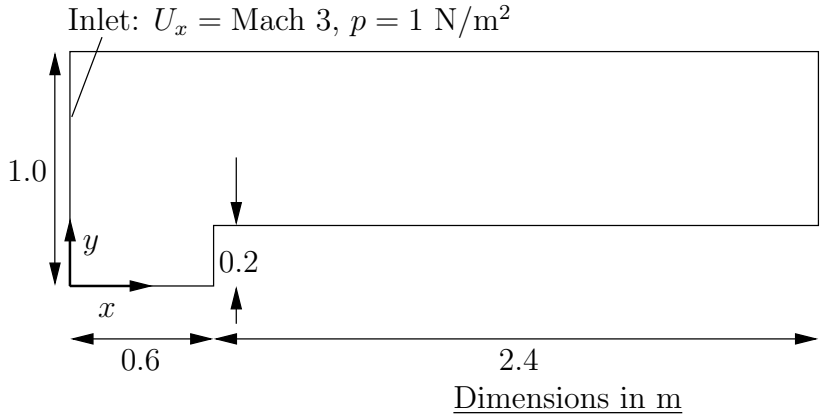



Figure 3.7: Geometry of the forward step geometry

***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 $\mu = 18.1 \mu$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 $k = 32.3$ $\mu$W/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:  2.4.0                            |
|   \\  /    A nd                | Web:      www.OpenFOAM.org                 |
|    \\/     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.

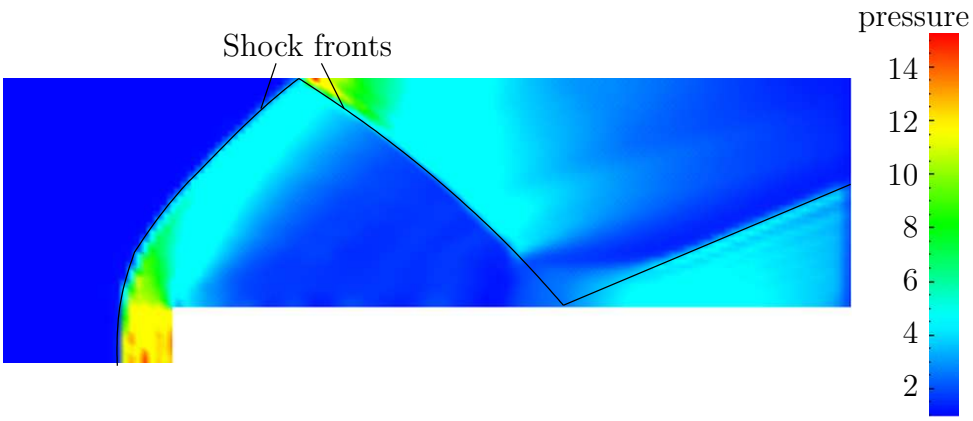



Figure 3.8: Shock fronts in the forward step problem

### 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

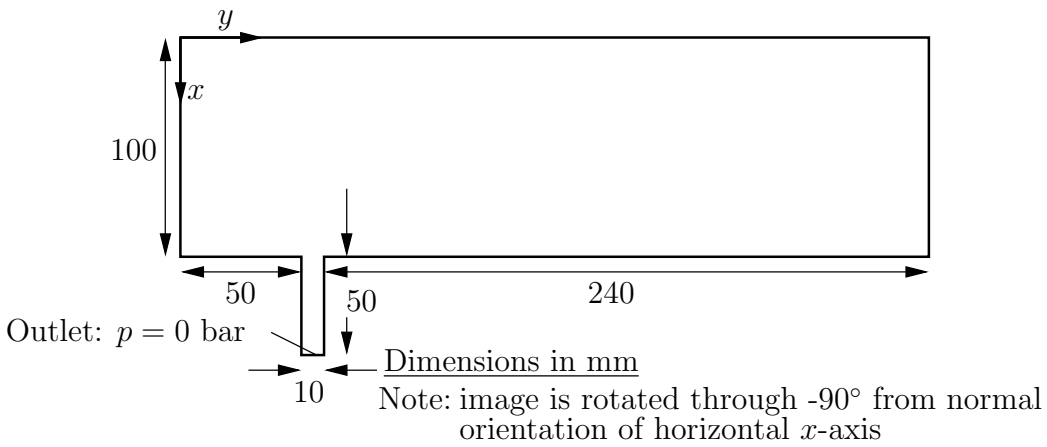



Figure 3.9: Geometry of a tank with outflow pipe

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

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

- Momentum equation for Newtonian fluid

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

***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:  2.4.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 0.1;

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

```
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

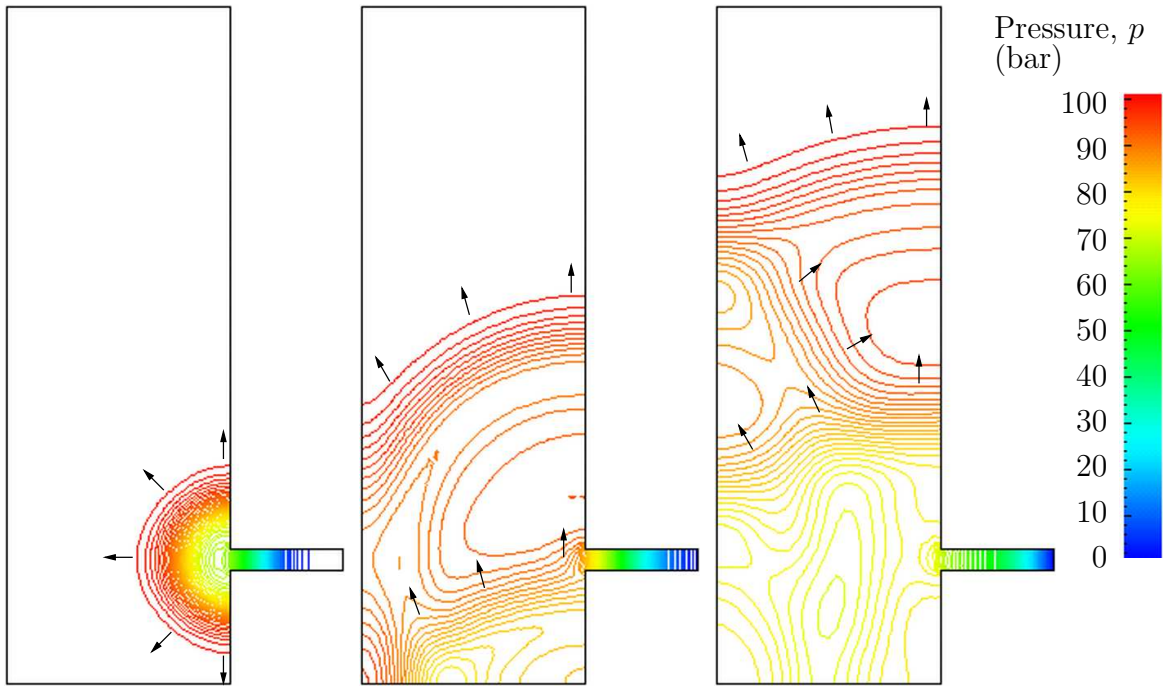
$$Co = \frac{U\,\Delta t}{\Delta x} \tag{3.18}$$

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.

```
1  /*--------------------------------*- C++ -*----------------------------------*\
2  | =========                 |                                                 |
3  | \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
4  |  \\    /   O peration     | Version:  2.4.0                                 |
5  |   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
6  |    \\/     M anipulation  |                                                 |
7  \*---------------------------------------------------------------------------*/
8  FoamFile
9  {
10     version     2.0;
11     format      ascii;
12     class       dictionary;
13     location    "system";
14     object      controlDict;
15 }
16 // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
17
18 application     sonicLiquidFoam;
19
20 startFrom       startTime;
21
22 startTime       0;
23
24 stopAt          endTime;
25
26 endTime         0.0001;
27
28 deltaT          5e-07;
29
30 writeControl    timeStep;
31
32 writeInterval   20;
33
34 purgeWrite      0;
35
36 writeFormat     ascii;
37
38 writePrecision  6;
39
```

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

### 3.4.4 Running the case

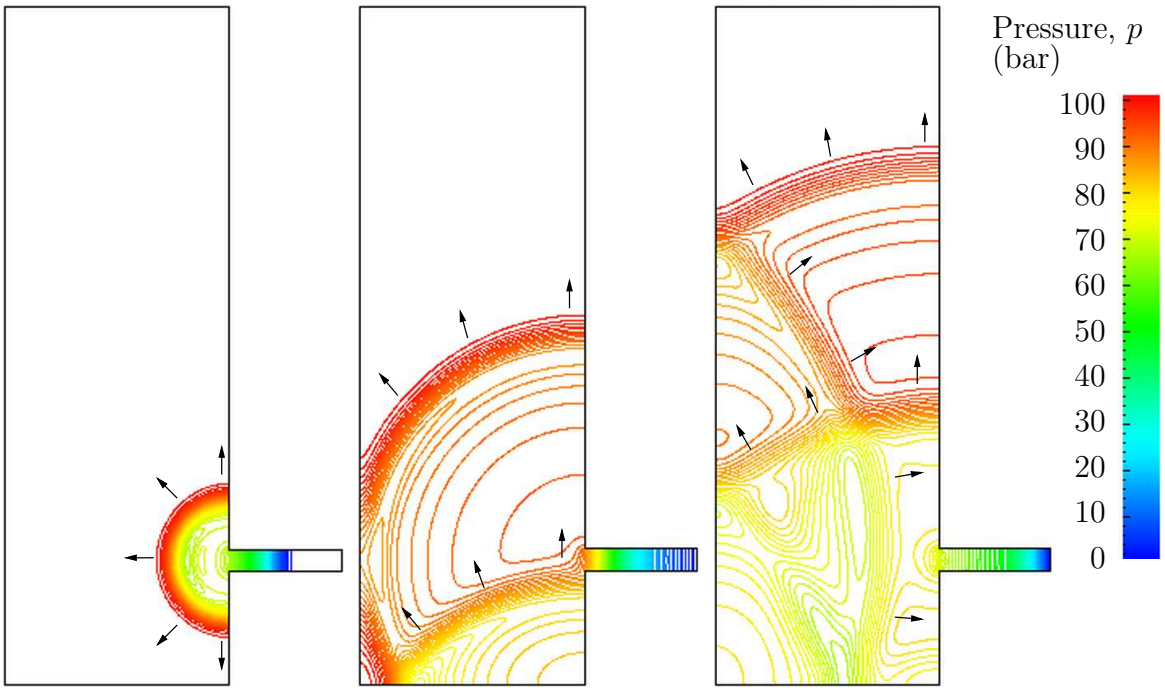



(a) At $t = 50$ µs (b) At $t = 100$ µs (c) At $t = 150$ µs

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***

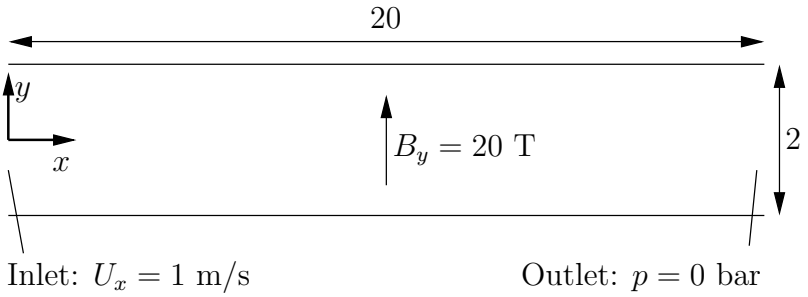



Figure 3.12: Geometry of the Hartmann problem

- 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:  2.4.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     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.

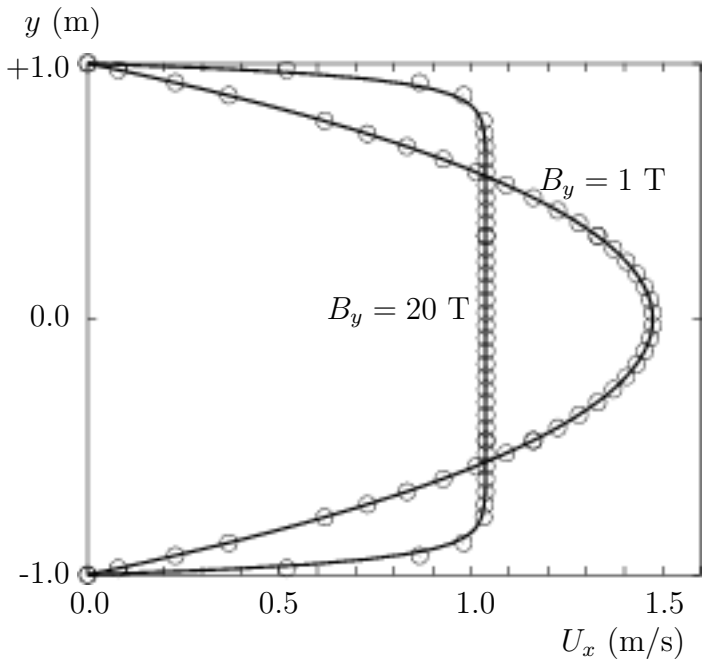



Figure 3.13: Velocity profile in the Hartmann problem for $B_y = 1$ T and $B_y = 20$ T.

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-23
+ tensor member function, P-23
- tensor member function, P-23
/ tensor member function, P-23
/*...*/ C++ syntax, U-79
// C++ syntax, U-79
OpenFOAM file syntax, U-108
# include C++ syntax, U-72, U-79
& tensor member function, P-23
&& tensor member function, P-23
^ tensor member function, P-23
<LESModel>Coeffs keyword, U-198
<RASModel>Coeffs keyword, U-198
<delta>Coeffs keyword, U-198
*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-21
addLayersControls keyword, U-153
adiabaticFlameT utility, U-99
adiabaticPerfectFluid model, U-103, U-193
adjointShapeOptimizationFoam solver, U-87
adjustableRunTime keyword entry, U-62, U-116
adjustTimeStep keyword, U-61, U-117
agglomerator keyword, U-128
algorithms tools, U-99
alphaContactAngle boundary condition, U-59
analytical solution, P-43
Animations window panel, U-176
anisotropicFilter model, U-105
Annotation window panel, U-25, U-176
ansysToFoam utility, U-93
APIfunctions model, U-103
applications, U-69
Apply button, U-172, U-176
applyBoundaryLayer utility, U-92
applyWallFunctionBoundaryConditions utility, U-92
arbitrarily unstructured, P-29
arc keyword entry, U-145
arc keyword, U-144
As keyword, U-196
ascii keyword entry, U-116
attachMesh utility, U-93
Auto Accept button, U-176
autoMesh library, U-100
autoPatch utility, U-93
autoRefineMesh utility, U-94
axes right-handed, U-143
axes right-handed rectangular Cartesian, P-13, U-18
axi-symmetric cases, U-140, U-151
axi-symmetric mesh, U-135

## B


background
process, U-25, U-82
backward
keyword entry, U-124
Backward differencing, P-37
barotropicCompressibilityModels
library, U-103
basicMultiComponentMixture model, U-102, U-194
basicSolidThermo
library, U-104
basicThermophysicalModels
library, U-102
binary
keyword entry, U-116
BirdCarreau model, U-106
blended differencing, P-36
block
expansion ratio, U-145
block keyword, U-144
blocking
keyword entry, U-81
blockMesh
library, U-100
blockMesh solver, P-45
blockMesh utility, U-37, U-92, U-141
blockMesh executable
vertex numbering, U-145
*blockMeshDict*
dictionary, U-18, U-20, U-36, U-49, U-141, U-151
blocks keyword, U-20, U-31, U-145
boundaries, U-135
boundary, U-135
*boundary*
dictionary, U-134, U-141
boundary keyword, U-147, U-148
boundary condition
alphaContactAngle, U-59
buoyantPressure, U-142
calculated, U-141
cyclic, U-140, U-149
directionMixed, U-141
empty, P-63, P-69, U-18, U-135, U-140
fixedGradient, U-141
fixedValue, U-141
fluxCorrectedVelocity, U-142
inlet, P-69
inletOutlet, U-142
mixed, U-141
movingWallVelocity, U-142
outlet, P-69
outletInlet, U-142
partialSlip, U-142
patch, U-139
pressureDirectedInletVelocity, U-142
pressureInletVelocity, U-142
pressureOutlet, P-63
pressureTransmissive, U-142
processor, U-140
setup, U-20
slip, U-142
supersonicFreeStream, U-142
surfaceNormalFixedValue, U-142
symmetryPlane, P-63, U-140
totalPressure, U-142
turbulentInlet, U-142
wall, U-40
wall, P-63, P-69, U-58, U-139, U-140
wedge, U-135, U-140, U-151
zeroGradient, U-141
boundary conditions, P-41
Dirichlet, P-41
inlet, P-42
Neumann, P-41
no-slip impermeable wall, P-42
outlet, P-42
physical, P-42
symmetry plane, P-42
boundaryField keyword, U-21, U-112
boundaryFoam solver, U-87
bounded
keyword entry, U-122, U-123
boxToCell keyword, U-60
boxTurb utility, U-92
breaking of a dam, U-56
buoyantBoussinesqPimpleFoam solver, U-90
buoyantBoussinesqSimpleFoam solver, U-90
buoyantPimpleFoam solver, U-90
buoyantPressure
boundary condition, U-142
buoyantSimpleFoam solver, U-90
button
Apply, U-172, U-176
Auto Accept, U-176
Choose Preset, U-174
Delete, U-172
Edit Color Map, U-174
Enable Line Series, U-35

Orientation Axes, U-25, U-176
Refresh Times, U-25
Rescale to Data Range, U-25
Reset, U-172
Set Ambient Color, U-175
Update GUI, U-173
Use Parallel Projection, U-25
Use parallel projection, U-175


## C


C++ syntax
`/*...*/`, U-79
`//`, U-79
`# include`, U-72, U-79
`cacheAgglomeration` keyword, U-128
calculated
boundary condition, U-141
`cAlpha` keyword, U-63
cases, U-107
`castellatedMesh` keyword, U-153
*castellatedMeshControls*
dictionary, U-154–U-156
`castellatedMeshControls` keyword, U-153
cavitatingDyMFoam solver, U-88
cavitatingFoam solver, U-88
cavity flow, U-17
ccm26ToFoam utility, U-93
CEI_ARCH
environment variable, U-186
CEI_HOME
environment variable, U-186
cell
expansion ratio, U-145
cell class, P-29
`cell`
keyword entry, U-187
`cellLimited`
keyword entry, U-122
`cellPoint`
keyword entry, U-187
`cellPointFace`
keyword entry, U-187
*cells*
dictionary, U-141
central differencing, P-36
cfdTools tools, U-100
cfx4ToFoam utility, U-93, U-160
changeDictionary utility, U-92
Charts window panel, U-176
checkMesh utility, U-93, U-162
chemFoam solver, U-89
chemistryModel
library, U-103
chemistryModel model, U-103
chemistrySolver model, U-103
chemkinToFoam utility, U-99
Choose Preset button, U-174
chtMultiRegionSimpleFoam solver, U-90
chtMultiRegionFoam solver, U-90
Chung
library, U-103
class
cell, P-29
dimensionSet, P-24, P-30, P-31
face, P-29
finiteVolumeCalculus, P-34
finiteVolumeMethod, P-34
fvMesh, P-29
fvSchemes, P-36
fvc, P-34
fvm, P-34
pointField, P-29
polyBoundaryMesh, P-29
polyMesh, P-29, U-131, U-133
polyPatchList, P-29
polyPatch, P-29
scalarField, P-27
scalar, P-22
slice, P-29
symmTensorField, P-27
symmTensorThirdField, P-27
tensorField, P-27
tensorThirdField, P-27
tensor, P-22
vectorField, P-27
vector, P-22, U-111
word, P-24, P-29
`class` keyword, U-109
`clockTime`
keyword entry, U-116
`cloud` keyword, U-189
cloudFunctionObjects
library, U-100
`cmptAv`
tensor member function, P-23
Co utility, U-95
coalChemistryFoam solver, U-90
coalCombustion
library, U-101
`cofactors`
tensor member function, P-23

coldEngineFoam solver, U-89
collapseEdges utility, U-94
Color By menu, U-175
Color Legend window, U-27
Color Legend window panel, U-174
Color Scale window panel, U-174
Colors window panel, U-176
compressibleInterDyMFoam solver, U-88
compressibleInterFoam solver, U-88
compressibleMultiphaseInterFoam solver, U-88
combinePatchFaces utility, U-95
comments, U-79
commsType keyword, U-81
compressed
    keyword entry, U-116
compressibleLESModels
    library, U-105
compressibleRASModels
    library, U-104
*constant* directory, U-107, U-193
constant model, U-102
constTransport model, U-103, U-194
containers tools, U-99
continuum
    mechanics, P-13
control
    of time, U-115
*controlDict*
    dictionary, P-65, U-22, U-31, U-42, U-51, U-62, U-107, U-168
*controlDict* file, P-48
convection, *see* divergence, P-36
convergence, U-39
conversion
    library, U-101
convertToMeters keyword, U-143, U-144
coordinate
    system, P-13
coordinate system, U-18
corrected
    keyword entry, U-122, U-123
Courant number, P-40, U-22
Cp keyword, U-196
cpuTime
    keyword entry, U-116
Crank Nicolson
    temporal discretisation, P-41
CrankNicolson
    keyword entry, U-124
createExternalCoupledPatchGeometry utility, U-92
createBaffles utility, U-93
createPatch utility, U-93
createTurbulenceFields utility, U-96
cross product, *see* tensor, vector cross product
CrossPowerLaw
    keyword entry, U-60
CrossPowerLaw model, U-106
cubeRootVolDelta model, U-105
cubicCorrected
    keyword entry, U-124
cubicCorrection
    keyword entry, U-121
curl, P-35
curl
    fvc member function, P-35
Current Time Controls menu, U-25, U-173
curve keyword, U-189
Cv keyword, U-196
cyclic
    boundary condition, U-140, U-149
cyclic
    keyword entry, U-140
cylinder
    flow around a, P-43

## D

d2dt2
    fvc member function, P-35
    fvm member function, P-35
dam
    breaking of a, U-56
datToFoam utility, U-93
db tools, U-99
ddt
    fvc member function, P-35
    fvm member function, P-35
DeardorffDiffStress model, U-105, U-106
debug keyword, U-153
decompose model, U-101
decomposePar utility, U-82, U-83, U-98
*decomposeParDict*
    dictionary, U-82
decomposition
    of field, U-82
    of mesh, U-82
decompositionMethods
    library, U-101
decompression of a tank, P-61
defaultFieldValues keyword, U-60

deformedGeom utility, U-94
Delete button, U-172
`delta` keyword, U-84, U-198
`deltaT` keyword, U-116
dependencies, U-72
dependency lists, U-72
`det`
    tensor member function, P-23
determinant, *see* tensor, determinant
`dev`
    tensor member function, P-23
`diag`
    tensor member function, P-23
`diagonal`
    keyword entry, U-126, U-127
`DIC`
    keyword entry, U-127
`DICGaussSeidel`
    keyword entry, U-127
dictionary
    *LESProperties*, U-198
    *PISO*, U-23
    *blockMeshDict*, U-18, U-20, U-36, U-49, U-141, U-151
    *boundary*, U-134, U-141
    *castellatedMeshControls*, U-154–U-156
    *cells*, U-141
    *controlDict*, P-65, U-22, U-31, U-42, U-51, U-62, U-107, U-168
    *decomposeParDict*, U-82
    *faces*, U-133, U-141
    *fvSchemes*, U-62, U-63, U-107, U-118
    *fvSolution*, U-107, U-125
    *mechanicalProperties*, U-51
    *neighbour*, U-134
    *owner*, U-133
    *points*, U-133, U-141
    *thermalProperties*, U-51
    *thermophysicalProperties*, U-193
    *transportProperties*, U-21, U-38, U-42
    *turbulenceProperties*, U-41, U-61, U-198
differencing
    Backward, P-37
    blended, P-36
    central, P-36
    Euler implicit, P-37
    Gamma, P-36
    MINMOD, P-36
    SUPERBEE, P-36
    upwind, P-36
    van Leer, P-36
`DILU`
    keyword entry, U-127
dimension
    checking in OpenFOAM, P-24, U-111
dimensional units, U-111
dimensioned<Type> template class, P-24
dimensionedTypes tools, U-100
`dimensions` keyword, U-21, U-112
dimensionSet class, P-24, P-30, P-31
dimensionSet tools, U-100
directionMixed
    boundary condition, U-141
directory
    *0.000000e+00*, U-108
    *0*, U-108
    *Make*, U-73
    *constant*, U-107, U-193
    *fluentInterface*, U-183
    *polyMesh*, U-107, U-133
    *processorN*, U-83
    *run*, U-107
    *system*, P-48, U-107
    *tutorials*, P-43, U-17
discretisation
    equation, P-31
Display window panel, U-24, U-25, U-172, U-173
`distance`
    keyword entry, U-156, U-189
distributed model, U-101
`distributed` keyword, U-84, U-85
distributionModels
    library, U-101
`div`
    fvc member function, P-35
    fvm member function, P-35
divergence, P-35, P-37
`divSchemes` keyword, U-118
dnsFoam solver, U-89
`doLayers` keyword, U-153
double inner product, *see* tensor,double inner product
DPMFoam solver, U-90
dsmc
    library, U-101
dsmcFieldsCalc utility, U-97
dsmcFoam solver, U-91
dsmcInitialise utility, U-92
`dx`
    keyword entry, U-187

dynamicFvMesh
library, U-100
dynamicMesh
library, U-100
dynLagrangian model, U-105
dynOneEqEddy model, U-105

## E

eConstThermo model, U-103, U-193
edgeGrading keyword, U-146
edgeMesh
library, U-101
edges keyword, U-144
Edit menu, U-175, U-176
Edit Color Map button, U-174
egrMixture model, U-102, U-194
electrostaticFoam solver, U-91
empty
boundary condition, P-63, P-69, U-18, U-135, U-140
empty
keyword entry, U-140
Enable Line Series button, U-35
endTime keyword, U-22, U-115, U-116
energy keyword, U-195
engine
library, U-101
engineCompRatio utility, U-97
engineFoam solver, U-89
engineSwirl utility, U-92
ensight74FoamExec utility, U-185
ENSIGHT7_INPUT
environment variable, U-186
ENSIGHT7_READER
environment variable, U-186
ensightFoamReader utility, U-95
enstrophy utility, U-95
environment variable
CEI_ARCH, U-186
CEI_HOME, U-186
ENSIGHT7_INPUT, U-186
ENSIGHT7_READER, U-186
FOAM_RUN, U-107
WM_ARCH_OPTION, U-76
WM_ARCH, U-76
WM_COMPILER_BIN, U-76
WM_COMPILER_DIR, U-76
WM_COMPILER_LIB, U-76
WM_COMPILER, U-76
WM_COMPILE_OPTION, U-76
WM_DIR, U-76
WM_MPLIB, U-76
WM_OPTIONS, U-76
WM_PRECISION_OPTION, U-76
WM_PROJECT_DIR, U-76
WM_PROJECT_INST_DIR, U-76
WM_PROJECT_USER_DIR, U-76
WM_PROJECT_VERSION, U-76
WM_PROJECT, U-76
wmake, U-76
equationOfState keyword, U-195
equilibriumCO utility, U-99
equilibriumFlameT utility, U-99
errorReduction keyword, U-161
Euler
keyword entry, U-124
Euler implicit
differencing, P-37
temporal discretisation, P-40
examples
decompression of a tank, P-61
flow around a cylinder, P-43
flow over backward step, P-50
Hartmann problem, P-67
supersonic flow over forward step, P-58
execFlowFunctionObjects utility, U-97
expandDictionary utility, U-99
expansionRatio keyword, U-160
explicit
temporal discretisation, P-40
extrude2DMesh utility, U-92
extrudeMesh utility, U-92
extrudeToRegionMesh utility, U-92

## F

face class, P-29
face keyword, U-189
faceAgglomerate utility, U-92
faceAreaPair
keyword entry, U-128
faceLimited
keyword entry, U-122
*faces*
dictionary, U-133, U-141
FDIC
keyword entry, U-127
featureAngle keyword, U-160
features keyword, U-154, U-155
field
U, U-23
p, U-23
decomposition, U-82

FieldField<Type> template class, P-30
fieldFunctionObjects
    library, U-100
fields, P-27
    mapping, U-168
fields tools, U-100
`fields` keyword, U-187
Field<Type> template class, P-27
`fieldValues` keyword, U-60
file
    *Make/files*, U-75
    *controlDict*, P-48
    *files*, U-73
    *g*, U-61
    *options*, U-73
    *snappyHexMeshDict*, U-152
    *transportProperties*, U-60
file format, U-108
fileFormats
    library, U-101
`fileModificationChecking` keyword, U-81
`fileModificationSkew` keyword, U-81
*files* file, U-73
`filteredLinear2`
    keyword entry, U-121
`finalLayerThickness` keyword, U-160
financialFoam solver, U-91
find script/alias, U-181
finite volume
    discretisation, P-25
    mesh, P-29
finiteVolume
    library, U-100
finiteVolume tools, U-100
finiteVolumeCalculus class, P-34
finiteVolumeMethod class, P-34
fireFoam solver, U-89
`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-81
flow
    free surface, U-56
    laminar, U-17
    steady, turbulent, P-50
    supersonic, P-58
    turbulent, U-17
flow around a cylinder, P-43
flow over backward step, P-50
flowType utility, U-95
fluent3DMeshToFoam utility, U-93
*fluentInterface* directory, U-183
fluentMeshToFoam utility, U-93, U-160
fluxCorrectedVelocity
    boundary condition, U-142
`fluxRequired` keyword, U-118
OpenFOAM
    cases, U-107
FOAM_RUN
    environment variable, U-107
foamCalc utility, U-33, U-97
foamCalcFunctions
    library, U-100
foamCorrectVrt script/alias, U-166
foamDataToFluent utility, U-95, U-183
foamDebugSwitches utility, U-99
`FoamFile` keyword, U-109
`foamFile`
    keyword entry, U-187
foamFormatConvert utility, U-99
foamHelp utility, U-99
foamInfoExec utility, U-99
foamJob script/alias, U-190
foamListTimes utility, U-97
foamLog script/alias, U-190
foamMeshToFluent utility, U-93, U-183
foamToEnsight utility, U-95
foamToEnsightParts utility, U-95
foamToGMV utility, U-95
foamToStarMesh utility, U-93
foamToSurface utility, U-93
foamToTecplot360 utility, U-95
foamToVTK utility, U-95
foamUpgradeCyclics utility, U-92
foamUpgradeFvSolution utility, U-92
foamyHexMeshBackgroundMesh utility, U-92
foamyHexMeshSurfaceSimplify utility, U-92
foamyHexMesh utility, U-92
foamyQuadMesh utility, U-93
forces
    library, U-100
foreground
    process, U-25
`format` keyword, U-109
`fourth`

keyword entry, U-122, U-123
functionObjectLibs keyword, U-181
functions keyword, U-117, U-180
fvc class, P-34
fvc member function
curl, P-35
d2dt2, P-35
ddt, P-35
div, P-35
gGrad, P-35
grad, P-35
laplacian, P-35
lsGrad, P-35
snGrad, P-35
snGradCorrection, P-35
sqrGradGrad, P-35
fvDOM
library, U-102
FVFunctionObjects
library, U-100
fvm class, P-34
fvm member function
d2dt2, P-35
ddt, P-35
div, P-35
laplacian, P-35
Su, P-35
SuSp, P-35
fvMatrices tools, U-100
fvMatrix template class, P-34
fvMesh class, P-29
fvMesh tools, U-100
fvMotionSolvers
library, U-101
*fvSchemes*
dictionary, U-62, U-63, U-107, U-118
fvSchemes class, P-36
fvSchemes
menu entry, U-52
*fvSolution*
dictionary, U-107, U-125

## G

*g* file, U-61
gambitToFoam utility, U-93, U-160
GAMG
keyword entry, U-53, U-126, U-127
Gamma
keyword entry, U-121
Gamma differencing, P-36
Gauss
keyword entry, U-122
Gauss's theorem, P-34
GaussSeidel
keyword entry, U-127
General window panel, U-175, U-176
general
keyword entry, U-116
genericFvPatchField
library, U-101
geometric-algebraic multi-grid, U-127
GeometricBoundaryField template class, P-30
geometricField<Type> template class, P-30
geometry keyword, U-153
gGrad
fvc member function, P-35
global tools, U-100
gmshToFoam utility, U-93
gnuplot
keyword entry, U-117, U-187
grad
fvc member function, P-35
(Grad Grad) squared, P-35
gradient, P-35, P-38
Gauss scheme, P-38
Gauss's theorem, U-52
least square fit, U-52
least squares method, P-38, U-52
surface normal, P-38
gradSchemes keyword, U-118
graph tools, U-100
graphFormat keyword, U-117
GuldersEGRLaminarFlameSpeed model, U-103
GuldersLaminarFlameSpeed model, U-102

## H

hConstThermo model, U-103, U-194
heheupsiReactionThermo model, U-102, U-194
Help menu, U-175
hePsiThermo model, U-102, U-194
heRhoThermo model, U-102, U-194
HerschelBulkley model, U-106
hExponentialThermo
library, U-104
Hf keyword, U-196
hierarchical
keyword entry, U-83, U-84
highCpCoeffs keyword, U-196
homogenousDynOneEqEddy model, U-105, U-106
homogenousDynSmagorinsky model, U-105
homogeneousMixture model, U-102, U-194
hPolynomialThermo model, U-103, U-194

## I

I
    tensor member function, P-23
icoFoam solver, U-17, U-21, U-22, U-25, U-87
icoPolynomial model, U-103, U-193
icoUncoupledKinematicParcelDyMFoam solver, U-90
icoUncoupledKinematicParcelFoam solver, U-90
ideasToFoam utility, U-160
ideasUnvToFoam utility, U-93
identities, *see* tensor, identities
identity, *see* tensor, identity
incompressibleLESModels
    library, U-105
incompressiblePerfectGas model, U-103, U-193
incompressibleRASModels
    library, U-104
incompressibleTransportModels
    library, P-53, U-106
incompressibleTurbulenceModels
    library, P-53
index
    notation, P-14, P-15
Information window panel, U-172
inhomogeneousMixture model, U-102, U-194
inlet
    boundary condition, P-69
inletOutlet
    boundary condition, U-142
inner product, *see* tensor, inner product
inotify
    keyword entry, U-81
inotifyMaster
    keyword entry, U-81
inside
    keyword entry, U-156
insideCells utility, U-94
interPhaseChangeDyMFoam solver, U-88
interPhaseChangeFoam solver, U-88
interDyMFoam solver, U-88
interfaceProperties
    library, U-106
interfaceProperties model, U-106
interFoam solver, U-88
interMixingFoam solver, U-88
internalField keyword, U-21, U-112
interpolation tools, U-100
interpolationScheme keyword, U-187
interpolations tools, U-100
interpolationSchemes keyword, U-118
inv
    tensor member function, P-23
iterations
    maximum, U-127

## J

janafThermo model, U-103, U-194
jobControl
    library, U-100
jplot
    keyword entry, U-117, U-187

## K

kEpsilon model, U-104
keyword
    As, U-196
    Cp, U-196
    Cv, U-196
    FoamFile, U-109
    Hf, U-196
    LESModel, U-198
    Pr, U-196
    RASModel, U-198
    Tcommon, U-196
    Thigh, U-196
    Tlow, U-196
    Ts, U-196
    addLayersControls, U-153
    adjustTimeStep, U-61, U-117
    agglomerator, U-128
    arc, U-144
    blocks, U-20, U-31, U-145
    block, U-144
    boundaryField, U-21, U-112
    boundary, U-147, U-148
    boxToCell, U-60
    cAlpha, U-63
    cacheAgglomeration, U-128
    castellatedMeshControls, U-153
    castellatedMesh, U-153
    class, U-109
    cloud, U-189
    commsType, U-81
    convertToMeters, U-143, U-144
    curve, U-189
    debug, U-153
    defaultFieldValues, U-60
    deltaT, U-116
    delta, U-84, U-198
    dimensions, U-21, U-112
    distributed, U-84, U-85

divSchemes, U-118
doLayers, U-153
edgeGrading, U-146
edges, U-144
endTime, U-22, U-115, U-116
energy, U-195
equationOfState, U-195
errorReduction, U-161
expansionRatio, U-160
face, U-189
featureAngle, U-160
features, U-154, U-155
fieldValues, U-60
fields, U-187
fileModificationChecking, U-81
fileModificationSkew, U-81
finalLayerThickness, U-160
firstTime, U-115
floatTransfer, U-81
fluxRequired, U-118
format, U-109
functionObjectLibs, U-181
functions, U-117, U-180
geometry, U-153
gradSchemes, U-118
graphFormat, U-117
highCpCoeffs, U-196
internalField, U-21, U-112
interpolationSchemes, U-118
interpolationScheme, U-187
laplacianSchemes, U-118
latestTime, U-38
layers, U-160
leastSquares, U-52
levels, U-156
libs, U-81, U-117
locationInMesh, U-155, U-156
location, U-109
lowCpCoeffs, U-196
manualCoeffs, U-84
maxAlphaCo, U-61
maxBoundarySkewness, U-161
maxConcave, U-161
maxCo, U-61, U-117
maxDeltaT, U-62
maxFaceThicknessRatio, U-160
maxGlobalCells, U-155
maxInternalSkewness, U-161
maxIter, U-127
maxLocalCells, U-155
maxNonOrtho, U-161
maxThicknessToMedialRatio, U-160
mergeLevels, U-128
mergePatchPairs, U-144
mergeTolerance, U-153
meshQualityControls, U-153
method, U-84
midPointAndFace, U-189
midPoint, U-189
minArea, U-161
minDeterminant, U-161
minFaceWeight, U-161
minFlatness, U-161
minMedianAxisAngle, U-160
minRefinementCells, U-155
minThickness, U-160
minTriangleTwist, U-161
minTwist, U-161
minVolRatio, U-161
minVol, U-161
mode, U-156
molWeight, U-195
mu, U-196
nAlphaSubCycles, U-63
nBufferCellsNoExtrude, U-160
nCellsBetweenLevels, U-155
nFaces, U-134
nFinestSweeps, U-128
nGrow, U-160
nLayerIter, U-160
nMoles, U-195
nPostSweeps, U-128
nPreSweeps, U-128
nRelaxIter, U-159, U-160
nRelaxedIter, U-160
nSmoothNormals, U-160
nSmoothPatch, U-159
nSmoothScale, U-161
nSmoothSurfaceNormals, U-160
nSmoothThickness, U-160
nSolveIter, U-159
neighbourPatch, U-149
numberOfSubdomains, U-84
n, U-84
object, U-109
order, U-84
outputControl, U-181
pRefCell, U-23, U-130
pRefValue, U-23, U-130
p_rhgRefCell, U-130

p_rhgRefValue, U-130
patchMap, U-168
patches, U-144
preconditioner, U-126, U-127
pressure, U-51
printCoeffs, U-41, U-198
processorWeights, U-83
processorWeights, U-84
purgeWrite, U-116
refGradient, U-141
refinementRegions, U-155, U-156
refinementSurfaces, U-155
refinementRegions, U-156
regions, U-60
relTol, U-53, U-126
relativeSizes, U-160
relaxed, U-161
resolveFeatureAngle, U-155
roots, U-84, U-85
runTimeModifiable, U-117
scotchCoeffs, U-84
setFormat, U-187
sets, U-187
simpleGrading, U-145
simulationType, U-41, U-61, U-198
smoother, U-128
snGradSchemes, U-118
snapControls, U-153
snap, U-153
solvers, U-125
solver, U-53, U-126
specie, U-195
spline, U-144
startFace, U-134
startFrom, U-22, U-115
startTime, U-22, U-115
stopAt, U-115
strategy, U-83, U-84
surfaceFormat, U-187
surfaces, U-187
thermoType, U-193
thermodynamics, U-195
timeFormat, U-116
timePrecision, U-117
timeScheme, U-118
tolerance, U-53, U-126, U-159
topoSetSource, U-60
traction, U-51
transport, U-195
turbulence, U-198
type, U-135, U-138
uniform, U-189
valueFraction, U-141
value, U-21, U-141
version, U-109
vertices, U-20, U-144
writeCompression, U-116
writeControl, U-22, U-62, U-116
writeFormat, U-55, U-116
writeInterval, U-22, U-32, U-116
writePrecision, U-116
<LESModel>Coeffs, U-198
<RASModel>Coeffs, U-198
<delta>Coeffs, U-198

keyword entry
CrankNicolson, U-124
CrossPowerLaw, U-60
DICGaussSeidel, U-127
DIC, U-127
DILU, U-127
Euler, U-124
FDIC, U-127
GAMG, U-53, U-126, U-127
Gamma, U-121
GaussSeidel, U-127
Gauss, U-122
LESModel, U-41, U-198
MGridGen, U-128
MUSCL, U-121
Newtonian, U-60
PBiCG, U-126
PCG, U-126
QUICK, U-124
RASModel, U-41, U-198
SFCD, U-121, U-124
UMIST, U-120
adjustableRunTime, U-62, U-116
arc, U-145
ascii, U-116
backward, U-124
binary, U-116
blocking, U-81
bounded, U-122, U-123
cellLimited, U-122
cellPointFace, U-187
cellPoint, U-187
cell, U-187
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-156, U-189
dx, U-187
empty, U-140
faceAreaPair, U-128
faceLimited, U-122
filteredLinear2, U-121
fixed, U-116
foamFile, U-187
fourth, U-122, U-123
general, U-116
gnuplot, U-117, U-187
hierarchical, U-83, U-84
inotifyMaster, U-81
inotify, U-81
inside, U-156
jplot, U-117, U-187
laminar, U-41, U-198
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-145
localEuler, U-124
manual, U-83, U-84
metis, U-84
midPoint, U-121
nextWrite, U-116
noWriteNow, U-116
nonBlocking, U-81
none, U-119, U-127
null, U-187
outputTime, U-181
outside, U-156
patch, U-140, U-188
polyLine, U-145
polySpline, U-145
processor, U-140
raw, U-117, U-187
runTime, U-32, U-116
scheduled, U-81
scientific, U-116
scotch, U-83, U-84
simpleSpline, U-145
simple, U-83, U-84
skewLinear, U-121, U-124
smoothSolver, U-126
startTime, U-22, U-115
steadyState, U-124
stl, U-187
symmetryPlane, U-140
timeStampMaster, U-81
timeStamp, U-81
timeStep, U-22, U-32, U-116, U-181
uncompressed, U-116
uncorrected, U-122, U-123
upwind, U-121, U-124
vanLeer, U-121
vtk, U-187
wall, U-140
wedge, U-140
writeControl, U-116
writeInterval, U-181
writeNow, U-115
xmgr, U-117, U-187
xyz, U-189
x, U-189
y, U-189
z, U-189
kivaToFoam utility, U-93
kkLOmega model, U-104
kOmega model, U-104
kOmegaSST model, U-104
kOmegaSSTSAS model, U-105
Kronecker delta, P-19

## L

lagrangian
    library, U-101
lagrangianIntermediate
    library, U-101
Lambda2 utility, U-95
LamBremhorstKE model, U-104
laminar model, U-104, U-105
laminar
    keyword entry, U-41, U-198
laminarFlameSpeedModels
    library, U-102
laplaceFilter model, U-105
Laplacian, P-36
laplacian, P-35
laplacian
    fvc member function, P-35
    fvm member function, P-35

laplacianFoam solver, U-86
`laplacianSchemes` keyword, U-118
`latestTime`
    keyword entry, U-115
`latestTime` keyword, U-38
LaunderGibsonRSTM model, U-104, U-105
LaunderSharmaKE model, U-104
`layers` keyword, U-160
`leastSquares`
    keyword entry, U-122
`leastSquares` keyword, U-52
LESdeltas
    library, U-105
LESfilters
    library, U-105
`LESModel`
    keyword entry, U-41, U-198
`LESModel` keyword, U-198
*LESProperties*
    dictionary, U-198
`levels` keyword, U-156
libraries, U-69
library
    Chung, U-103
    FVFunctionObjects, U-100
    LESdeltas, U-105
    LESfilters, U-105
    MGridGenGAMGAgglomeration, U-101
    ODE, U-101
    OSspecific, U-101
    OpenFOAM, U-99
    P1, U-102
    PV3FoamReader, U-171
    PVFoamReader, U-171
    SLGThermo, U-104
    Wallis, U-103
    autoMesh, U-100
    barotropicCompressibilityModels, U-103
    basicSolidThermo, U-104
    basicThermophysicalModels, U-102
    blockMesh, U-100
    chemistryModel, U-103
    cloudFunctionObjects, U-100
    coalCombustion, U-101
    compressibleLESModels, U-105
    compressibleRASModels, U-104
    conversion, U-101
    decompositionMethods, U-101
    distributionModels, U-101
    dsmc, U-101
    dynamicFvMesh, U-100
    dynamicMesh, U-100
    edgeMesh, U-101
    engine, U-101
    fieldFunctionObjects, U-100
    fileFormats, U-101
    finiteVolume, U-100
    foamCalcFunctions, U-100
    forces, U-100
    fvDOM, U-102
    fvMotionSolvers, U-101
    genericFvPatchField, U-101
    hExponentialThermo, U-104
    incompressibleLESModels, U-105
    incompressibleRASModels, U-104
    incompressibleTransportModels, P-53, U-106
    incompressibleTurbulenceModels, P-53
    interfaceProperties, U-106
    jobControl, U-100
    lagrangianIntermediate, U-101
    lagrangian, U-101
    laminarFlameSpeedModels, U-102
    linear, U-103
    liquidMixtureProperties, U-104
    liquidProperties, U-104
    meshTools, U-101
    molecularMeasurements, U-101
    molecule, U-101
    opaqueSolid, U-102
    pairPatchAgglomeration, U-101
    postCalc, U-100
    potential, U-101
    primitive, P-21
    radiationModels, U-102
    randomProcesses, U-101
    reactionThermophysicalModels, U-102
    sampling, U-100
    solidChemistryModel, U-104
    solidMixtureProperties, U-104
    solidParticle, U-101
    solidProperties, U-104
    solidSpecie, U-104
    solidThermo, U-104
    specie, U-103
    spray, U-101
    surfMesh, U-101
    surfaceFilmModels, U-106
    systemCall, U-100
    thermophysicalFunctions, U-103
    thermophysical, U-193

topoChangerFvMesh, U-101
triSurface, U-101
turbulence, U-101
twoPhaseProperties, U-106
utilityFunctionObjects, U-100
viewFactor, U-102
vtkFoam, U-171
vtkPV3Foam, U-171
libs keyword, U-81, U-117
lid-driven cavity flow, U-17
LienCubicKE model, U-104
LienCubicKELowRe model, U-104
LienLeschzinerLowRe model, U-104
Lights window panel, U-176
limited
keyword entry, U-122, U-123
limitedCubic
keyword entry, U-121
limitedLinear
keyword entry, U-121
line
keyword entry, U-145
Line Style menu, U-35
linear
library, U-103
linear
keyword entry, U-121, U-124
linearUpwind
keyword entry, U-121, U-124
liquid
electrically-conducting, P-67
liquidMixtureProperties
library, U-104
liquidProperties
library, U-104
lists, P-27
List<Type> template class, P-27
localEuler
keyword entry, U-124
location keyword, U-109
locationInMesh keyword, U-155, U-156
lowCpCoeffs keyword, U-196
lowReOneEqEddy model, U-105
LRDDiffStress model, U-105
LRR model, U-104
lsGrad
fvc member function, P-35
LTSInterFoam solver, U-89
LTSReactingFoam solver, U-89
LTSReactingParcelFoam solver, U-90


## M


Mach utility, U-95
mag
tensor member function, P-23
magneticFoam solver, U-91
magnetohydrodynamics, P-67
magSqr
tensor member function, P-23
*Make* directory, U-73
make script/alias, U-71
*Make/files* file, U-75
manual
keyword entry, U-83, U-84
manualCoeffs keyword, U-84
mapFields utility, U-31, U-38, U-42, U-55, U-92, U-168
mapping
fields, U-168
Marker Style menu, U-35
matrices tools, U-100
max
tensor member function, P-23
maxAlphaCo keyword, U-61
maxBoundarySkewness keyword, U-161
maxCo keyword, U-61, U-117
maxConcave keyword, U-161
maxDeltaT keyword, U-62
maxDeltaxyz model, U-105
maxFaceThicknessRatio keyword, U-160
maxGlobalCells keyword, U-155
maximum iterations, U-127
maxInternalSkewness keyword, U-161
maxIter keyword, U-127
maxLocalCells keyword, U-155
maxNonOrtho keyword, U-161
maxThicknessToMedialRatio keyword, U-160
mdEquilibrationFoam solver, U-91
mdFoam solver, U-91
mdInitialise utility, U-92
*mechanicalProperties*
dictionary, U-51
memory tools, U-100
menu
Color By, U-175
Current Time Controls, U-25, U-173
Edit, U-175, U-176
Help, U-175
Line Style, U-35
Marker Style, U-35
VCR Controls, U-25, U-173

View, U-175
menu entry
Plot Over Line, U-34
Save Animation, U-177
Save Screenshot, U-177
Settings, U-176
Show Color Legend, U-27
Solid Color, U-175
Toolbars, U-175
View Settings..., U-24
View Settings, U-25, U-175
Wireframe, U-175
fvSchemes, U-52
mergeLevels keyword, U-128
mergeMeshes utility, U-94
mergeOrSplitBaffles utility, U-94
mergePatchPairs keyword, U-144
mergeTolerance keyword, U-153
mesh
1-dimensional, U-135
1D, U-135
2-dimensional, U-135
2D, U-135
axi-symmetric, U-135
basic, P-29
block structured, U-141
decomposition, U-82
description, U-131
finite volume, P-29
generation, U-141, U-151
grading, U-141, U-145
grading, example of, P-50
non-orthogonal, P-43
refinement, P-61
resolution, U-29
specification, U-131
split-hex, U-152
Stereolithography (STL), U-152
surface, U-152
validity constraints, U-131
Mesh Parts window panel, U-24
meshes tools, U-100
meshQualityControls keyword, U-153
meshTools
library, U-101
message passing interface
openMPI, U-84
method keyword, U-84
metis
keyword entry, U-84
metisDecomp model, U-101
MGridGenGAMGAgglomeration
library, U-101
MGridGen
keyword entry, U-128
mhdFoam solver, P-69, U-91
midPoint
keyword entry, U-121
midPoint keyword, U-189
midPointAndFace keyword, U-189
min
tensor member function, P-23
minArea keyword, U-161
minDeterminant keyword, U-161
minFaceWeight keyword, U-161
minFlatness keyword, U-161
minMedianAxisAngle keyword, U-160
MINMOD differencing, P-36
minRefinementCells keyword, U-155
minThickness keyword, U-160
minTriangleTwist keyword, U-161
minTwist keyword, U-161
minVol keyword, U-161
minVolRatio keyword, U-161
mirrorMesh utility, U-94
mixed
boundary condition, U-141
mixedSmagorinsky model, U-105
mixtureAdiabaticFlameT utility, U-99
mode keyword, U-156
model
APIfunctions, U-103
BirdCarreau, U-106
CrossPowerLaw, U-106
DeardorffDiffStress, U-105, U-106
GuldersEGRLaminarFlameSpeed, U-103
GuldersLaminarFlameSpeed, U-102
HerschelBulkley, U-106
LRDDiffStress, U-105
LRR, U-104
LamBremhorstKE, U-104
LaunderGibsonRSTM, U-104, U-105
LaunderSharmaKE, U-104
LienCubicKELowRe, U-104
LienCubicKE, U-104
LienLeschzinerLowRe, U-104
NSRDSfunctions, U-103
Newtonian, U-106
NonlinearKEShih, U-104
PrandtlDelta, U-105

RNGkEpsilon, U-104
RaviPetersen, U-103
Smagorinsky2, U-105
Smagorinsky, U-105
SpalartAllmarasDDES, U-105
SpalartAllmarasIDDES, U-105
SpalartAllmaras, U-104–U-106
adiabaticPerfectFluid, U-103, U-193
anisotropicFilter, U-105
basicMultiComponentMixture, U-102, U-194
chemistryModel, U-103
chemistrySolver, U-103
constTransport, U-103, U-194
constant, U-102
cubeRootVolDelta, U-105
decompose, U-101
distributed, U-101
dynLagrangian, U-105
dynOneEqEddy, U-105
eConstThermo, U-103, U-193
egrMixture, U-102, U-194
hConstThermo, U-103, U-194
hPolynomialThermo, U-103, U-194
hePsiThermo, U-102, U-194
heRhoThermo, U-102, U-194
heheupsiReactionThermo, U-102, U-194
homogenousDynOneEqEddy, U-105, U-106
homogenousDynSmagorinsky, U-105
homogeneousMixture, U-102, U-194
icoPolynomial, U-103, U-193
incompressiblePerfectGas, U-103, U-193
inhomogeneousMixture, U-102, U-194
interfaceProperties, U-106
janafThermo, U-103, U-194
kEpsilon, U-104
kOmegaSSTSAS, U-105
kOmegaSST, U-104
kOmega, U-104
kkLOmega, U-104
laminar, U-104, U-105
laplaceFilter, U-105
lowReOneEqEddy, U-105
maxDeltaxyz, U-105
metisDecomp, U-101
mixedSmagorinsky, U-105
multiComponentMixture, U-102, U-194
oneEqEddy, U-105
perfectFluid, U-103, U-193
polynomialTransport, U-103, U-194
powerLaw, U-106
psiReactionThermo, U-102, U-194
psiuReactionThermo, U-102, U-194
ptsotchDecomp, U-101
pureMixture, U-102, U-194
qZeta, U-104
reactingMixture, U-102, U-194
realizableKE, U-104, U-105
reconstruct, U-101
rhoConst, U-103, U-193
rhoReactionThermo, U-102, U-194
scaleSimilarity, U-105
scotchDecomp, U-101
simpleFilter, U-105
singleStepReactingMixture, U-102, U-194
smoothDelta, U-105
specieThermo, U-103, U-194
spectEddyVisc, U-105
sutherlandTransport, U-103, U-194
v2f, U-104, U-105
vanDriestDelta, U-105, U-106
veryInhomogeneousMixture, U-102, U-194
modifyMesh utility, U-95
molecularMeasurements
library, U-101
molecule
library, U-101
`molWeight` keyword, U-195
moveDynamicMesh utility, U-94
moveEngineMesh utility, U-94
moveMesh utility, U-94
movingWallVelocity
boundary condition, U-142
MPI
openMPI, U-84
MRFInterFoam solver, U-89
MRFMultiphaseInterFoam solver, U-89
mshToFoam utility, U-93
`mu` keyword, U-196
multiComponentMixture model, U-102, U-194
multigrid
geometric-algebraic, U-127
multiphaseEulerFoam solver, U-89
multiphaseInterFoam solver, U-89
`MUSCL`
keyword entry, U-121

## N

`n` keyword, U-84
nabla
operator, P-25
`nAlphaSubCycles` keyword, U-63

nBufferCellsNoExtrude keyword, U-160
nCellsBetweenLevels keyword, U-155
*neighbour*
dictionary, U-134
neighbourPatch keyword, U-149
netgenNeutralToFoam utility, U-93
Newtonian
keyword entry, U-60
Newtonian model, U-106
nextWrite
keyword entry, U-116
nFaces keyword, U-134
nFinestSweeps keyword, U-128
nGrow keyword, U-160
nLayerIter keyword, U-160
nMoles keyword, U-195
non-orthogonal mesh, P-43
nonBlocking
keyword entry, U-81
none
keyword entry, U-119, U-127
NonlinearKEShih model, U-104
nonNewtonianIcoFoam solver, U-87
noWriteNow
keyword entry, U-116
nPostSweeps keyword, U-128
nPreSweeps keyword, U-128
nRelaxedIter keyword, U-160
nRelaxIter keyword, U-159, U-160
nSmoothNormals keyword, U-160
nSmoothPatch keyword, U-159
nSmoothScale keyword, U-161
nSmoothSurfaceNormals keyword, U-160
nSmoothThickness keyword, U-160
nSolveIter keyword, U-159
NSRDSfunctions model, U-103
null
keyword entry, U-187
numberOfSubdomains keyword, U-84

## O

object keyword, U-109
objToVTK utility, U-94
ODE
library, U-101
oneEqEddy model, U-105
Opacity text box, U-175
opaqueSolid
library, U-102
OpenFOAM
applications, U-69
file format, U-108
libraries, U-69
OpenFOAM
library, U-99
OpenFOAM file syntax
//, U-108
openMPI
message passing interface, U-84
MPI, U-84
operator
scalar, P-26
vector, P-25
Options window, U-176
*options* file, U-73
order keyword, U-84
Orientation Axes button, U-25, U-176
orientFaceZone utility, U-94
OSspecific
library, U-101
outer product, *see* tensor, outer product
outlet
boundary condition, P-69
outletInlet
boundary condition, U-142
outputControl keyword, U-181
outputTime
keyword entry, U-181
outside
keyword entry, U-156
*owner*
dictionary, U-133

## P

p field, U-23
P1
library, U-102
p_rhgRefCell keyword, U-130
p_rhgRefValue keyword, U-130
pairPatchAgglomeration
library, U-101
paraFoam, U-23, U-171
parallel
running, U-82
partialSlip
boundary condition, U-142
particleTracks utility, U-96
patch
boundary condition, U-139
patch
keyword entry, U-140, U-188
patchAverage utility, U-96

patches keyword, U-144
patchIntegrate utility, U-96
patchMap keyword, U-168
patchSummary utility, U-99
PBiCG
    keyword entry, U-126
PCG
    keyword entry, U-126
pdfPlot utility, U-97
PDRFoam solver, U-89
PDRMesh utility, U-95
Pe utility, U-95
perfectFluid model, U-103, U-193
permutation symbol, P-18
pimpleDyMFoam solver, U-87
pimpleFoam solver, U-87
Pipeline Browser window, U-24, U-172
*PISO*
    dictionary, U-23
pisoFoam solver, U-17, U-87
Plot Over Line
    menu entry, U-34
plot3dToFoam utility, U-93
pointField class, P-29
pointField<Type> template class, P-31
*points*
    dictionary, U-133, U-141
polyBoundaryMesh class, P-29
polyDualMesh utility, U-94
polyLine
    keyword entry, U-145
*polyMesh* directory, U-107, U-133
polyMesh class, P-29, U-131, U-133
polynomialTransport model, U-103, U-194
polyPatch class, P-29
polyPatchList class, P-29
polySpline
    keyword entry, U-145
porousInterFoam solver, U-89
porousSimpleFoam solver, U-87
post-processing, U-171
    post-processing
      paraFoam, U-171
postCalc
    library, U-100
postChannel utility, U-97
potentialFreeSurfaceFoam solver, U-89
potential
    library, U-101
potentialFoam solver, P-44, U-86
pow
    tensor member function, P-23
powerLaw model, U-106
pPrime2 utility, U-96
Pr keyword, U-196
PrandtlDelta model, U-105
preconditioner keyword, U-126, U-127
pRefCell keyword, U-23, U-130
pRefValue keyword, U-23, U-130
pressure keyword, U-51
pressure waves
    in liquids, P-62
pressureDirectedInletVelocity
    boundary condition, U-142
pressureInletVelocity
    boundary condition, U-142
pressureOutlet
    boundary condition, P-63
pressureTransmissive
    boundary condition, U-142
primitive
    library, P-21
primitives tools, U-100
printCoeffs keyword, U-41, U-198
processorWeights keyword, U-83
probeLocations utility, U-96
process
    background, U-25, U-82
    foreground, U-25
processor
    boundary condition, U-140
processor
    keyword entry, U-140
*processorN* directory, U-83
processorWeights keyword, U-84
Properties window panel, U-25, U-172, U-173
psiReactionThermo model, U-102, U-194
psiuReactionThermo model, U-102, U-194
ptot utility, U-97
ptsotchDecomp model, U-101
pureMixture model, U-102, U-194
purgeWrite keyword, U-116
PV3FoamReader
    library, U-171
PVFoamReader
    library, U-171

## Q

Q utility, U-95
QUICK
    keyword entry, U-124

qZeta model, U-104

## R

R utility, U-96
radiationModels
    library, U-102
randomProcesses
    library, U-101
`RASModel`
    keyword entry, U-41, U-198
`RASModel` keyword, U-198
RaviPetersen model, U-103
`raw`
    keyword entry, U-117, U-187
reactingFoam solver, U-89
reactingMixture model, U-102, U-194
reactingParcelFilmFoam solver, U-90
reactingParcelFoam solver, U-90
reactionThermophysicalModels
    library, U-102
realizableKE model, U-104, U-105
reconstruct model, U-101
reconstructPar utility, U-86
reconstructParMesh utility, U-98
redistributePar utility, U-98
`refGradient` keyword, U-141
refineHexMesh utility, U-95
`refinementRegions` keyword, U-156
refinementLevel utility, U-95
`refinementRegions` keyword, U-155, U-156
`refinementSurfaces` keyword, U-155
refineMesh utility, U-94
refineWallLayer utility, U-95
Refresh Times button, U-25
`regions` keyword, U-60
relative tolerance, U-126
`relativeSizes` keyword, U-160
`relaxed` keyword, U-161
`relTol` keyword, U-53, U-126
removeFaces utility, U-95
Render View window, U-176
Render View window panel, U-176
renumberMesh utility, U-94
Rescale to Data Range button, U-25
Reset button, U-172
`resolveFeatureAngle` keyword, U-155
restart, U-38
Reynolds number, U-17, U-21
rhoPorousSimpleFoam solver, U-87
rhoReactingBuoyantFoam solver, U-89
rhoCentralDyMFoam solver, U-87
rhoCentralFoam solver, U-87
rhoConst model, U-103, U-193
rhoLTSPimpleFoam solver, U-87
rhoPimpleFoam solver, U-87
rhoPimplecFoam solver, U-87
rhoReactingFoam solver, U-90
rhoReactionThermo model, U-102, U-194
rhoSimpleFoam solver, U-88
rhoSimplecFoam solver, U-87
rmdepall script/alias, U-77
RNGkEpsilon model, U-104
`roots` keyword, U-84, U-85
rotateMesh utility, U-94
run
    parallel, U-82
*run* directory, U-107
`runTime`
    keyword entry, U-32, U-116
`runTimeModifiable` keyword, U-117

## S

sammToFoam utility, U-93
sample utility, U-96, U-186
sampling
    library, U-100
`Save Animation`
    menu entry, U-177
`Save Screenshot`
    menu entry, U-177
scalar, P-14
    operator, P-26
scalar class, P-22
scalarField class, P-27
scalarTransportFoam solver, U-86
`scale`
    tensor member function, P-23
scalePoints utility, U-165
scaleSimilarity model, U-105
`scheduled`
    keyword entry, U-81
`scientific`
    keyword entry, U-116
`scotch`
    keyword entry, U-83, U-84
`scotchCoeffs` keyword, U-84
scotchDecomp model, U-101
script/alias
    find, U-181
    foamCorrectVrt, U-166
    foamJob, U-190
    foamLog, U-190

make, U-71
rmdepall, U-77
wclean, U-76
wmake, U-71
second time derivative, P-35
Seed window, U-177
selectCells utility, U-95
Set Ambient Color button, U-175
setFields utility, U-59, U-60, U-92
`setFormat` keyword, U-187
`sets` keyword, U-187
setSet utility, U-94
setsToZones utility, U-94
`Settings`
menu entry, U-176
settlingFoam solver, U-89
`SFCD`
keyword entry, U-121, U-124
shallowWaterFoam solver, U-87
shape, U-145
`Show Color Legend`
menu entry, U-27
SI units, U-112
simpleReactingParcelFoam solver, U-91
`simple`
keyword entry, U-83, U-84
simpleFilter model, U-105
simpleFoam solver, P-53, U-87
`simpleGrading` keyword, U-145
`simpleSpline`
keyword entry, U-145
`simulationType` keyword, U-41, U-61, U-198
singleCellMesh utility, U-94
singleStepReactingMixture model, U-102, U-194
`skew`
tensor member function, P-23
`skewLinear`
keyword entry, U-121, U-124
SLGThermo
library, U-104
slice class, P-29
slip
boundary condition, U-142
Smagorinsky model, U-105
Smagorinsky2 model, U-105
smapToFoam utility, U-95
smoothDelta model, U-105
`smoother` keyword, U-128
`smoothSolver`
keyword entry, U-126
`snap` keyword, U-153
`snapControls` keyword, U-153
snappyHexMesh utility
background mesh, U-153
cell removal, U-156
cell splitting, U-154
mesh layers, U-157
meshing process, U-152
snapping to surfaces, U-157
snappyHexMesh utility, U-93, U-151
*snappyHexMeshDict* file, U-152
`snGrad`
fvc member function, P-35
`snGradCorrection`
fvc member function, P-35
`snGradSchemes` keyword, U-118
`Solid Color`
menu entry, U-175
solidChemistryModel
library, U-104
solidDisplacementFoam solver, U-91
solidDisplacementFoam solver, U-51
solidEquilibriumDisplacementFoam solver, U-91
solidMixtureProperties
library, U-104
solidParticle
library, U-101
solidProperties
library, U-104
solidSpecie
library, U-104
solidThermo
library, U-104
solver
DPMFoam, U-90
LTSInterFoam, U-89
LTSReactingFoam, U-89
LTSReactingParcelFoam, U-90
MRFInterFoam, U-89
MRFMultiphaseInterFoam, U-89
PDRFoam, U-89
SRFPimpleFoam, U-87
SRFSimpleFoam, U-87
XiFoam, U-90
adjointShapeOptimizationFoam, U-87
blockMesh, P-45
boundaryFoam, U-87
buoyantBoussinesqPimpleFoam, U-90
buoyantBoussinesqSimpleFoam, U-90
buoyantPimpleFoam, U-90

buoyantSimpleFoam, U-90
cavitatingDyMFoam, U-88
cavitatingFoam, U-88
chemFoam, U-89
chtMultiRegionFoam, U-90
chtMultiRegionSimpleFoam, U-90
coalChemistryFoam, U-90
coldEngineFoam, U-89
compressibleInterDyMFoam, U-88
compressibleInterFoam, U-88
compressibleMultiphaseInterFoam, U-88
dnsFoam, U-89
dsmcFoam, U-91
electrostaticFoam, U-91
engineFoam, U-89
financialFoam, U-91
fireFoam, U-89
icoFoam, U-17, U-21, U-22, U-25, U-87
icoUncoupledKinematicParcelDyMFoam, U-90
icoUncoupledKinematicParcelFoam, U-90
interDyMFoam, U-88
interFoam, U-88
interMixingFoam, U-88
interPhaseChangeDyMFoam, U-88
interPhaseChangeFoam, U-88
laplacianFoam, U-86
magneticFoam, U-91
mdEquilibrationFoam, U-91
mdFoam, U-91
mhdFoam, P-69, U-91
multiphaseEulerFoam, U-89
multiphaseInterFoam, U-89
nonNewtonianIcoFoam, U-87
pimpleDyMFoam, U-87
pimpleFoam, U-87
pisoFoam, U-17, U-87
porousInterFoam, U-89
porousSimpleFoam, U-87
potentialFreeSurfaceFoam, U-89
potentialFoam, P-44, U-86
reactingFoam, U-89
reactingParcelFilmFoam, U-90
reactingParcelFoam, U-90
rhoCentralDyMFoam, U-87
rhoCentralFoam, U-87
rhoLTSPimpleFoam, U-87
rhoPimpleFoam, U-87
rhoPimplecFoam, U-87
rhoReactingFoam, U-90
rhoSimpleFoam, U-88
rhoSimplecFoam, U-87
rhoPorousSimpleFoam, U-87
rhoReactingBuoyantFoam, U-89
scalarTransportFoam, U-86
settlingFoam, U-89
shallowWaterFoam, U-87
simpleReactingParcelFoam, U-91
simpleFoam, P-53, U-87
solidDisplacementFoam, U-91
solidDisplacementFoam, U-51
solidEquilibriumDisplacementFoam, U-91
sonicDyMFoam, U-88
sonicFoam, P-59, U-88
sonicLiquidFoam, P-63, U-88
sprayEngineFoam, U-91
sprayFoam, U-91
thermoFoam, U-90
twoLiquidMixingFoam, U-89
twoPhaseEulerFoam, U-89
uncoupledKinematicParcelFoam, U-91
`solver` keyword, U-53, U-126
solver relative tolerance, U-126
solver tolerance, U-126
`solvers` keyword, U-125
sonicDyMFoam solver, U-88
sonicFoam solver, P-59, U-88
sonicLiquidFoam solver, P-63, U-88
source, P-35
SpalartAllmaras model, U-104–U-106
SpalartAllmarasDDES model, U-105
SpalartAllmarasIDDES model, U-105
specie
library, U-103
`specie` keyword, U-195
specieThermo model, U-103, U-194
spectEddyVisc model, U-105
`spline` keyword, U-144
splitCells utility, U-95
splitMesh utility, U-94
splitMeshRegions utility, U-94
spray
library, U-101
sprayEngineFoam solver, U-91
sprayFoam solver, U-91
`sqr`
tensor member function, P-23
`sqrGradGrad`
fvc member function, P-35
SRFPimpleFoam solver, U-87

SRFSimpleFoam solver, U-87
star3ToFoam utility, U-93
star4ToFoam utility, U-93
`startFace` keyword, U-134
`startFrom` keyword, U-22, U-115
starToFoam utility, U-160
`startTime`
    keyword entry, U-22, U-115
`startTime` keyword, U-22, U-115
steady flow
    turbulent, P-50
steadyParticleTracks utility, U-96
`steadyState`
    keyword entry, U-124
Stereolithography (STL), U-152
stitchMesh utility, U-94
`stl`
    keyword entry, U-187
`stopAt` keyword, U-115
`strategy` keyword, U-83, U-84
streamFunction utility, U-95
stress analysis of plate with hole, U-44
stressComponents utility, U-96
Style window panel, U-24, U-175
`Su`
    fvm member function, P-35
subsetMesh utility, U-94
summation convention, P-15
SUPERBEE differencing, P-36
supersonic flow, P-58
supersonic flow over forward step, P-58
supersonicFreeStream
    boundary condition, U-142
surfaceLambdaMuSmooth utility, U-97
surface mesh, U-152
surfaceAdd utility, U-97
surfaceAutoPatch utility, U-97
surfaceBooleanFeatures utility, U-97
surfaceCheck utility, U-97
surfaceClean utility, U-97
surfaceCoarsen utility, U-97
surfaceConvert utility, U-97
surfaceFeatureConvert utility, U-97
surfaceFeatureExtract utility, U-97, U-155
surfaceField<Type> template class, P-31
surfaceFilmModels
    library, U-106
surfaceFind utility, U-97
`surfaceFormat` keyword, U-187
surfaceHookUp utility, U-97
surfaceInertia utility, U-97
surfaceMesh tools, U-100
surfaceMeshConvert utility, U-97
surfaceMeshConvertTesting utility, U-97
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-187
surfaceSplitByPatch utility, U-98
surfaceSplitByTopology utility, U-98
surfaceSplitNonManifolds utility, U-98
surfaceSubset utility, U-98
surfaceToPatch utility, U-98
surfaceTransformPoints utility, U-98
surfMesh
    library, U-101
`SuSp`
    fvm member function, P-35
sutherlandTransport model, U-103, U-194
`symm`
    tensor member function, P-23
symmetryPlane
    boundary condition, P-63, U-140
`symmetryPlane`
    keyword entry, U-140
symmTensorField class, P-27
symmTensorThirdField class, P-27
*system* directory, P-48, U-107
systemCall
    library, U-100

## T

`T()`
    tensor member function, P-23
`Tcommon` keyword, U-196
template class
    GeometricBoundaryField, P-30
    fvMatrix, P-34
    dimensioned<Type>, P-24
    FieldField<Type>, P-30
    Field<Type>, P-27
    geometricField<Type>, P-30
    List<Type>, P-27
    pointField<Type>, P-31

surfaceField<Type>, P-31
volField<Type>, P-31
temporal discretisation, P-40
Crank Nicolson, P-41
Euler implicit, P-40
explicit, P-40
in OpenFOAM, P-41
temporalInterpolate utility, U-97
tensor, P-13
addition, P-16
algebraic operations, P-16
algebraic operations in OpenFOAM, P-22
antisymmetric, *see* tensor, skew
calculus, P-25
classes in OpenFOAM, P-21
cofactors, P-20
component average, P-18
component maximum, P-18
component minimum, P-18
determinant, P-20
deviatoric, P-20
diagonal, P-20
dimension, P-14
double inner product, P-17
geometric transformation, P-19
Hodge dual, P-21
hydrostatic, P-20
identities, P-19
identity, P-19
inner product, P-16
inverse, P-21
magnitude, P-18
magnitude squared, P-18
mathematics, P-13
notation, P-15
$n$th power, P-18
outer product, P-17
rank, P-14
rank 3, P-15
scalar division, P-16
scalar multiplication, P-16
scale function, P-18
second rank, P-14
skew, P-20
square of, P-18
subtraction, P-16
symmetric, P-20
symmetric rank 2, P-14
symmetric rank 3, P-15
trace, P-20
transformation, P-19
transpose, P-14, P-20
triple inner product, P-17
vector cross product, P-18
tensor class, P-22
tensor member function
*, P-23
+, P-23
-, P-23
/, P-23
&, P-23
&&, P-23
^, P-23
cmptAv, P-23
cofactors, P-23
det, P-23
dev, P-23
diag, P-23
I, P-23
inv, P-23
mag, P-23
magSqr, P-23
max, P-23
min, P-23
pow, P-23
scale, P-23
skew, P-23
sqr, P-23
symm, P-23
T(), P-23
tr, P-23
transform, P-23
tensorField class, P-27
tensorThirdField class, P-27
tetgenToFoam utility, U-93
text box
Opacity, U-175
*thermalProperties*
dictionary, U-51
thermodynamics keyword, U-195
thermoFoam solver, U-90
thermophysical
library, U-193
thermophysicalFunctions
library, U-103
*thermophysicalProperties*
dictionary, U-193
thermoType keyword, U-193
Thigh keyword, U-196
time

control, U-115
time derivative, P-35
first, P-37
second, P-35, P-37
time step, U-22
`timeFormat` keyword, U-116
`timePrecision` keyword, U-117
`timeScheme` keyword, U-118
`timeStamp`
keyword entry, U-81
`timeStampMaster`
keyword entry, U-81
`timeStep`
keyword entry, U-22, U-32, U-116, U-181
`Tlow` keyword, U-196
tolerance
solver, U-126
solver relative, U-126
`tolerance` keyword, U-53, U-126, U-159
`Toolbars`
menu entry, U-175
tools
algorithms, U-99
cfdTools, U-100
containers, U-99
db, U-99
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-94
`topoSetSource` keyword, U-60
totalPressure
boundary condition, U-142
`tr`
tensor member function, P-23
trace, *see* tensor, trace
`traction` keyword, U-51
`transform`
tensor member function, P-23
transformPoints utility, U-94
`transport` keyword, U-195
*transportProperties*
dictionary, U-21, U-38, U-42
*transportProperties* file, U-60
triple inner product, P-17
triSurface
library, U-101
`Ts` keyword, U-196
turbulence
dissipation, U-40
kinetic energy, U-40
length scale, U-41
turbulence
library, U-101
`turbulence` keyword, U-198
turbulence model
RAS, U-40
*turbulenceProperties*
dictionary, U-41, U-61, U-198
turbulent flow
steady, P-50
turbulentInlet
boundary condition, U-142
tutorials
breaking of a dam, U-56
lid-driven cavity flow, U-17
stress analysis of plate with hole, U-44
*tutorials* directory, P-43, U-17
twoLiquidMixingFoam solver, U-89
twoPhaseEulerFoam solver, U-89
twoPhaseProperties
library, U-106
`type` keyword, U-135, U-138


## U


`U` field, U-23
Ucomponents utility, P-70
`UMIST`
keyword entry, U-120
`uncompressed`
keyword entry, U-116
`uncorrected`
keyword entry, U-122, U-123
uncoupledKinematicParcelFoam solver, U-91
`uniform` keyword, U-189
units

base, U-112
of measurement, P-24, U-111
S.I. base, P-24
SI, U-112
Système International, U-112
United States Customary System, U-112
USCS, U-112
Update GUI button, U-173
uprime utility, U-95
upwind
keyword entry, U-121, U-124
upwind differencing, P-36, U-62
USCS units, U-112
Use Parallel Projection button, U-25
Use parallel projection button, U-175
utility
Co, U-95
Lambda2, U-95
Mach, U-95
PDRMesh, U-95
Pe, U-95
Q, U-95
R, U-96
Ucomponents, P-70
adiabaticFlameT, U-99
ansysToFoam, U-93
applyBoundaryLayer, U-92
applyWallFunctionBoundaryConditions, U-92
attachMesh, U-93
autoPatch, U-93
autoRefineMesh, U-94
blockMesh, U-37, U-92, U-141
boxTurb, U-92
ccm26ToFoam, U-93
cfx4ToFoam, U-93, U-160
changeDictionary, U-92
checkMesh, U-93, U-162
chemkinToFoam, U-99
collapseEdges, U-94
combinePatchFaces, U-95
createBaffles, U-93
createPatch, U-93
createTurbulenceFields, U-96
createExternalCoupledPatchGeometry, U-92
datToFoam, U-93
decomposePar, U-82, U-83, U-98
deformedGeom, U-94
dsmcFieldsCalc, U-97
dsmcInitialise, U-92
engineCompRatio, U-97
engineSwirl, U-92
ensight74FoamExec, U-185
ensightFoamReader, U-95
enstrophy, U-95
equilibriumCO, U-99
equilibriumFlameT, U-99
execFlowFunctionObjects, U-97
expandDictionary, U-99
extrude2DMesh, U-92
extrudeMesh, U-92
extrudeToRegionMesh, U-92
faceAgglomerate, U-92
flattenMesh, U-94
flowType, U-95
fluent3DMeshToFoam, U-93
fluentMeshToFoam, U-93, U-160
foamCalc, U-33, U-97
foamDataToFluent, U-95, U-183
foamDebugSwitches, U-99
foamFormatConvert, U-99
foamHelp, U-99
foamInfoExec, U-99
foamListTimes, U-97
foamMeshToFluent, U-93, U-183
foamToEnsightParts, U-95
foamToEnsight, U-95
foamToGMV, U-95
foamToStarMesh, U-93
foamToSurface, U-93
foamToTecplot360, U-95
foamToVTK, U-95
foamUpgradeCyclics, U-92
foamUpgradeFvSolution, U-92
foamyHexMesh, U-92
foamyQuadMesh, U-93
foamyHexMeshBackgroundMesh, U-92
foamyHexMeshSurfaceSimplify, U-92
gambitToFoam, U-93, U-160
gmshToFoam, U-93
ideasToFoam, U-160
ideasUnvToFoam, U-93
insideCells, U-94
kivaToFoam, U-93
mapFields, U-31, U-38, U-42, U-55, U-92, U-168
mdInitialise, U-92
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-93
netgenNeutralToFoam, U-93
objToVTK, U-94
orientFaceZone, U-94
pPrime2, U-96
particleTracks, U-96
patchAverage, U-96
patchIntegrate, U-96
patchSummary, U-99
pdfPlot, U-97
plot3dToFoam, U-93
polyDualMesh, U-94
postChannel, U-97
probeLocations, U-96
ptot, U-97
reconstructParMesh, U-98
reconstructPar, U-86
redistributePar, U-98
refineHexMesh, U-95
refineMesh, U-94
refineWallLayer, U-95
refinementLevel, U-95
removeFaces, U-95
renumberMesh, U-94
rotateMesh, U-94
sammToFoam, U-93
sample, U-96, U-186
scalePoints, U-165
selectCells, U-95
setFields, U-59, U-60, U-92
setSet, U-94
setsToZones, U-94
singleCellMesh, U-94
smapToFoam, U-95
snappyHexMesh, U-93, U-151
splitCells, U-95
splitMeshRegions, U-94
splitMesh, U-94
star3ToFoam, U-93
star4ToFoam, U-93
starToFoam, U-160
steadyParticleTracks, U-96
stitchMesh, U-94
streamFunction, U-95
stressComponents, U-96
subsetMesh, U-94
surfaceLambdaMuSmooth, U-97
surfaceAdd, U-97
surfaceAutoPatch, U-97
surfaceBooleanFeatures, U-97
surfaceCheck, U-97
surfaceClean, U-97
surfaceCoarsen, U-97
surfaceConvert, U-97
surfaceFeatureConvert, U-97
surfaceFeatureExtract, U-97, U-155
surfaceFind, U-97
surfaceHookUp, U-97
surfaceInertia, U-97
surfaceMeshConvertTesting, U-97
surfaceMeshConvert, U-97
surfaceMeshExport, U-98
surfaceMeshImport, U-98
surfaceMeshInfo, U-98
surfaceMeshTriangulate, U-98
surfaceOrient, U-98
surfacePointMerge, U-98
surfaceRedistributePar, U-98
surfaceRefineRedGreen, U-98
surfaceSplitByPatch, U-98
surfaceSplitByTopology, U-98
surfaceSplitNonManifolds, U-98
surfaceSubset, U-98
surfaceToPatch, U-98
surfaceTransformPoints, U-98
temporalInterpolate, U-97
tetgenToFoam, U-93
topoSet, U-94
transformPoints, U-94
uprime, U-95
viewFactorsGen, U-92
vorticity, U-96
vtkUnstructuredToFoam, U-93
wallFunctionTable, U-92
wallGradU, U-96
wallHeatFlux, U-96
wallShearStress, U-96
wdot, U-97
writeCellCentres, U-97
writeMeshObj, U-93
yPlusLES, U-96
yPlusRAS, U-96
zipUpMesh, U-94
utilityFunctionObjects
library, U-100

## V

v2f model, U-104, U-105
`value` keyword, U-21, U-141
`valueFraction` keyword, U-141
van Leer differencing, P-36
vanDriestDelta model, U-105, U-106
`vanLeer`
    keyword entry, U-121
`VCR Controls` menu, U-25, U-173
vector, P-14
    operator, P-25
    unit, P-18
vector class, P-22, U-111
vector product, *see* tensor, vector cross product
vectorField class, P-27
`version` keyword, U-109
`vertices` keyword, U-20, U-144
veryInhomogeneousMixture model, U-102, U-194
`View` menu, U-175
`View Settings`
    menu entry, U-25, U-175
View Settings (Render View) window, U-175
`View Settings...`
    menu entry, U-24
viewFactor
    library, U-102
viewFactorsGen utility, U-92
viscosity
    kinematic, U-21, U-42
volField<Type> template class, P-31
volMesh tools, U-100
vorticity utility, U-96
`vtk`
    keyword entry, U-187
vtkFoam
    library, U-171
vtkPV3Foam
    library, U-171
vtkUnstructuredToFoam utility, U-93

## W

wall
    boundary condition, P-63, P-69, U-58, U-139, U-140
`wall`
    keyword entry, U-140
wallFunctionTable utility, U-92
wallGradU utility, U-96
wallHeatFlux utility, U-96
Wallis
    library, U-103
wallShearStress utility, U-96
wclean script/alias, U-76
wdot utility, U-97
wedge
    boundary condition, U-135, U-140, U-151
`wedge`
    keyword entry, U-140
window
    *Color Legend*, U-27
    *Options*, U-176
    *Pipeline Browser*, U-24, U-172
    *Render View*, U-176
    *Seed*, U-177
    *View Settings (Render View)*, U-175
window panel
    *Animations*, U-176
    *Annotation*, U-25, U-176
    *Charts*, U-176
    *Color Legend*, U-174
    *Color Scale*, U-174
    *Colors*, U-176
    *Display*, U-24, U-25, U-172, U-173
    *General*, U-175, U-176
    *Information*, U-172
    *Lights*, U-176
    *Mesh Parts*, U-24
    *Properties*, U-25, U-172, U-173
    *Render View*, U-176
    *Style*, U-24, U-175
`Wireframe`
    menu entry, U-175
WM_ARCH
    environment variable, U-76
WM_ARCH_OPTION
    environment variable, U-76
WM_COMPILE_OPTION
    environment variable, U-76
WM_COMPILER
    environment variable, U-76
WM_COMPILER_BIN
    environment variable, U-76
WM_COMPILER_DIR
    environment variable, U-76
WM_COMPILER_LIB
    environment variable, U-76
WM_DIR
    environment variable, U-76
WM_MPLIB
    environment variable, U-76
WM_OPTIONS

environment variable, U-76
WM_PRECISION_OPTION
environment variable, U-76
WM_PROJECT
environment variable, U-76
WM_PROJECT_DIR
environment variable, U-76
WM_PROJECT_INST_DIR
environment variable, U-76
WM_PROJECT_USER_DIR
environment variable, U-76
WM_PROJECT_VERSION
environment variable, U-76
wmake
platforms, U-73
wmake script/alias, U-71
word class, P-24, P-29
writeCellCentres utility, U-97
writeCompression keyword, U-116
writeControl
keyword entry, U-116
writeControl keyword, U-22, U-62, U-116
writeFormat keyword, U-55, U-116
writeInterval
keyword entry, U-181
writeInterval keyword, U-22, U-32, U-116
writeMeshObj utility, U-93
writeNow
keyword entry, U-115
writePrecision keyword, U-116

## X

x
keyword entry, U-189
XiFoam solver, U-90
xmgr
keyword entry, U-117, U-187
xyz
keyword entry, U-189

## Y

y
keyword entry, U-189
yPlusLES utility, U-96
yPlusRAS utility, U-96

## Z

z
keyword entry, U-189
zeroGradient
boundary condition, U-141
zipUpMesh utility, U-94