



*The Open Source CFD Toolbox*

## Programmer's Guide

Version 3.0.1  
13th December 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 L<sup>A</sup>T<sub>E</sub>X.

## 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, MERCHANTABILITY, 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

<b>Copyright Notice</b>	<b>P-2</b>
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
<b>Trademarks</b>	<b>P-7</b>
<b>Contents</b>	<b>P-9</b>
<b>1 Tensor mathematics</b>	<b>P-13</b>
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
<b>2 Discretisation procedures</b>	<b>P-25</b>
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 <code>geometricField</code> 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
<b>3</b>	<b>Examples of the use of OpenFOAM</b>	<b>P-43</b>
3.1	Flow around a cylinder . . . . .	P-43
3.1.1	Problem specification . . . . .	P-44
3.1.2	Note on <code>potentialFoam</code> . . . . .	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

Contents	P-11
3.5.1 Problem specification . . . . .	P-67
3.5.2 Mesh generation . . . . .	P-69
3.5.3 Running the case . . . . .	P-70
<b>Index</b>	<b>P-73</b>



# 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.

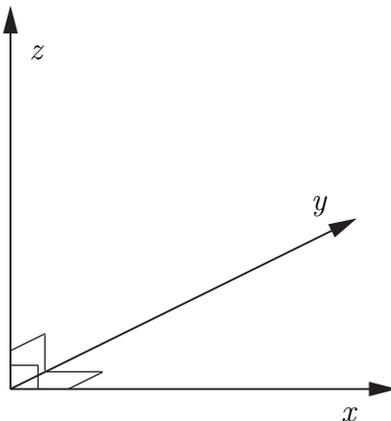


Figure 1.1: Right handed axes

### 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:

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

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

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

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.*

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

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

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

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,

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

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

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

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

### 1.3.1 The inner product

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

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

$$s = a_i b_i = a_1 b_1 + a_2 b_2 + a_3 b_3 \quad (1.7)$$

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

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

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

$$b_i = a_j T_{ji} = \begin{pmatrix} a_1 T_{11} + a_2 T_{21} + a_3 T_{31} \\ a_1 T_{12} + a_2 T_{22} + a_3 T_{32} \\ a_1 T_{13} + a_2 T_{23} + a_3 T_{33} \end{pmatrix} \quad (1.9)$$

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

$$P_{ij} = T_{ik} S_{kj} \quad (1.10)$$

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

- 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

$$T_{ij} = a_k P_{kij} \quad (1.11)$$

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

$$T_{ij} = P_{ijk} a_k \quad (1.12)$$

- 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

$$Q_{ijk} = T_{il} P_{ljk} \quad (1.13)$$

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

$$Q_{ijk} = P_{ijl} T_{lk} \quad (1.14)$$

### 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

$$s = T_{ij} S_{ij} = T_{11} S_{11} + T_{12} S_{12} + T_{13} S_{13} + T_{21} S_{21} + T_{22} S_{22} + T_{23} S_{23} + T_{31} S_{31} + T_{32} S_{32} + T_{33} S_{33} \quad (1.15)$$

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

$$a_i = T_{jk} P_{jki} \quad (1.16)$$

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

$$a_i = P_{ijk} T_{jk} \quad (1.17)$$

### 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  $s = \mathbf{P} \cdot \mathbf{Q}$  which can be evaluated as the sum of the 27 products of the tensor components

$$s = P_{ijk} Q_{ijk} \quad (1.18)$$

### 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  $\mathbf{T} = \mathbf{ab} = (\mathbf{ba})^T$  whose components are evaluated as:

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

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

$$P_{ijk} = a_i T_{jk} \quad (1.20)$$

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

$$P_{ijk} = T_{ij} a_k \quad (1.21)$$

### 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

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

where the *permutation symbol* is defined by

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

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  $\mathbf{a}^2 = \mathbf{a}\mathbf{a}$ .

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

**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

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

### 1.3.7 Geometric transformation and the identity tensor

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

$$\mathbf{a}^* = \mathbf{T} \cdot \mathbf{a} \quad (1.25)$$

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

$$\mathbf{S}^* = \mathbf{T} \cdot \mathbf{S} \cdot \mathbf{T}^T \quad (1.26)$$

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

$$\mathbf{a} = \mathbf{I} \cdot \mathbf{a} \quad (1.27)$$

and therefore

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

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  $\mathbf{a}$ .

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

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

$$e_{ijk}e_{irs} = \delta_{jr}\delta_{ks} - \delta_{js}\delta_{kr} \quad (1.30)$$

### 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  $\mathbf{T}^T = T_{ji}$  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.  $\mathbf{T} = \mathbf{T}^T$ . A skew or antisymmetric tensor has  $\mathbf{T} = -\mathbf{T}^T$  which intuitively implies that  $T_{11} = T_{22} = T_{33} = 0$ . Every second order tensor can be decomposed into symmetric and skew parts by

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

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

$$\text{tr } \mathbf{T} = T_{11} + T_{22} + T_{33} \quad (1.32)$$

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

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

**Deviatoric and hydrostatic tensors** Every second rank tensor  $\mathbf{T}$  can be decomposed into a deviatoric component, for which  $\text{tr } \mathbf{T} = 0$  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:

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

**Determinant** The determinant of a second rank tensor is evaluated by

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

**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

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

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

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

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

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

**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	<code>x()</code> , <code>y()</code> , <code>z()</code>
2	Tensor	tensor	<code>xx()</code> , <code>xy()</code> , <code>xz()</code> ...

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		$\mathbf{a} + \mathbf{b}$	<code>a + b</code>
Subtraction		$\mathbf{a} - \mathbf{b}$	<code>a - b</code>
Scalar multiplication		$s\mathbf{a}$	<code>s * a</code>
Scalar division		$\mathbf{a}/s$	<code>a / s</code>
Outer product	rank $\mathbf{a}, \mathbf{b} \geq 1$	$\mathbf{a}\mathbf{b}$	<code>a * b</code>
Inner product	rank $\mathbf{a}, \mathbf{b} \geq 1$	$\mathbf{a} \cdot \mathbf{b}$	<code>a &amp; b</code>
Double inner product	rank $\mathbf{a}, \mathbf{b} \geq 2$	$\mathbf{a} : \mathbf{b}$	<code>a &amp;&amp; b</code>
Cross product	rank $\mathbf{a}, \mathbf{b} = 1$	$\mathbf{a} \times \mathbf{b}$	<code>a ^ b</code>
Square		$\mathbf{a}^2$	<code>sqr(a)</code>
Magnitude squared		$ \mathbf{a} ^2$	<code>magSqr(a)</code>
Magnitude		$ \mathbf{a} $	<code>mag(a)</code>
Power	$n = 0, 1, \dots, 4$	$\mathbf{a}^n$	<code>pow(a,n)</code>
Component average	$i = 1, \dots, N$	$\bar{a}_i$	<code>cmptAv(a)</code>
Component maximum	$i = 1, \dots, N$	$\max(a_i)$	<code>max(a)</code>
Component minimum	$i = 1, \dots, N$	$\min(a_i)$	<code>min(a)</code>
Scale		$\text{scale}(\mathbf{a}, \mathbf{b})$	<code>scale(a,b)</code>
Geometric transformation	transforms $\mathbf{a}$ using tensor $\mathbf{T}$		<code>transform(T,a)</code>

#### Operations exclusive to tensors of rank 2

*Continued on next page*

*Continued from previous page*

Operation	Comment	Mathematical Description	Description in OpenFOAM
Transpose		$\mathbf{T}^T$	T.T()
Diagonal		diag $\mathbf{T}$	diag(T)
Trace		tr $\mathbf{T}$	tr(T)
Deviatoric component		dev $\mathbf{T}$	dev(T)
Symmetric component		symm $\mathbf{T}$	symm(T)
Skew-symmetric component		skew $\mathbf{T}$	skew(T)
Determinant		det $\mathbf{T}$	det(T)
Cofactors		cof $\mathbf{T}$	cof(T)
Inverse		inv $\mathbf{T}$	inv(T)
Hodge dual		* $\mathbf{T}$	*T
<b>Operations exclusive to scalars</b>			
Sign (boolean)		sgn( $s$ )	sign(s)
Positive (boolean)		$s \geq 0$	pos(s)
Negative (boolean)		$s < 0$	neg(s)
Limit	$n$ scalar	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 <sub>10</sub> $s$	log10(s)
Sine		sin $s$	sin(s)
Cosine		cos $s$	cos(s)
Tangent		tan $s$	tan(s)
Arc sine		asin $s$	asin(s)
Arc cosine		acos $s$	acos(s)
Arc tangent		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		asinh $s$	asinh(s)
Hyperbolic arc cosine		acosh $s$	acosh(s)
Hyperbolic arc tangent		atanh $s$	atanh(s)
Error function		erf $s$	erf(s)
Complement error function		erfc $s$	erfc(s)
Logarithm gamma function		ln $\Gamma s$	lgamma(s)
Type 1 Bessel function of order 0		J <sub>0</sub> $s$	j0(s)
Type 1 Bessel function of order 1		J <sub>1</sub> $s$	j1(s)
Type 2 Bessel function of order 0		Y <sub>0</sub> $s$	y0(s)
Type 2 Bessel function of order 1		Y <sub>1</sub> $s$	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<sup>3</sup>), pressure in Pascals (kg m s<sup>-2</sup>). 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<sup>-2</sup> since the first entry in the `pressureDims` array, 1, corresponds to k<sup>1</sup>, the second entry, -1, corresponds to m<sup>-1</sup> *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

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

# 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  $\partial_i$ :

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

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

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

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

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

### 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

$$\nabla \cdot \mathbf{a} = \partial_i a_i = \frac{\partial a_1}{\partial x_1} + \frac{\partial a_2}{\partial x_2} + \frac{\partial a_3}{\partial x_3} \quad (2.4)$$

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)

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

### 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

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

The curl is related to the gradient by

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

### 2.1.4 Laplacian

The Laplacian is an operation that can be defined mathematically by a combination of the divergence and gradient operators by  $\nabla^2 \equiv \nabla \cdot \nabla$ . 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

$$\nabla^2 \equiv \partial^2 \equiv \frac{\partial^2}{\partial x_1^2} + \frac{\partial^2}{\partial x_2^2} + \frac{\partial^2}{\partial x_3^2} \quad (2.8)$$

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

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

### 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

$$\frac{D\phi}{Dt} = \lim_{\Delta t \rightarrow 0} \frac{\Delta\phi}{\Delta t} \quad (2.10)$$

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:

$$\frac{D\phi}{Dt} = \frac{\partial\phi}{\partial t} + \mathbf{U} \cdot \nabla\phi \quad (2.11)$$

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.

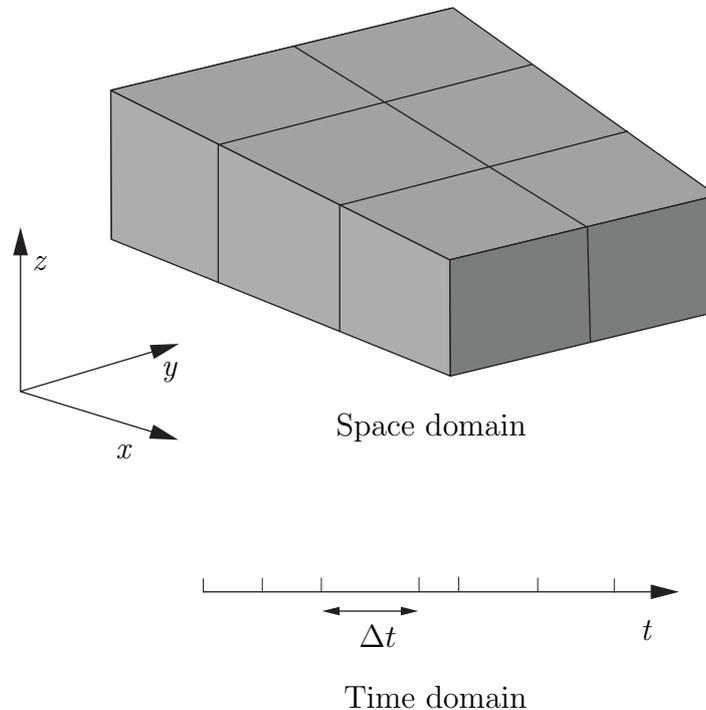


Figure 2.1: Discretisation of the solution domain

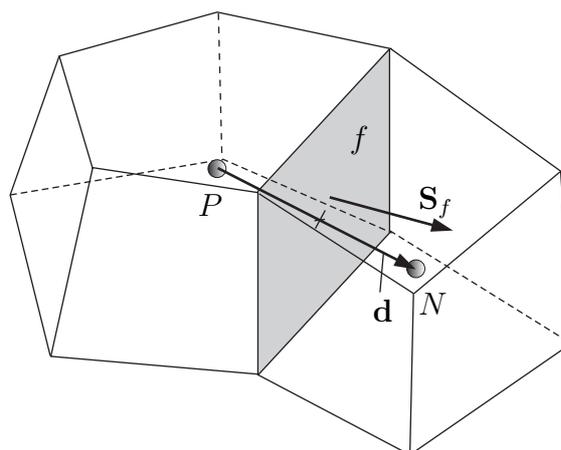


Figure 2.2: Parameters in finite volume discretisation

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.*.

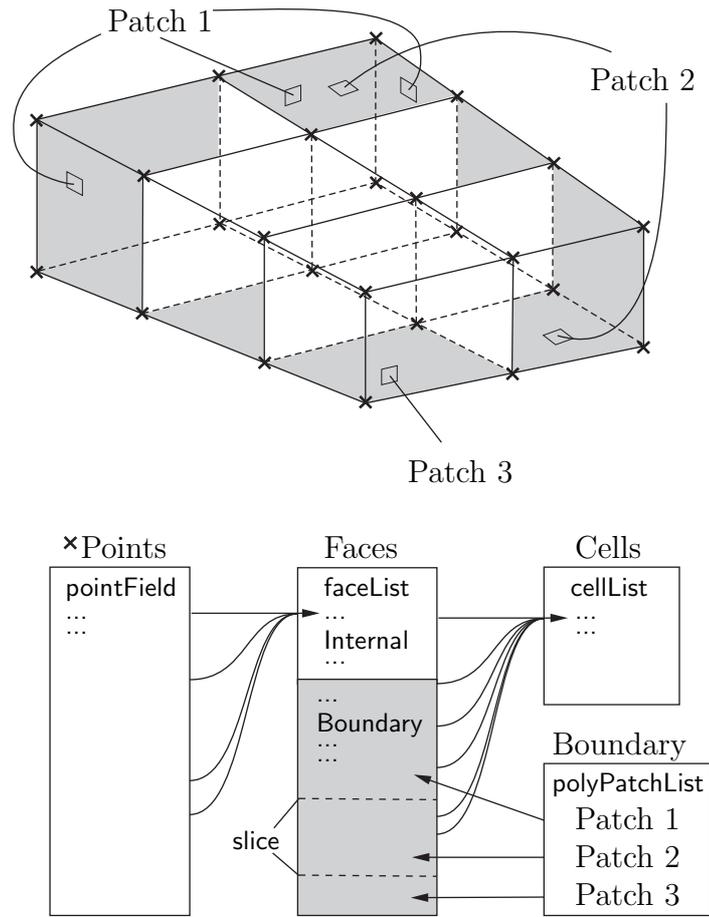


Figure 2.3: Schematic of the basic mesh description used in OpenFOAM

### 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	$V$	<code>v()</code>
surfaceVectorField	Face area vectors	$\mathbf{S}_f$	<code>Sf()</code>
surfaceScalarField	Face area magnitudes	$ \mathbf{S}_f $	<code>magSf()</code>
volVectorField	Cell centres	$\mathbf{C}$	<code>c()</code>
surfaceVectorField	Face centres	$\mathbf{C}_f$	<code>Cf()</code>
surfaceScalarField	Face motion fluxes **	$\phi_g$	<code>phi()</code>

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<sup>1</sup>.

## 2.4 Equation discretisation

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

$$[A] [x] = [b] \quad (2.12)$$

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.

<sup>1</sup>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`.

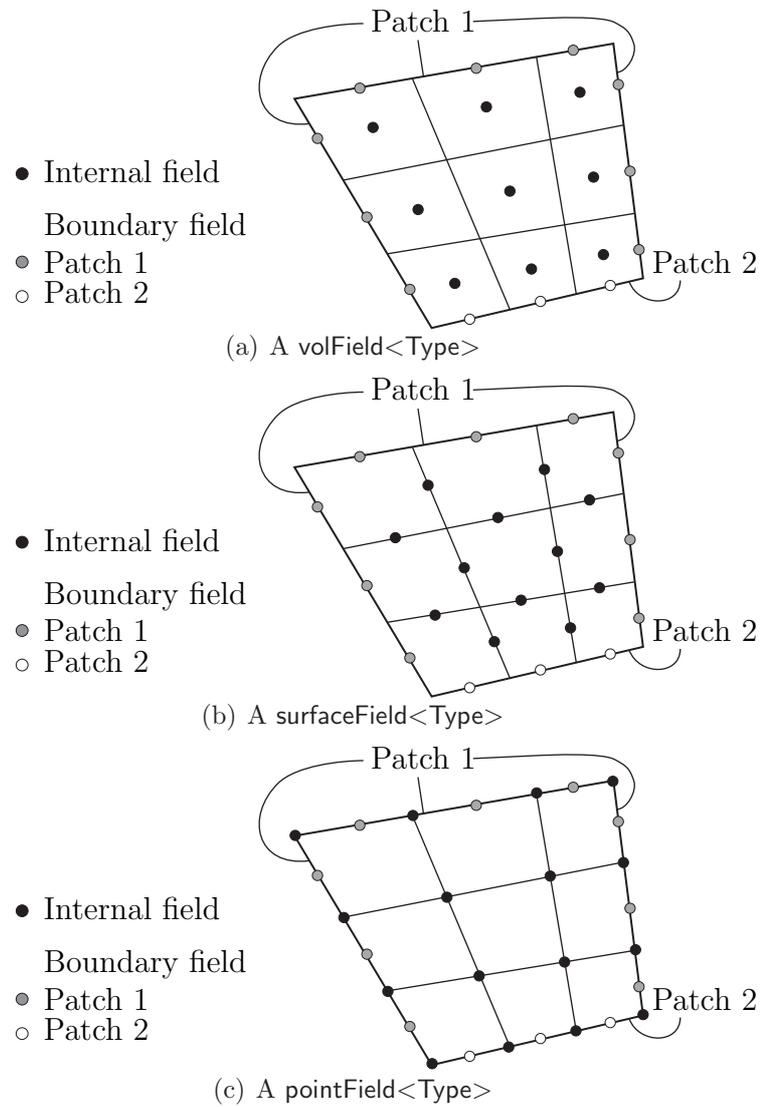


Figure 2.4: Types of `geometricField<Type>` defined on a mesh with 2 boundary patches (in 2 dimensions for simplicity)



[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.*  $\nabla^2$ ,  $\nabla \cdot$  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.

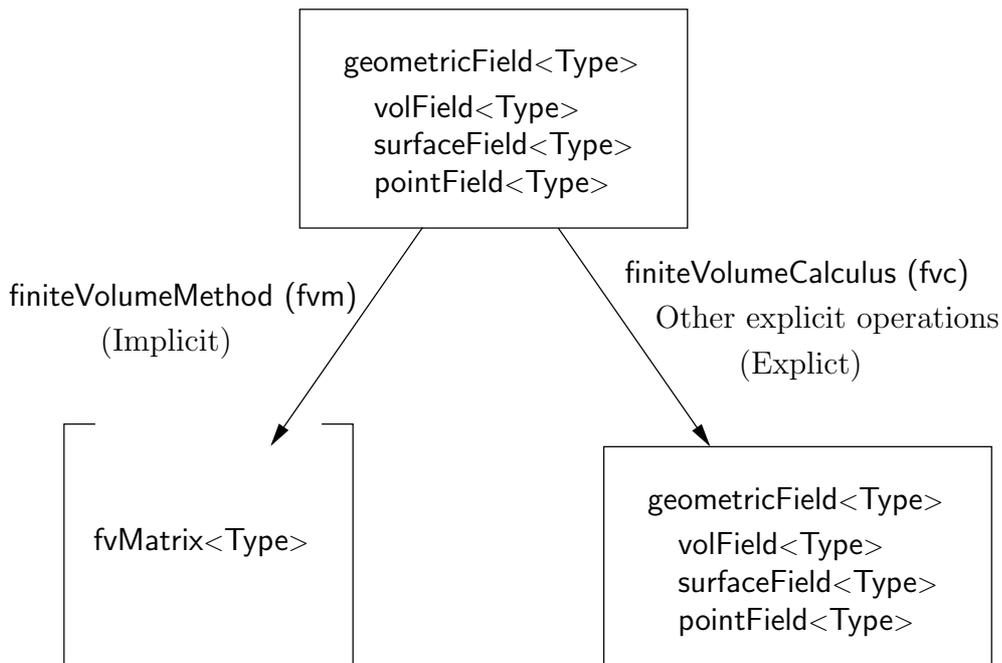


Figure 2.6: A `geometricField<Type>` and its operators

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	$\nabla^2\phi$ $\nabla \cdot \Gamma \nabla \phi$	laplacian(phi) laplacian(Gamma, phi)
Time derivative	Imp/Exp	$\frac{\partial \phi}{\partial t}$ $\frac{\partial \rho \phi}{\partial t}$	ddt(phi) 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	$\nabla \cdot (\psi)$ $\nabla \cdot (\psi \phi)$	div(psi, scheme)* div(psi, phi, word)* div(psi, phi)
Divergence	Exp	$\nabla \cdot \chi$	div(chi)
Gradient	Exp	$\nabla \chi$ $\nabla \phi$	grad(chi) gGrad(phi) lsGrad(phi) snGrad(phi) snGradCorrection(phi)
Grad-grad squared	Exp	$ \nabla \nabla \phi ^2$	sqrGradGrad(phi)
Curl	Exp	$\nabla \times \phi$	curl(phi)
Source	Imp Imp/Exp†	$\rho \phi$	Sp(rho, phi) 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  $\nabla \cdot \phi$ , 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:

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

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  $\mathbf{S}_f$ :

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

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:

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

The face field  $\phi_f$  can be evaluated using a variety of schemes:

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

$$\phi_f = f_x \phi_P + (1 - f_x) \phi_N \quad (2.17)$$

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

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

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

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

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:

$$\frac{\partial}{\partial t} \int_V \rho \phi \, dV \quad (2.20)$$

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:

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

**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`:

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

### 2.4.4 Second time derivative

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

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

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:

$$\int_V \nabla \cdot \phi \, dV = \int_S d\mathbf{S} \cdot \phi = \sum_f \mathbf{S}_f \cdot \phi_f \quad (2.24)$$

The `fv::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.* `fv::gGrad`, `fv::lsGrad` *etc.*, or by using the `fv::grad` function combined with the appropriate `gradScheme` keyword in an input file

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

$$\int_V \nabla \phi \, dV = \int_S d\mathbf{S} \phi = \sum_f \mathbf{S}_f \phi_f \quad (2.25)$$

As with the `fv::div` function, the Gaussian integration `fv::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 `fv::grad` function with `timeScheme leastSquares` or directly using the `fv::lsGrad` function. The discretisation is performed as by first calculating the tensor  $\mathbf{G}$  at every point  $P$  by summing over neighbours  $N$ :

$$\mathbf{G} = \sum_N w_N^2 \mathbf{d}\mathbf{d} \quad (2.26)$$

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:

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

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

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

This gradient is called by the function `fv::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 `fv::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

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

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

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

The second term can be expressed as

$$\int_t^{t+\Delta t} \left[ \int_V \mathcal{A}\phi \, dV \right] dt = \int_t^{t+\Delta t} \mathcal{A}^* \phi \, dt \quad (2.32)$$

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  $\phi^n$ .

$$\int_t^{t+\Delta t} \mathcal{A}^* \phi \, dt = \mathcal{A}^* \phi^n \Delta t \quad (2.33)$$

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  $\phi^o$ .

$$\int_t^{t+\Delta t} \mathcal{A}^* \phi \, dt = \mathcal{A}^* \phi^o \Delta t \quad (2.34)$$

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

$$Co = \frac{\mathbf{U}_f \cdot \mathbf{d}}{|\mathbf{d}|^2 \Delta t} \quad (2.35)$$

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  $\phi^o$ .

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

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

$$\frac{\partial \phi}{\partial t} = \kappa \nabla^2 \phi \quad (2.37)$$

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,

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

**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

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

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

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

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

$$\mathbf{S}_f \cdot (\nabla\phi)_f = |S_f| g_b \quad (2.41)$$

### 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.

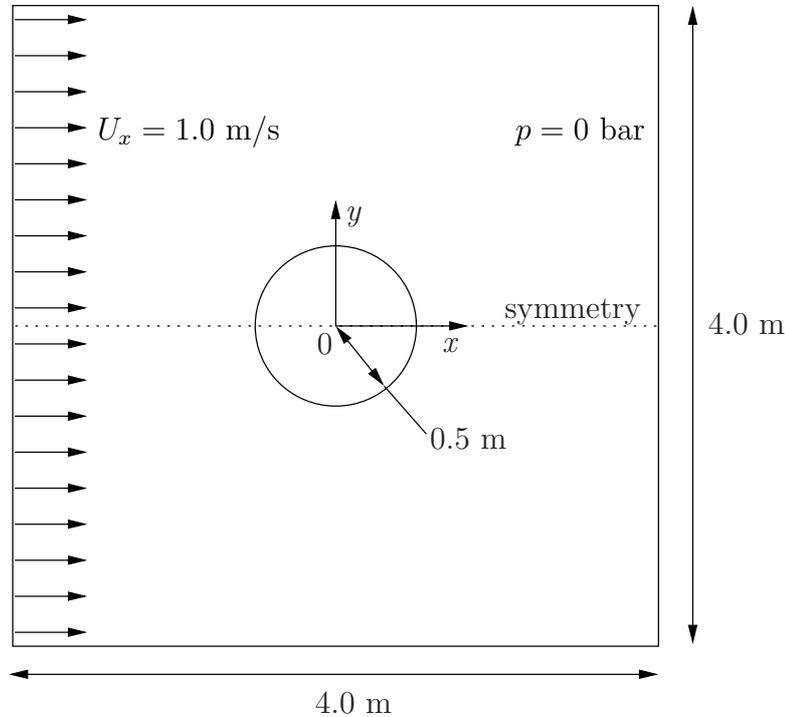


Figure 3.1: Geometry of flow round a cylinder

#### Governing equations

- Mass continuity for an incompressible fluid

$$\nabla \cdot \mathbf{U} = 0 \quad (3.1)$$

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

$$\nabla^2 p = 0 \quad (3.2)$$

#### Boundary conditions

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

**Initial conditions**  $U = 0 \text{ m/s}$ ,  $p = 0 \text{ Pa}$  — 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  $\mathbf{U}$  field for a problem. When running certain cases, this can be useful for avoiding instabilities due to the initial field being unstable. In short, `potentialFoam` creates a conservative field from a non-conservative initial field supplied by the user.

### 3.1.3 Mesh generation

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

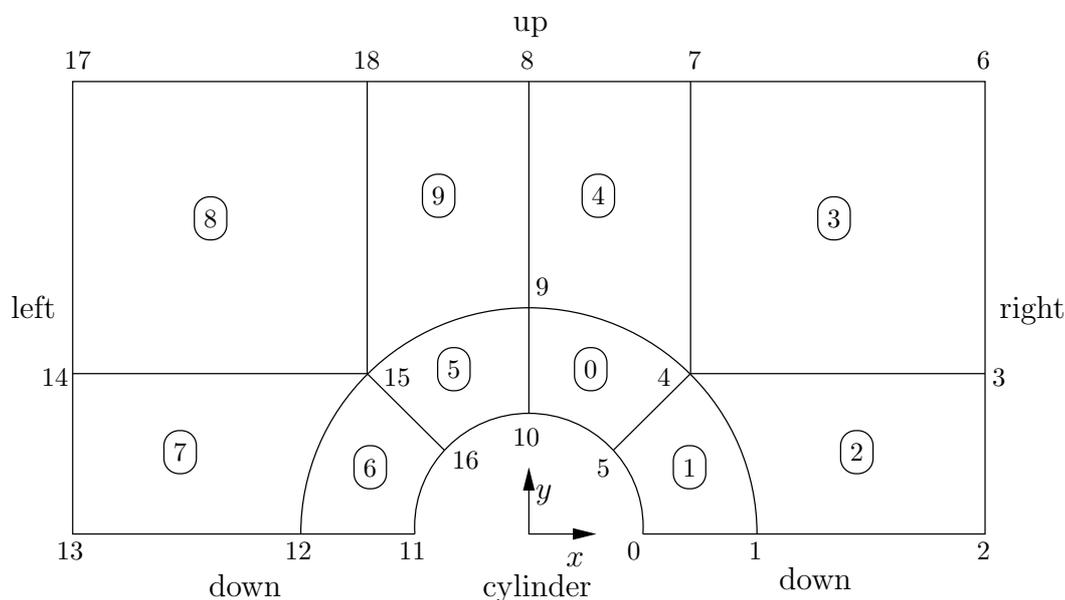


Figure 3.2: Blocks in cylinder geometry

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

```

1  /*----- C++ -----*/
2  |=====|
3  | \ \ \ / | F i e l d | OpenFOAM: The Open Source CFD Toolbox
4  | \ \ \ / | O p e r a t i o n | Version: 3.0.1
5  | \ \ \ / | A n d | Web: www.OpenFOAM.org
6  | \ \ \ / | M a n i p u l a t i o n |
7  /*-----*/
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;

```



```

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  $\mathbf{U}$  is small along a plane coinciding with our boundary. We therefore impose the condition that the normal component is zero, *i.e.* specify the boundary as a `symmetryPlane`, thereby ensuring that the comparison with the analytical is reasonable.

### 3.1.5 Running the case

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

```

1  /*-----* C++ -*-----*\
2  |=====|
3  | \\\ /  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
4  |  \\  O p e r a t i o n  | Version:  3.0.1
5  |   \\  A n d             | Web:      www.OpenFOAM.org
6  |    \\  M a n i p u l a t i o n  |
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      potentialFoam;
19
20 startFrom        startTime;
21
22 startTime        0;
23
24 stopAt           endTime;
25
26 endTime          1;
27
28 deltaT           1;
29
30 writeControl     timeStep;
31
32 writeInterval    1;
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 functions
49 {
50     difference
51     {
52         // Load the library containing the 'coded' functionObject
53         functionObjectLibs ("libutilityFunctionObjects.so");
54         type coded;
55         // Name of on-the-fly generated functionObject
56         redirectType error;
57         code

```

```

58     #f
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***

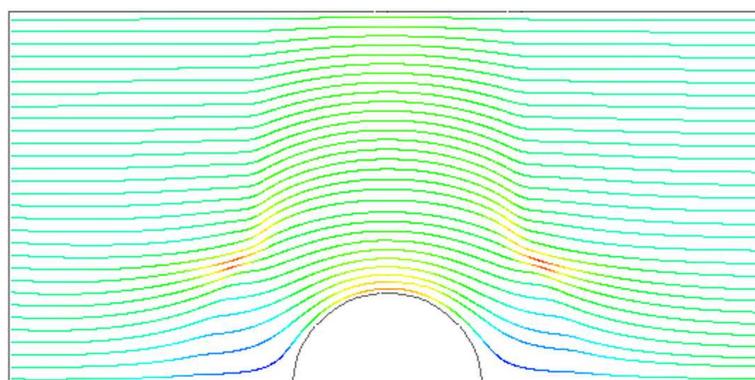
- Mass continuity for incompressible flow

$$\nabla \cdot \mathbf{U} = 0 \quad (3.3)$$

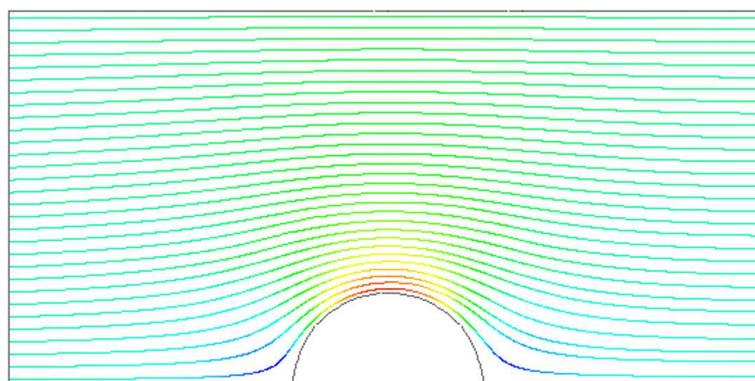
- Steady flow momentum equation

$$\nabla \cdot (\mathbf{UU}) + \nabla \cdot \mathbf{R} = -\nabla p \quad (3.4)$$

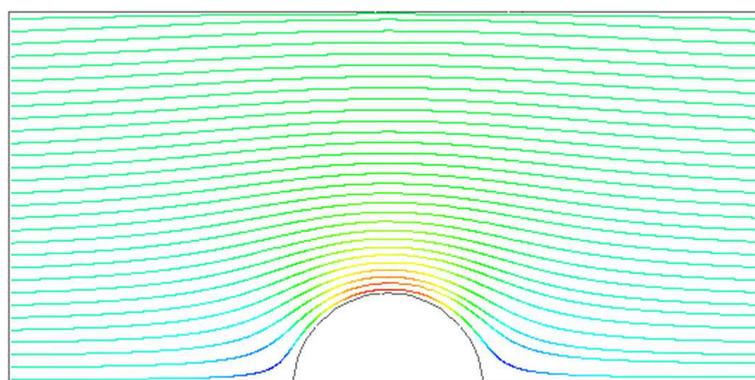
where  $p$  is kinematic pressure and (in slightly over-simplistic terms)  $\mathbf{R} = \nu_{eff} \nabla \mathbf{U}$  is the viscous stress term with an effective kinematic viscosity  $\nu_{eff}$ , calculated from selected transport and turbulence models.



(a) With no non-orthogonal correction



(b) With non-orthogonal correction



(c) Analytical solution

Figure 3.3: Streamlines of potential flow

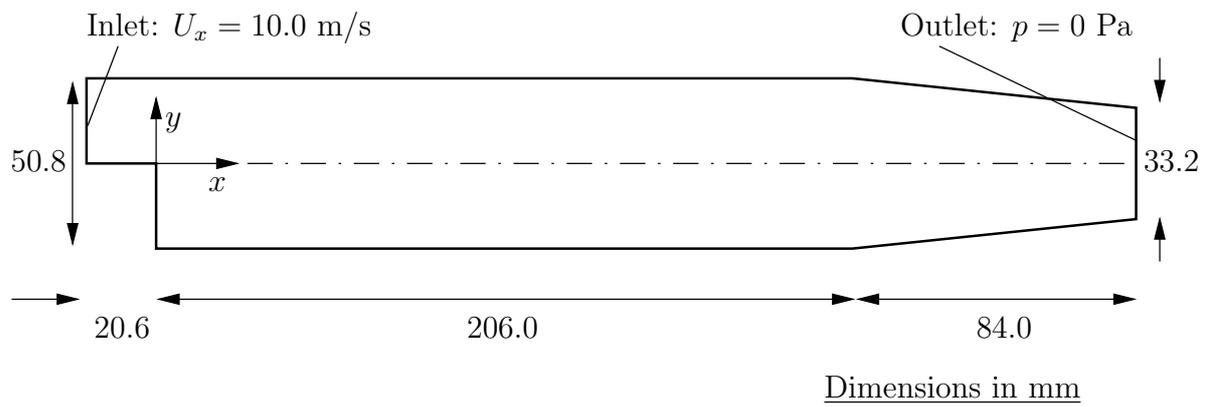


Figure 3.4: Geometry of backward-facing step

**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  $\nu = \mu/\rho = 18.1 \times 10^{-6}/1.293 = 14.0 \mu\text{m}^2/\text{s}$

### 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 i e l d           | OpenFOAM: The Open Source CFD Toolbox
4  |   \ \ /   | O p e r a t i o n   | Version: 3.0.1
5  |    \ \ /   | A n d               | Web: www.OpenFOAM.org
6  |     \ \ /   | M a n i p u l a t i o n |
7  |-----*-----*/
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;
13     object       blockMeshDict;

```

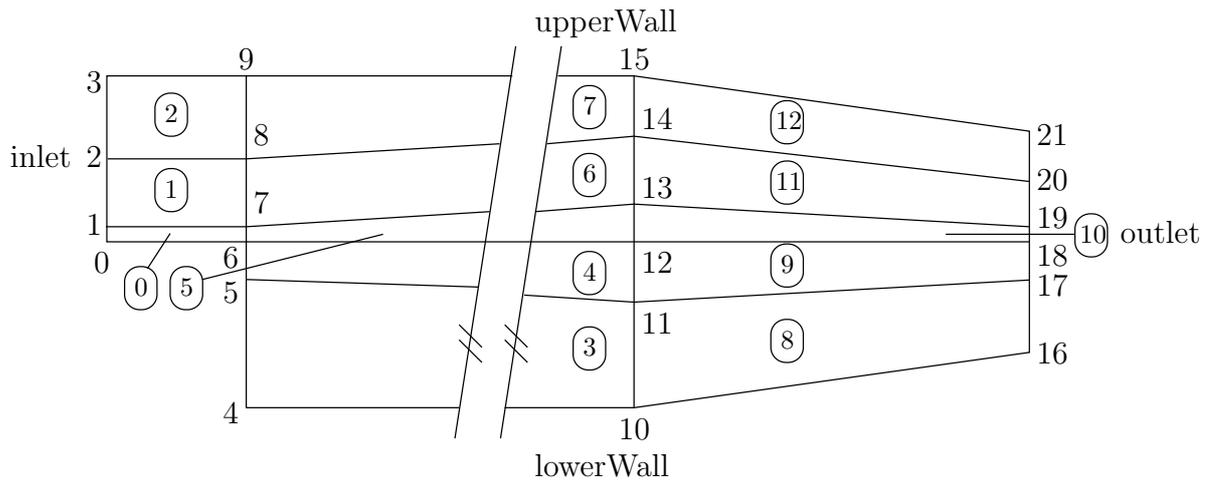


Figure 3.5: Blocks in backward-facing step

```

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  $\mathbf{U}$ , 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  $\mathbf{U}$ ,  $k$  and  $\varepsilon$ .  $\mathbf{U}$  is given in the problem specification, but the values of  $k$  and  $\varepsilon$  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  $\mathbf{U}$  at the inlet. We have

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

and

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

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

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

At the outlet we need only specify the pressure  $p = 0\text{Pa}$ .

### 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

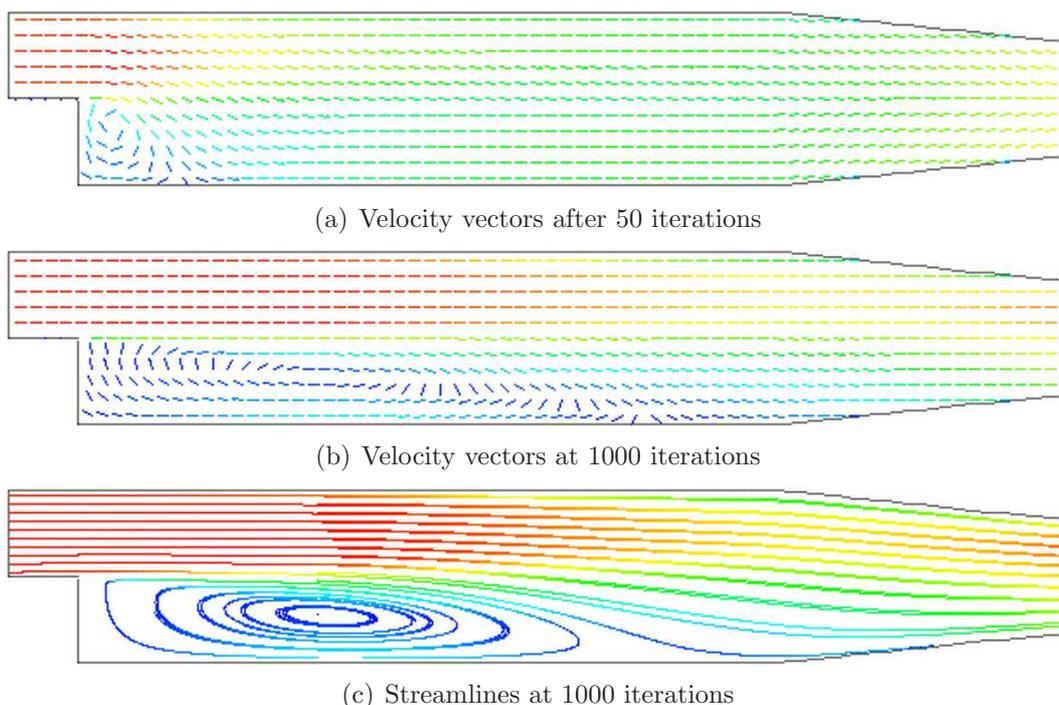


Figure 3.6: Development of a vortex in the backward-facing step.

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

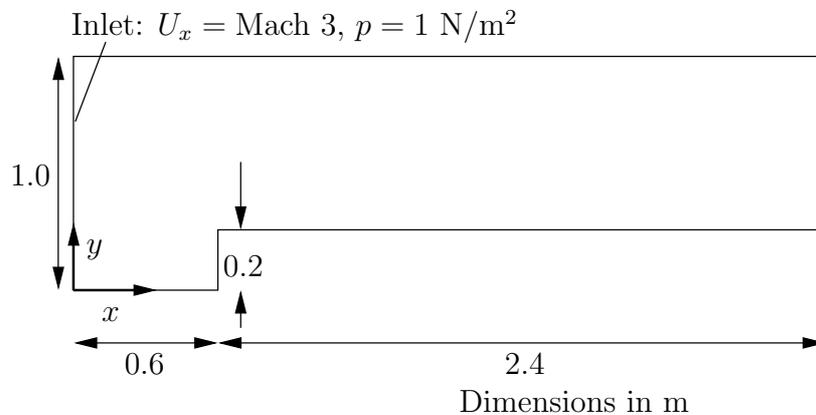


Figure 3.7: Geometry of the forward step geometry

#### Governing equations

- Mass continuity

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

- Ideal gas

$$p = \rho RT \quad (3.9)$$

- Momentum equation for Newtonian fluid

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

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

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

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

#### **Boundary conditions**

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

#### **Transport properties**

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

#### **Thermodynamic properties**

- Specific heat at constant volume  $C_v = 1.78571$  J/kg K
- Gas constant  $R = 0.714286$  J/kg K
- Conductivity  $k = 32.3 \mu\text{W/m K}$

**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  $c = \sqrt{\gamma R T} = 1$  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,  $C_p - C_v = R$ , *i.e.* the ratio of specific heats

$$\gamma = C_p / C_v = \frac{R}{C_v} + 1 \quad (3.12)$$

### **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:

```

1  /*-----*-- C++ -*-----*\
2  |=====|
3  |  \ \  /  | F i e l d      | OpenFOAM: The Open Source CFD Toolbox
4  |  \ \  /  | O p e r a t i o n | Version: 3.0.1
5  |  \ \  /  | A n d           | Web:      www.OpenFOAM.org
6  |  \ \  /  | M a n i p u l a t i o n |
7  \*-----*--\
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;
13     object       blockMeshDict;
14 }
15 // * * * * * //
16
17 convertToMeters 1;
18
19 vertices
20 (
21     (0 0 -0.05)
22     (0.6 0 -0.05)
23     (0 0.2 -0.05)
24     (0.6 0.2 -0.05)
25     (3 0.2 -0.05)
26     (0 1 -0.05)
27     (0.6 1 -0.05)
28     (3 1 -0.05)
29     (0 0 0.05)
30     (0.6 0 0.05)
31     (0 0.2 0.05)
32     (0.6 0.2 0.05)
33     (3 0.2 0.05)
34     (0 1 0.05)
35     (0.6 1 0.05)
36     (3 1 0.05)
37 );
38
39 blocks
40 (
41     hex (0 1 3 2 8 9 11 10) (25 10 1) simpleGrading (1 1 1)
42     hex (2 3 6 5 10 11 14 13) (25 40 1) simpleGrading (1 1 1)
43     hex (3 4 7 6 11 12 15 14) (100 40 1) simpleGrading (1 1 1)
44 );
45
46 edges
47 (
48 );
49
50 boundary
51 (
52     inlet
53     {
54         type patch;
55         faces
56         (
57             (0 8 10 2)
58             (2 10 13 5)
59         );
60     }
61     outlet
62     {
63         type patch;
64         faces
65         (
66             (4 7 15 12)
67         );
68     }
69     bottom
70     {
71         type symmetryPlane;
72         faces
73         (
74             (0 1 9 8)
75         );
76     }
77     top
78     {
79         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.

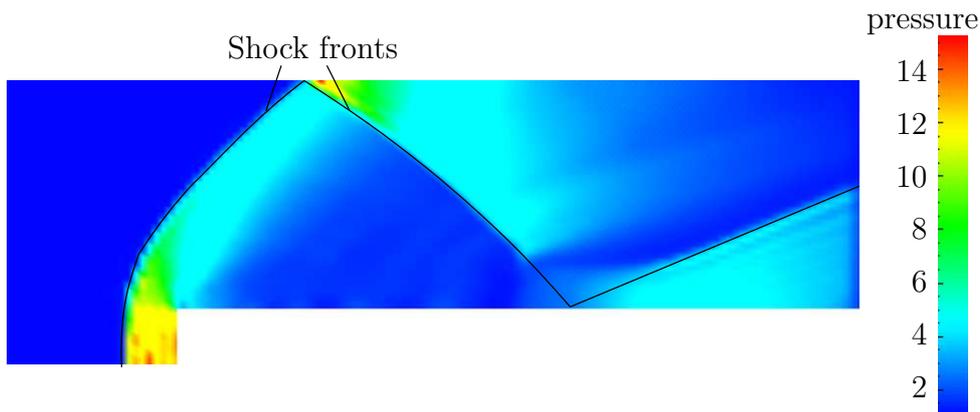


Figure 3.8: Shock fronts in the forward step problem

### 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

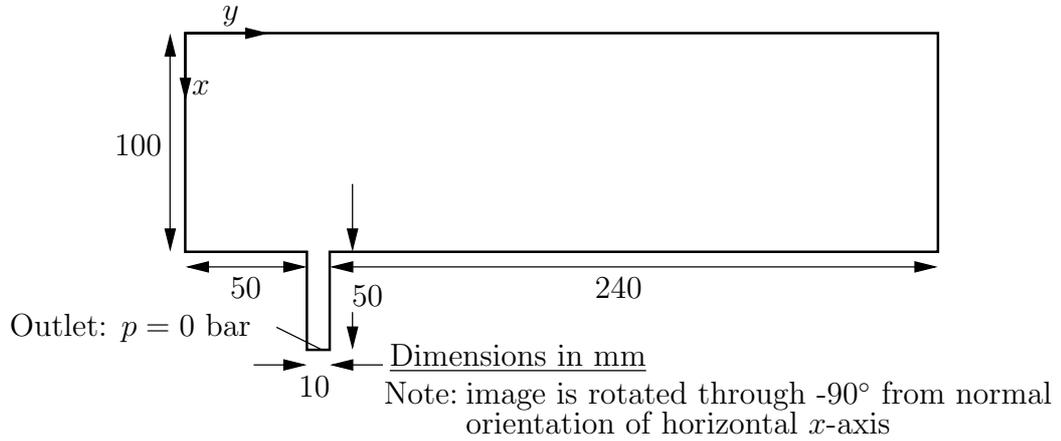


Figure 3.9: Geometry of a tank with outflow pipe

**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

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

- The barotropic relationship

$$\frac{\partial \rho}{\partial p} = \frac{\rho}{K} = \psi \quad (3.14)$$

where  $K$  is the bulk modulus

- Equation 3.14 is linearised as

$$\rho \approx \rho_0 + \psi (p - p_0) \quad (3.15)$$

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

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

**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<sup>3</sup>
- Reference pressure  $p_0 = 1$  bar
- Compressibility of water  $\psi = 4.54 \times 10^{-7}$  s<sup>2</sup>/m<sup>2</sup>

**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:

```

1  /*-----*-----*-----*-----*-----*-----*-----*-----*-----*\
2  |=====| F i e l d | OpenFOAM: The Open Source CFD Toolbox
3  |  \ \  /  | O p e r a t i o n | Version: 3.0.1
4  |  \ \  /  | A n d | Web: www.OpenFOAM.org
5  |  \ \  /  | M a n i p u l a t i o n |
6  |  \ \  /  |-----*-----*-----*-----*-----*-----*-----*\
7  \*-----*-----*-----*-----*-----*-----*-----*-----*-----*/
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;
13     object       blockMeshDict;
14 }
15 // * * * * *
16
17 convertToMeters 0.1;
18
19 vertices
20 (
21     (0 0 -0.1)
22     (1 0 -0.1)
23     (0 0.5 -0.1)
24     (1 0.5 -0.1)
25     (1.5 0.5 -0.1)
26     (0 0.6 -0.1)
27     (1 0.6 -0.1)
28     (1.5 0.6 -0.1)
29     (0 3 -0.1)
30     (1 3 -0.1)
31     (0 0 0.1)
32     (1 0 0.1)
33     (0 0.5 0.1)
34     (1 0.5 0.1)
35     (1.5 0.5 0.1)
36     (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:

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

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

$$Co = \frac{U \Delta t}{\Delta x} \quad (3.18)$$

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 i e l d      | OpenFOAM: The Open Source CFD Toolbox
4  |  \\ /  O p e r a t i o n | Version: 3.0.1
5  |   \\ /  A n d         | Web: www.OpenFOAM.org
6  |    \\ /  M a n i p u l a t i o n |
7  |-----*/
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;
13     location     "system";
14     object       controlDict;
15 }
16 // * * * * *
17 application     sonicLiquidFoam;
18 startFrom       startTime;
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

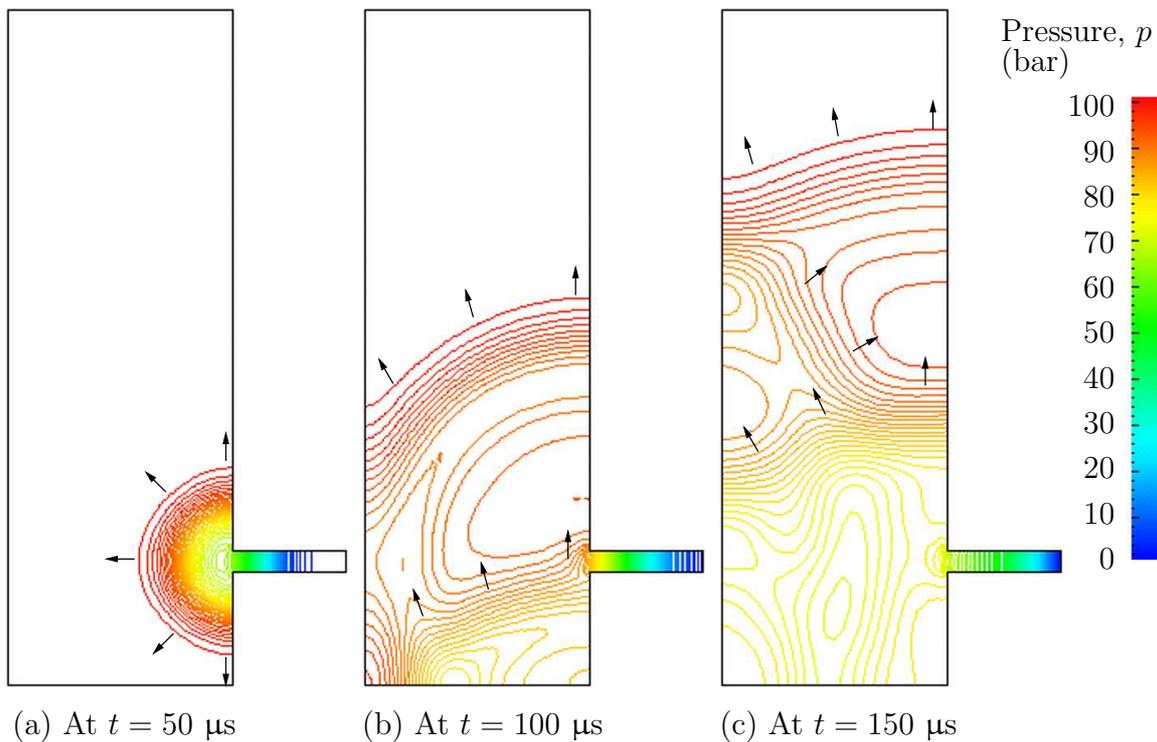


Figure 3.10: Propagation of pressure waves

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.

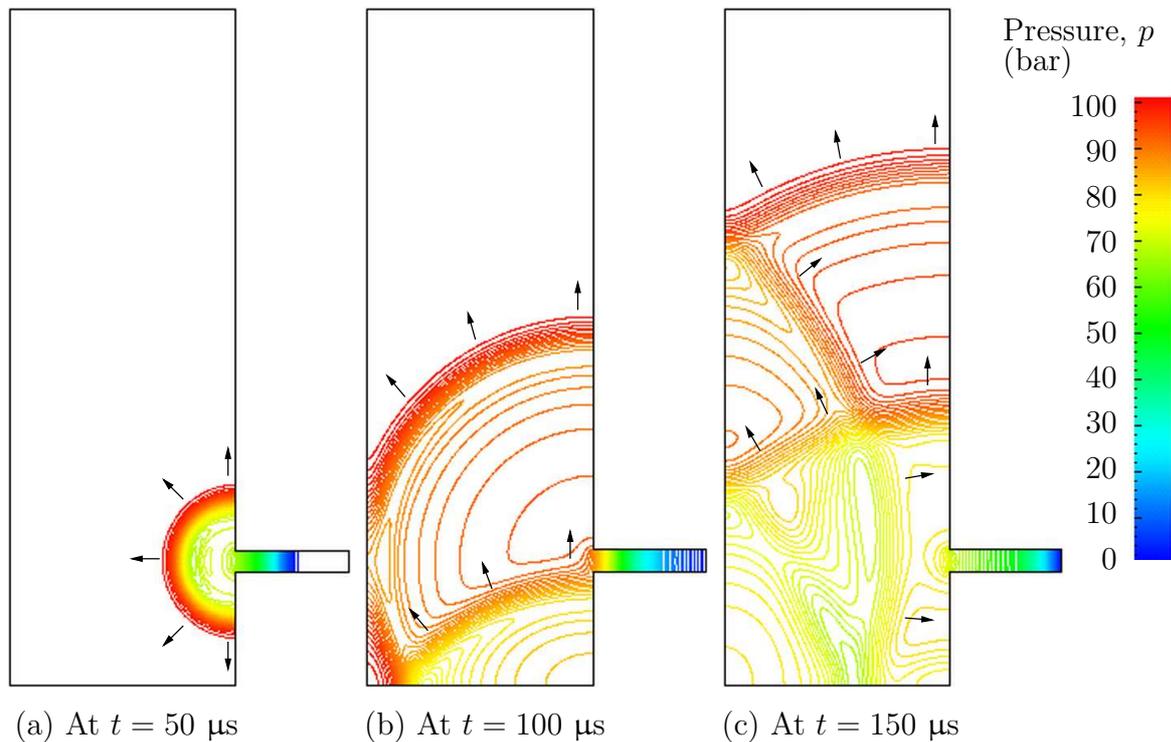


Figure 3.11: Propagation of pressure waves with refined mesh

### 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**

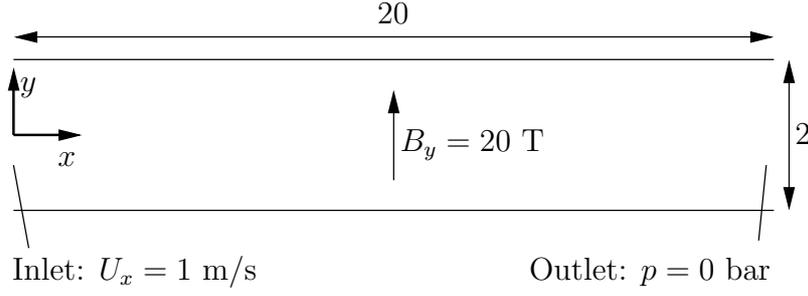


Figure 3.12: Geometry of the Hartmann problem

- Mass continuity for incompressible fluid

$$\nabla \cdot \mathbf{U} = 0 \quad (3.19)$$

- Momentum equation for incompressible fluid

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

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

- Maxwell's equations

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \quad (3.21)$$

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

$$\nabla \cdot \mathbf{B} = 0 \quad (3.22)$$

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

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

$$\nabla \cdot \mathbf{J} = 0 \quad (3.24)$$

- Constitutive law

$$\mathbf{B} = \mu\mathbf{H} \quad (3.25)$$

- Ohm's law

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

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

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

**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**  $\mathbf{U} = 0$  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  $\sigma = 1$  ( $\Omega$  m)<sup>-1</sup>
- Permeability  $\mu = 1$  H/m

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

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

**3.5.2 Mesh generation**

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

```

1  /*-----* C++ *-----*/
2  |=====|
3  | \\\ / | F i e l d | OpenFOAM: The Open Source CFD Toolbox
4  | \\\ / | O p e r a t i o n | Version: 3.0.1
5  | \\\ / | A n d | Web: www.OpenFOAM.org
6  | \\\ / | M a n i p u l a t i o n |
7  /*-----*/
8  FoamFile
9  {
10     version      2.0;
11     format       ascii;
12     class        dictionary;
13     object       blockMeshDict;
14 }
15 // * * * * * //
16
17 convertToMeters 1;
18
19 vertices
20 (
21     (0 -1 0)
22     (20 -1 0)
23     (20 1 0)
24     (0 1 0)
25     (0 -1 0.1)
26     (20 -1 0.1)
27     (20 1 0.1)
28     (0 1 0.1)
29 );
30
31 blocks
32 (
33     hex (0 1 2 3 4 5 6 7) (100 40 1) simpleGrading (1 1 1)
34 );
35
36 edges

```

```

37  (
38  );
39
40  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  $\mathbf{U}$  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

$$M = BL\sqrt{\frac{\sigma}{\rho\nu}} \quad (3.28)$$

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  $\mathbf{B}$  to 1 T and re-run the code and `Ucomponents`. In this case,  $M = 1$  and the electromagnetic body forces no longer dominate.

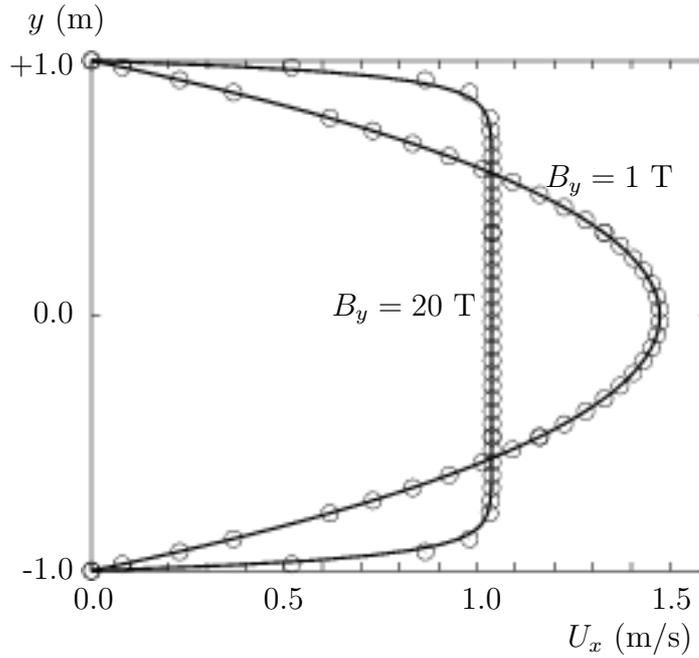


Figure 3.13: Velocity profile in the Hartmann problem for  $B_y = 1$  T and  $B_y = 20$  T.

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:

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

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-78
  - //
    - C++ syntax, U-78
    - OpenFOAM file syntax, U-106
  - # include
    - C++ syntax, U-72, U-78
  - &
    - 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-106
  - 1-dimensional mesh, U-133
  - 1D mesh, U-133
  - 2-dimensional mesh, U-133
  - 2D mesh, U-133
- ## Numbers
- 0 directory, U-106
- ## A
- access functions, P-21
  - addLayersControls keyword, U-150
  - adiabaticFlameT utility, U-97
  - adiabaticPerfectFluid model, U-101, U-195
  - adjointShapeOptimizationFoam solver, U-86
  - adjustableRunTime
    - keyword entry, U-62, U-114
  - adjustTimeStep keyword, U-62, U-115
  - agglomerator keyword, U-125
  - algorithms tools, U-98
  - alphaContactAngle
    - boundary condition, U-59
  - analytical solution, P-43
  - Animations window panel, U-174
  - anisotropicFilter model, U-103
  - Annotation window panel, U-24
  - ansysToFoam utility, U-91
  - APIfunctions model, U-102
  - applications, U-69
  - Apply button, U-170, U-174
  - applyBoundaryLayer utility, U-90
  - applyWallFunctionBoundaryConditions utility, U-90
  - arbitrarily unstructured, P-29
  - arc
    - keyword entry, U-142
  - arc keyword, U-141
  - As keyword, U-193
  - ascii
    - keyword entry, U-114
  - attachMesh utility, U-92
  - Auto Accept button, U-174
  - autoMesh
    - library, U-99
  - autoPatch utility, U-92
  - autoRefineMesh utility, U-93
  - axes
    - right-handed, U-140
    - right-handed rectangular Cartesian, P-13, U-18
  - axi-symmetric cases, U-137, U-148
  - axi-symmetric mesh, U-133

**B**

- background
  - process, U-24, U-81
- backward
  - keyword entry, U-122
- Backward differencing, P-37
- barotropicCompressibilityModels
  - library, U-101
- basicMultiComponentMixture model, U-100
- basicSolidThermo
  - library, U-102
- basicThermophysicalModels
  - library, U-100
- binary
  - keyword entry, U-114
- BirdCarreau model, U-104
- blended differencing, P-36
- block
  - expansion ratio, U-143
- block keyword, U-141
- blocking
  - keyword entry, U-80
- blockMesh
  - library, U-99
- blockMesh solver, P-45
- blockMesh utility, U-38, U-91, U-140
- blockMesh executable
  - vertex numbering, U-143
- blockMeshDict*
  - dictionary, U-18, U-20, U-36, U-49, U-140, U-149
- blocks keyword, U-20, U-31, U-142
- boundaries, U-133
- boundary, U-133
- boundary*
  - dictionary, U-132, U-140
- boundary keyword, U-145
- boundary condition
  - alphaContactAngle, U-59
  - buoyantPressure, U-139
  - calculated, U-138
  - cyclic, U-137, U-146
  - directionMixed, U-138
  - empty, P-63, P-69, U-18, U-133, U-137
  - fixedGradient, U-138
  - fixedValue, U-138
  - fluxCorrectedVelocity, U-139
  - inlet, P-69
  - inletOutlet, U-139
  - mixed, U-138
  - movingWallVelocity, U-139
  - outlet, P-69
  - outletInlet, U-139
  - partialSlip, U-139
  - patch, U-136
  - pressureDirectedInletVelocity, U-139
  - pressureInletVelocity, U-139
  - pressureOutlet, P-63
  - pressureTransmissive, U-139
  - processor, U-138
  - setup, U-20
  - slip, U-139
  - supersonicFreeStream, U-139
  - surfaceNormalFixedValue, U-139
  - symmetryPlane, P-63, U-137
  - totalPressure, U-139
  - turbulentInlet, U-139
  - wall, U-41
  - wall, P-63, P-69, U-59, U-137
  - wedge, U-133, U-137, U-148
  - zeroGradient, U-138
- 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-110
- boundaryFoam solver, U-86
- bounded
  - keyword entry, U-120, U-121
- boxToCell keyword, U-60
- boxTurb utility, U-90
- breaking of a dam, U-56
- BSpline
  - keyword entry, U-142
- buoyantBoussinesqPimpleFoam solver, U-88
- buoyantBoussinesqSimpleFoam solver, U-88
- buoyantPimpleFoam solver, U-88
- buoyantPressure
  - boundary condition, U-139
- buoyantSimpleFoam solver, U-88
- burntProducts keyword, U-193
- button
  - Apply, U-170, U-174
  - Auto Accept, U-174
  - Camera Parallel Projection, U-174
  - Choose Preset, U-173

- Delete, U-170
  - Edit Color Map, U-172
  - Enable Line Series, U-35
  - Lights, U-174
  - Orientation Axes, U-24
  - Refresh Times, U-25, U-171
  - Rescale to Data Range, U-25
  - Reset, U-170
  - Set Ambient Color, U-173
  - Update GUI, U-171
  - Use Parallel Projection, U-24
- C**
- C++ syntax
    - /\*...\*/*, U-78
    - //*, U-78
    - # include*, U-72, U-78
  - cacheAgglomeration* keyword, U-126
  - calculated
    - boundary condition, U-138
  - cAlpha* keyword, U-63
  - Camera Parallel Projection button, U-174
  - cases, U-105
  - castellatedMesh* keyword, U-150
  - castellatedMeshControls*
    - dictionary, U-152–U-154
  - castellatedMeshControls* keyword, U-150
  - cavitatingDyMFoam* solver, U-87
  - cavitatingFoam* solver, U-87
  - cavity flow, U-17
  - CEI\_ARCH
    - environment variable, U-184
  - CEI\_HOME
    - environment variable, U-184
  - cell
    - expansion ratio, U-143
  - cell class, P-29
  - cell
    - keyword entry, U-185
  - cellLimited*
    - keyword entry, U-120
  - cellPoint*
    - keyword entry, U-185
  - cellPointFace*
    - keyword entry, U-185
  - cells*
    - dictionary, U-140
  - central differencing, P-36
  - cfTools* tools, U-98
  - cfx4ToFoam* utility, U-91, U-159
  - changeDictionary* utility, U-90
  - Charts window panel, U-174
  - checkMesh* utility, U-92, U-160
  - chemFoam* solver, U-88
  - chemistryModel*
    - library, U-102
  - chemistryModel* model, U-102
  - chemistrySolver* model, U-102
  - chemkinToFoam* utility, U-97
  - Choose Preset button, U-173
  - chtMultiRegionSimpleFoam* solver, U-88
  - chtMultiRegionFoam* solver, U-88
  - Chung
    - library, U-101
  - 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-129, U-131
    - 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-109
    - word*, P-24, P-29
  - class* keyword, U-107
  - clockTime*
    - keyword entry, U-114
  - cloud* keyword, U-187
  - cloudFunctionObjects*
    - library, U-98
  - cmptAv*
    - tensor member function, P-23
  - Co utility, U-93
  - coalChemistryFoam* solver, U-89
  - coalCombustion*

- library, U-99
  - cofactors
    - tensor member function, P-23
  - coldEngineFoam solver, U-88
  - collapseEdges utility, U-93
  - Color By menu, U-173
  - Color Legend window, U-29
  - Color Legend window panel, U-173
  - Color Scale window panel, U-173
  - Colors window panel, U-174
  - compressibleInterDyMFoam solver, U-87
  - compressibleInterFoam solver, U-87
  - compressibleMultiphaseInterFoam solver, U-87
  - combinePatchFaces utility, U-93
  - comments, U-78
  - commsType keyword, U-80
  - compressed
    - keyword entry, U-114
  - compressibleLESModels
    - library, U-104
  - compressibleRASModels
    - library, U-103
  - constant* directory, U-105, U-191
  - constant model, U-101
  - constTransport model, U-101
  - containers tools, U-98
  - continuum
    - mechanics, P-13
  - control
    - of time, U-113
  - controlDict*
    - dictionary, P-65, U-22, U-31, U-42, U-51, U-62, U-105, U-165
  - controlDict* file, P-48
  - convection, *see* divergence, P-36
  - convergence, U-39
  - conversion
    - library, U-99
  - convertToMeters keyword, U-140
  - convertToMeters keyword, U-141
  - coordinate
    - system, P-13
  - coordinate system, U-18
  - corrected
    - keyword entry, U-120, U-121
  - Courant number, P-40, U-22
  - Cp keyword, U-193
  - cpuTime
    - keyword entry, U-114
  - Crank Nicolson
    - temporal discretisation, P-41
  - CrankNicolson
    - keyword entry, U-122
  - createExternalCoupledPatchGeometry utility, U-90
  - createBaffles utility, U-92
  - createPatch utility, U-92
  - createTurbulenceFields utility, U-94
  - cross product, *see* tensor, vector cross product
  - CrossPowerLaw
    - keyword entry, U-60
  - CrossPowerLaw model, U-104
  - cubeRootVolDelta model, U-103
  - cubicCorrected
    - keyword entry, U-122
  - cubicCorrection
    - keyword entry, U-119
  - curl, P-35
  - curl
    - fvc member function, P-35
  - Current Time Controls menu, U-25, U-171
  - curve keyword, U-187
  - Cv keyword, U-193
  - cyclic
    - boundary condition, U-137, U-146
  - cyclic
    - keyword entry, U-137
  - 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-91
  - db tools, U-98
  - ddt
    - fvc member function, P-35
    - fvm member function, P-35
  - DeardorffDiffStress model, U-103, U-104
  - debug keyword, U-150
  - decompose model, U-100
  - decomposePar utility, U-82, U-83, U-97
  - decomposeParDict*
    - dictionary, U-82
  - decomposition
    - of field, U-82
    - of mesh, U-82
  - decompositionMethods

- library, U-99
- decompression of a tank, P-61
- defaultFieldValues keyword, U-60
- deformedGeom utility, U-92
- Delete button, U-170
- delta keyword, U-83, U-198
- deltaT keyword, U-114
- 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-124, U-125
- DIC
  - keyword entry, U-125
- DICGaussSeidel
  - keyword entry, U-125
- dictionary
  - PISO*, U-23
  - blockMeshDict*, U-18, U-20, U-36, U-49, U-140, U-149
  - boundary*, U-132, U-140
  - castellatedMeshControls*, U-152–U-154
  - cells*, U-140
  - controlDict*, P-65, U-22, U-31, U-42, U-51, U-62, U-105, U-165
  - decomposeParDict*, U-82
  - faces*, U-131, U-140
  - fvSchemes*, U-63, U-105, U-116
  - fvSolution*, U-105, U-123
  - mechanicalProperties*, U-51
  - neighbour*, U-132
  - owner*, U-131
  - points*, U-131, U-140
  - thermalProperties*, U-51
  - thermophysicalProperties*, U-191
  - transportProperties*, U-21, U-39, U-42, U-199
  - turbulenceProperties*, U-41, U-61, U-197
- 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-125
- dimension
  - checking in OpenFOAM, P-24, U-109
- dimensional units, U-109
- dimensioned<Type> template class, P-24
- dimensionedTypes tools, U-98
- dimensions keyword, U-21, U-110
- dimensionSet class, P-24, P-30, P-31
- dimensionSet tools, U-98
- directionMixed
  - boundary condition, U-138
- directory
  - 0.000000e+00*, U-106
  - 0*, U-106
  - Make*, U-73
  - constant*, U-105, U-191
  - fluentInterface*, U-181
  - polyMesh*, U-105, U-131
  - processorN*, U-83
  - run*, U-105
  - system*, P-48, U-105
  - tutorials*, P-43, U-17
- discretisation
  - equation, P-31
- Display window panel, U-24, U-25, U-170, U-172
- distance
  - keyword entry, U-154, U-187
- distributed model, U-100
- distributed keyword, U-83, U-84
- distributionModels
  - library, U-99
- div
  - fv member function, P-35
  - fvm member function, P-35
- divergence, P-35, P-37
- divSchemes keyword, U-116
- dnsFoam solver, U-88
- doLayers keyword, U-150
- double inner product, *see* tensor, double inner product
- DPMFoam solver, U-89
- driftFluxFoam solver, U-87
- dsmc
  - library, U-99
- dsmcFieldsCalc utility, U-95
- dsmcFoam solver, U-89

- dsmcInitialise utility, U-90  
 dx  
     keyword entry, U-185  
 dynamicFvMesh  
     library, U-99  
 dynamicMesh  
     library, U-99  
 dynLagrangian model, U-103  
 dynOneEqEddy model, U-103
- ## E
- eConstThermo model, U-101  
 edgeGrading keyword, U-143  
 edgeMesh  
     library, U-99  
 edges keyword, U-141  
 Edit menu, U-174  
 Edit Color Map button, U-172  
 egrMixture model, U-100  
 egrMixture keyword, U-193  
 electrostaticFoam solver, U-89  
 empty  
     boundary condition, P-63, P-69, U-18, U-133, U-137  
 empty  
     keyword entry, U-137  
 Enable Line Series button, U-35  
 endTime keyword, U-22, U-113, U-114  
 energy keyword, U-192, U-196  
 engine  
     library, U-99  
 engineCompRatio utility, U-95  
 engineFoam solver, U-88  
 engineSwirl utility, U-90  
 ensight74FoamExec utility, U-183  
 ENSIGHT7\_INPUT  
     environment variable, U-184  
 ENSIGHT7\_READER  
     environment variable, U-184  
 ensightFoamReader utility, U-93  
 enstrophy utility, U-93  
 environment variable  
     CEI\_ARCH, U-184  
     CEI\_HOME, U-184  
     ENSIGHT7\_INPUT, U-184  
     ENSIGHT7\_READER, U-184  
     FOAM\_RUN, U-105  
     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-192  
 equilibriumCO utility, U-97  
 equilibriumFlameT utility, U-97  
 errorReduction keyword, U-158  
 Euler  
     keyword entry, U-122  
 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-95  
 expandDictionary utility, U-97  
 expansionRatio keyword, U-157  
 explicit  
     temporal discretisation, P-40  
 extrude2DMesh utility, U-91  
 extrudeMesh utility, U-91  
 extrudeToRegionMesh utility, U-91
- ## F
- face class, P-29  
 face keyword, U-187  
 faceAgglomerate utility, U-90  
 faceAreaPair  
     keyword entry, U-125  
 faceLimited  
     keyword entry, U-120  
 faces  
     dictionary, U-131, U-140  
 FDIC  
     keyword entry, U-125  
 featureAngle keyword, U-157  
 features keyword, U-152

- field
  - U, U-23
  - p, U-23
  - decomposition, U-82
- FieldField<Type> template class, P-30
- fieldFunctionObjects
  - library, U-98
- fields, P-27
  - mapping, U-165
- fields tools, U-98
- fields keyword, U-185
- Field<Type> template class, P-27
- fieldValues keyword, U-60
- file
  - Make/files*, U-75
  - controlDict*, P-48
  - files*, U-73
  - g*, U-60
  - options*, U-73
  - snappyHexMeshDict*, U-150
  - transportProperties*, U-60
- file format, U-106
- fileFormats
  - library, U-99
- fileModificationChecking keyword, U-80
- fileModificationSkew keyword, U-80
- files* file, U-73
- filteredLinear2
  - keyword entry, U-119
- finalLayerThickness keyword, U-157
- financialFoam solver, U-90
- find script/alias, U-179
- finite volume
  - discretisation, P-25
  - mesh, P-29
- finiteVolume
  - library, U-98
- finiteVolume tools, U-98
- finiteVolumeCalculus class, P-34
- finiteVolumeMethod class, P-34
- fireFoam solver, U-88
- firstTime keyword, U-113
- fixed
  - keyword entry, U-114
- fixedGradient
  - boundary condition, U-138
- fixedValue
  - boundary condition, U-138
- flattenMesh utility, U-92
- floatTransfer keyword, U-80
- 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-93
- fluent3DMeshToFoam utility, U-91
- fluentInterface* directory, U-181
- fluentMeshToFoam utility, U-91, U-159
- fluxCorrectedVelocity
  - boundary condition, U-139
- fluxRequired keyword, U-116
- OpenFOAM
  - cases, U-105
- FOAM\_RUN
  - environment variable, U-105
- foamCalc utility, U-33
- foamCalcFunctions
  - library, U-98
- foamChemistryFile keyword, U-193
- foamCorrectVrt script/alias, U-164
- foamDataToFluent utility, U-93, U-181
- foamDebugSwitches utility, U-97
- FoamFile keyword, U-107
- foamFile
  - keyword entry, U-185
- foamFormatConvert utility, U-97
- foamHelp utility, U-97
- foamInfoExec utility, U-97
- foamJob script/alias, U-188
- foamListTimes utility, U-95
- foamLog script/alias, U-188
- foamMeshToFluent utility, U-91, U-181
- foamToEnight utility, U-93
- foamToEnightParts utility, U-93
- foamToGMV utility, U-93
- foamToStarMesh utility, U-91
- foamToSurface utility, U-91
- foamToTecplot360 utility, U-93
- foamToTetDualMesh utility, U-93
- foamToVTK utility, U-93
- foamUpgradeCyclics utility, U-90
- foamUpgradeFvSolution utility, U-90
- foamyHexMeshBackgroundMesh utility, U-91
- foamyHexMeshSurfaceSimplify utility, U-91
- foamyHexMesh utility, U-91
- foamyQuadMesh utility, U-91

forces  
   library, U-98  
 foreground  
   process, U-24  
 format keyword, U-107  
 fourth  
   keyword entry, U-120, U-121  
 fuel keyword, U-193  
 functionObjectLibs keyword, U-179  
 functions keyword, U-115, U-177  
 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-101  
 FVFunctionObjects  
   library, U-98  
 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-98  
 fvMatrix template class, P-34  
 fvMesh class, P-29  
 fvMesh tools, U-98  
 fvMotionSolvers  
   library, U-99  
 fvSchemes  
   dictionary, U-63, U-105, U-116  
 fvSchemes class, P-36  
 fvSchemes  
   menu entry, U-52  
 fvSolution  
   dictionary, U-105, U-123

## G

g file, U-60

gambitToFoam utility, U-91, U-159  
 GAMG  
   keyword entry, U-53, U-124, U-125  
 Gamma  
   keyword entry, U-119  
 Gamma differencing, P-36  
 Gauss  
   keyword entry, U-120  
 Gauss's theorem, P-34  
 GaussSeidel  
   keyword entry, U-125  
 General window panel, U-174  
 general  
   keyword entry, U-114  
 genericFvPatchField  
   library, U-99  
 geometric-algebraic multi-grid, U-125  
 GeometricBoundaryField template class, P-30  
 geometricField<Type> template class, P-30  
 geometry keyword, U-150  
 gGrad  
   fvc member function, P-35  
 global tools, U-98  
 gmshToFoam utility, U-91  
 gnuplot  
   keyword entry, U-115, U-185  
 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-116  
 graph tools, U-98  
 graphFormat keyword, U-115  
 GuldersonEgRlaminarFlameSpeed model, U-101  
 GuldersonLaminarFlameSpeed model, U-101

## H

hConstThermo model, U-101  
 heheuPsiThermo model, U-100  
 heheuPsiThermo  
   keyword entry, U-192  
 Help menu, U-173  
 hePsiThermo model, U-100  
 hePsiThermo  
   keyword entry, U-192  
 heRhoThermo model, U-100

- heRhoThermo  
     keyword entry, U-192  
 HerschelBulkley model, U-104  
 hExponentialThermo  
     library, U-102  
 Hf keyword, U-193  
 hierarchical  
     keyword entry, U-82, U-83  
 highCpCoeffs keyword, U-194  
 homogenousDynOneEqEddy model, U-103, U-104  
 homogenousDynSmagorinsky model, U-103  
 homogeneousMixture model, U-100  
 homogeneousMixture keyword, U-193  
 hPolynomialThermo model, U-101
- ## I
- I  
     tensor member function, P-23  
 icoFoam solver, U-17, U-21, U-22, U-24, U-86  
 icoPolynomial model, U-101, U-195  
 icoUncoupledKinematicParcelFoam solver, U-89  
 ideasToFoam utility, U-159  
 ideasUnvToFoam utility, U-91  
 identities, *see* tensor, identities  
 identity, *see* tensor, identity  
 incompressibleLESMoels  
     library, U-103  
 incompressiblePerfectGas model, U-101, U-195  
 incompressibleRASMoels  
     library, U-102  
 incompressibleTransportMoels  
     library, P-53, U-104  
 incompressibleTurbulenceMoels  
     library, P-53  
 index  
     notation, P-14, P-15  
 Information window panel, U-170  
 inhomogeneousMixture model, U-100  
 inhomogeneousMixture keyword, U-193  
 inlet  
     boundary condition, P-69  
 inletOutlet  
     boundary condition, U-139  
 inner product, *see* tensor, inner product  
 inotify  
     keyword entry, U-80  
 inotifyMaster  
     keyword entry, U-80  
 inside  
     keyword entry, U-154  
 insideCells utility, U-92  
 interPhaseChangeDyMFoam solver, U-87  
 interPhaseChangeFoam solver, U-87  
 interfaceProperties  
     library, U-104  
 interfaceProperties model, U-104  
 interFoam solver, U-87  
 interMixingFoam solver, U-87  
 internalField keyword, U-21, U-110  
 interpolation tools, U-98  
 interpolationScheme keyword, U-185  
 interpolations tools, U-98  
 interpolationSchemes keyword, U-116  
 inv  
     tensor member function, P-23  
 iterations  
     maximum, U-124
- ## J
- janafThermo model, U-101  
 jobControl  
     library, U-98  
 jplot  
     keyword entry, U-115, U-185
- ## K
- kEpsilon model, U-102, U-103  
 keyword  
     As, U-193  
     Cp, U-193  
     Cv, U-193  
     FoamFile, U-107  
     Hf, U-193  
     LESMoel, U-198  
     N2, U-193  
     O2, U-193  
     Pr, U-193  
     RASMoel, U-198  
     Tcommon, U-194  
     Thigh, U-194  
     Tlow, U-194  
     Ts, U-193  
     addLayersControls, U-150  
     adjustTimeStep, U-62, U-115  
     agglomerator, U-125  
     arc, U-141  
     blocks, U-20, U-31, U-142  
     block, U-141  
     boundaryField, U-21, U-110  
     boundary, U-145  
     boxToCell, U-60  
     burntProducts, U-193

- cAlpha, U-63
- cacheAgglomeration, U-126
- castellatedMeshControls, U-150
- castellatedMesh, U-150
- class, U-107
- cloud, U-187
- commsType, U-80
- convertToMeters, U-141
- convertToMeters, U-140
- curve, U-187
- debug, U-150
- defaultFieldValues, U-60
- deltaT, U-114
- delta, U-83, U-198
- dimensions, U-21, U-110
- distributed, U-83, U-84
- divSchemes, U-116
- doLayers, U-150
- edgeGrading, U-143
- edges, U-141
- egrMixture, U-193
- endTime, U-22, U-113, U-114
- energy, U-192, U-196
- equationOfState, U-192
- errorReduction, U-158
- expansionRatio, U-157
- face, U-187
- featureAngle, U-157
- features, U-152
- fieldValues, U-60
- fields, U-185
- fileModificationChecking, U-80
- fileModificationSkew, U-80
- finalLayerThickness, U-157
- firstTime, U-113
- floatTransfer, U-80
- fluxRequired, U-116
- foamChemistryFile, U-193
- format, U-107
- fuel, U-193
- functionObjectLibs, U-179
- functions, U-115, U-177
- geometry, U-150
- gradSchemes, U-116
- graphFormat, U-115
- highCpCoeffs, U-194
- homogeneousMixture, U-193
- inhomogeneousMixture, U-193
- internalField, U-21, U-110
- interpolationSchemes, U-116
- interpolationScheme, U-185
- laplacianSchemes, U-116
- latestTime, U-39
- layers, U-157
- leastSquares, U-52
- levels, U-154
- libs, U-80, U-115
- locationInMesh, U-152, U-154
- location, U-107
- lowCpCoeffs, U-194
- manualCoeffs, U-83
- maxAlphaCo, U-62
- maxBoundarySkewness, U-158
- maxConcave, U-158
- maxCo, U-62, U-115
- maxDeltaT, U-62
- maxFaceThicknessRatio, U-157
- maxGlobalCells, U-152
- maxInternalSkewness, U-158
- maxIter, U-124
- maxLocalCells, U-152
- maxNonOrtho, U-158
- maxThicknessToMedialRatio, U-157
- mergeLevels, U-126
- mergePatchPairs, U-141
- mergeTolerance, U-150
- meshQualityControls, U-150
- method, U-83
- midPointAndFace, U-187
- midPoint, U-187
- minArea, U-158
- minDeterminant, U-158
- minFaceWeight, U-158
- minFlatness, U-158
- minMedianAxisAngle, U-157
- minRefinementCells, U-152
- minThickness, U-157
- minTriangleTwist, U-158
- minTwist, U-158
- minVolRatio, U-158
- minVol, U-158
- mixture, U-193
- mode, U-154
- molWeight, U-196
- multiComponentMixture, U-193
- mu, U-193
- nAlphaSubCycles, U-63
- nBufferCellsNoExtrude, U-157
- nCellsBetweenLevels, U-152
- nFaces, U-132

- nFinestSweeps, U-126
  - nGrow, U-157
  - nLayerIter, U-157
  - nMoles, U-196
  - nPostSweeps, U-126
  - nPreSweeps, U-126
  - nRelaxIter, U-155, U-157
  - nRelaxedIter, U-157
  - nSmoothNormals, U-157
  - nSmoothPatch, U-155
  - nSmoothScale, U-158
  - nSmoothSurfaceNormals, U-157
  - nSmoothThickness, U-157
  - nSolveIter, U-155
  - neighbourPatch, U-146
  - numberOfSubdomains, U-83
  - nu, U-199
  - n, U-83
  - object, U-107
  - order, U-83
  - outputControl, U-179
  - oxidant, U-193
  - pRefCell, U-23, U-128
  - pRefValue, U-23, U-127
  - p\_rhgRefCell, U-128
  - p\_rhgRefValue, U-128
  - patchMap, U-166
  - patches, U-141
  - preconditioner, U-124, U-125
  - pressure, U-51
  - printCoeffs, U-42, U-198
  - processorWeights, U-82
  - processorWeights, U-83
  - purgeWrite, U-114
  - refGradient, U-138
  - refinementRegions, U-152, U-154
  - refinementSurfaces, U-152, U-153
  - refinementRegions, U-154
  - regions, U-60
  - relTol, U-53, U-124
  - relativeSizes, U-157
  - relaxed, U-158
  - resolveFeatureAngle, U-152, U-153
  - roots, U-83, U-84
  - runTimeModifiable, U-115
  - scotchCoeffs, U-83
  - setFormat, U-185
  - sets, U-185
  - simpleGrading, U-143
  - simulationType, U-41, U-61, U-197
  - singleStepReactingMixture, U-193
  - smoother, U-126
  - snGradSchemes, U-116
  - snapControls, U-150
  - snap, U-150
  - solvers, U-123
  - solver, U-53, U-123
  - specie, U-196
  - spline, U-141
  - startFace, U-132
  - startFrom, U-22, U-113
  - startTime, U-22, U-113
  - stopAt, U-113
  - strategy, U-82, U-83
  - surfaceFormat, U-185
  - surfaces, U-185
  - thermoType, U-191
  - thermodynamics, U-196
  - timeFormat, U-114
  - timePrecision, U-115
  - timeScheme, U-116
  - tolerance, U-53, U-124, U-155
  - topoSetSource, U-60
  - traction, U-51
  - transport, U-192, U-196
  - turbulence, U-198
  - type, U-135, U-192
  - uniform, U-187
  - valueFraction, U-138
  - value, U-21, U-138
  - version, U-107
  - vertices, U-20, U-141
  - veryInhomogeneousMixture, U-193
  - writeCompression, U-114
  - writeControl, U-22, U-62, U-114
  - writeFormat, U-55, U-114
  - writeInterval, U-23, U-32, U-114
  - writePrecision, U-114
  - <LESModel>Coeffs, U-198
  - <RASModel>Coeffs, U-198
  - <delta>Coeffs, U-198
- keyword entry
- BSpline, U-142
  - CrankNicolson, U-122
  - CrossPowerLaw, U-60
  - DICGaussSeidel, U-125
  - DIC, U-125
  - DILU, U-125
  - Euler, U-122
  - FDIC, U-125

GAMG, U-53, U-124, U-125  
Gamma, U-119  
GaussSeidel, U-125  
Gauss, U-120  
LESModel, U-41  
LES, U-41, U-197  
MGridGen, U-125  
MUSCL, U-119  
Newtonian, U-60  
PBiCG, U-124  
PCG, U-124  
QUICK, U-122  
RASModel, U-41  
RAS, U-41, U-197  
SFCD, U-119, U-122  
UMIST, U-117  
adjustableRunTime, U-62, U-114  
arc, U-142  
ascii, U-114  
backward, U-122  
binary, U-114  
blocking, U-80  
bounded, U-120, U-121  
cellLimited, U-120  
cellPointFace, U-185  
cellPoint, U-185  
cell, U-185  
clockTime, U-114  
compressed, U-114  
corrected, U-120, U-121  
cpuTime, U-114  
cubicCorrected, U-122  
cubicCorrection, U-119  
cyclic, U-137  
diagonal, U-124, U-125  
distance, U-154, U-187  
dx, U-185  
empty, U-137  
faceAreaPair, U-125  
faceLimited, U-120  
filteredLinear2, U-119  
fixed, U-114  
foamFile, U-185  
fourth, U-120, U-121  
general, U-114  
gnuplot, U-115, U-185  
hePsiThermo, U-192  
heRhoThermo, U-192  
heheuPsiThermo, U-192  
hierarchical, U-82, U-83  
inotifyMaster, U-80  
inotify, U-80  
inside, U-154  
jplot, U-115, U-185  
laminar, U-41, U-197  
latestTime, U-113  
leastSquares, U-120  
limitedCubic, U-119  
limitedLinear, U-119  
limited, U-120, U-121  
linearUpwind, U-119, U-122  
linear, U-119, U-122  
line, U-142  
localEuler, U-122  
manual, U-82, U-83  
metis, U-83  
midPoint, U-119  
nextWrite, U-114  
noWriteNow, U-114  
nonBlocking, U-80  
none, U-117, U-125  
null, U-185  
outputTime, U-179  
outside, U-154  
patch, U-137, U-186  
polyLine, U-142  
processor, U-137  
pureMixture, U-193  
raw, U-115, U-185  
reactingMixture, U-193  
runTime, U-32, U-114  
scheduled, U-80  
scientific, U-114  
scotch, U-82, U-83  
simple, U-82, U-83  
skewLinear, U-119, U-122  
smoothSolver, U-124  
spline, U-142  
startTime, U-22, U-113  
steadyState, U-122  
stl, U-185  
symmetryPlane, U-137  
timeStampMaster, U-80  
timeStamp, U-80  
timeStep, U-23, U-32, U-114, U-179  
uncompressed, U-114  
uncorrected, U-120, U-121  
upwind, U-119, U-122  
vanLeer, U-119  
vtk, U-185

- wall, U-137
  - wedge, U-137
  - writeControl, U-114
  - writeInterval, U-179
  - writeNow, U-113
  - xmgr, U-115, U-185
  - xyz, U-187
  - x, U-187
  - y, U-187
  - z, U-187
  - kivaToFoam utility, U-91
  - kkLOmega model, U-102
  - kOmega model, U-102
  - kOmegaSST model, U-102, U-103
  - kOmegaSSTSAS model, U-103
  - Kronecker delta, P-19
- L**
- lagrangian
    - library, U-99
  - lagrangianIntermediate
    - library, U-99
  - Lambda2 utility, U-94
  - LamBremhorstKE model, U-102
  - laminar model, U-102, U-103
  - laminar
    - keyword entry, U-41, U-197
  - laminarFlameSpeedModels
    - library, U-101
  - laplaceFilter model, U-103
  - Laplacian, P-36
  - laplacian, P-35
  - laplacian
    - fvc member function, P-35
    - fvm member function, P-35
  - laplacianFoam solver, U-86
  - laplacianSchemes keyword, U-116
  - latestTime
    - keyword entry, U-113
  - latestTime keyword, U-39
  - LaunderGibsonRSTM model, U-102, U-103
  - LaunderSharmaKE model, U-102, U-103
  - layers keyword, U-157
  - leastSquares
    - keyword entry, U-120
  - leastSquares keyword, U-52
  - LES
    - keyword entry, U-41, U-197
  - LESdeltas
    - library, U-103
  - LESfilters
    - library, U-103
  - LESModel
    - keyword entry, U-41
  - LESModel keyword, U-198
  - levels keyword, U-154
  - libraries, U-69
  - library
    - Chung, U-101
    - FVFunctionObjects, U-98
    - LESdeltas, U-103
    - LESfilters, U-103
    - MGridGenGAMGAgglomeration, U-99
    - ODE, U-99
    - OSspecific, U-99
    - OpenFOAM, U-98
    - P1, U-101
    - PV4FoamReader, U-169
    - SLGThermo, U-102
    - Wallis, U-101
    - autoMesh, U-99
    - barotropicCompressibilityModels, U-101
    - basicSolidThermo, U-102
    - basicThermophysicalModels, U-100
    - blockMesh, U-99
    - chemistryModel, U-102
    - cloudFunctionObjects, U-98
    - coalCombustion, U-99
    - compressibleLESModels, U-104
    - compressibleRASModels, U-103
    - conversion, U-99
    - decompositionMethods, U-99
    - distributionModels, U-99
    - dsmc, U-99
    - dynamicFvMesh, U-99
    - dynamicMesh, U-99
    - edgeMesh, U-99
    - engine, U-99
    - fieldFunctionObjects, U-98
    - fileFormats, U-99
    - finiteVolume, U-98
    - foamCalcFunctions, U-98
    - forces, U-98
    - fvDOM, U-101
    - fvmotionSolvers, U-99
    - genericFvPatchField, U-99
    - hExponentialThermo, U-102
    - incompressibleLESModels, U-103
    - incompressibleRASModels, U-102
    - incompressibleTransportModels, P-53, U-104
    - incompressibleTurbulenceModels, P-53

interfaceProperties, U-104  
 jobControl, U-98  
 lagrangianIntermediate, U-99  
 lagrangian, U-99  
 laminarFlameSpeedModels, U-101  
 linear, U-101  
 liquidMixtureProperties, U-102  
 liquidProperties, U-102  
 meshTools, U-99  
 molecularMeasurements, U-99  
 molecule, U-99  
 opaqueSolid, U-101  
 pairPatchAgglomeration, U-99  
 postCalc, U-98  
 potential, U-99  
 primitive, P-21  
 radiationModels, U-100  
 randomProcesses, U-99  
 reactionThermophysicalModels, U-100  
 sampling, U-98  
 solidChemistryModel, U-102  
 solidMixtureProperties, U-102  
 solidParticle, U-99  
 solidProperties, U-102  
 solidSpecie, U-102  
 solidThermo, U-102  
 specie, U-101  
 spray, U-99  
 surfMesh, U-99  
 surfaceFilmModels, U-104  
 systemCall, U-99  
 thermophysicalFunctions, U-101  
 thermophysical, U-191  
 topoChangerFvMesh, U-99  
 triSurface, U-99  
 turbulence, U-99  
 twoPhaseProperties, U-104  
 utilityFunctionObjects, U-99  
 viewFactor, U-101  
 vtkPV4Foam, U-169  
 libs keyword, U-80, U-115  
 lid-driven cavity flow, U-17  
 LienCubicKE model, U-102  
 LienCubicKELowRe model, U-102  
 LienLeschzinerLowRe model, U-102  
 Lights button, U-174  
 limited  
     keyword entry, U-120, U-121  
 limitedCubic  
     keyword entry, U-119

limitedLinear  
     keyword entry, U-119  
 line  
     keyword entry, U-142  
 Line Style menu, U-35  
 linear  
     library, U-101  
 linear model, U-195  
 linear  
     keyword entry, U-119, U-122  
 linearUpwind  
     keyword entry, U-119, U-122  
 liquid  
     electrically-conducting, P-67  
 liquidMixtureProperties  
     library, U-102  
 liquidProperties  
     library, U-102  
 lists, P-27  
 List<Type> template class, P-27  
 localEuler  
     keyword entry, U-122  
 location keyword, U-107  
 locationInMesh keyword, U-152, U-154  
 lowCpCoeffs keyword, U-194  
 lowReOneEqEddy model, U-104  
 LRDDiffStress model, U-103  
 LRR model, U-102, U-103  
 lsGrad  
     fvc member function, P-35

## M

Mach utility, U-94  
 mag  
     tensor member function, P-23  
 magneticFoam solver, U-89  
 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-82, U-83  
 manualCoeffs keyword, U-83  
 mapFields utility, U-31, U-38, U-42, U-56, U-90,  
     U-165  
 mapFieldsPar utility, U-90  
 mapping  
     fields, U-165  
 Marker Style menu, U-35

- matrices tools, U-98
- max
  - tensor member function, P-23
- maxAlphaCo keyword, U-62
- maxBoundarySkewness keyword, U-158
- maxCo keyword, U-62, U-115
- maxConcave keyword, U-158
- maxDeltaT keyword, U-62
- maxDeltaxyz model, U-103
- maxFaceThicknessRatio keyword, U-157
- maxGlobalCells keyword, U-152
- maximum iterations, U-124
- maxInternalSkewness keyword, U-158
- maxIter keyword, U-124
- maxLocalCells keyword, U-152
- maxNonOrtho keyword, U-158
- maxThicknessToMedialRatio keyword, U-157
- mdEquilibrationFoam solver, U-89
- mdFoam solver, U-89
- mdlInitialise utility, U-90
- mechanicalProperties*
  - dictionary, U-51
- memory tools, U-98
- menu
  - Color By, U-173
  - Current Time Controls, U-25, U-171
  - Edit, U-174
  - Help, U-173
  - Line Style, U-35
  - Marker Style, U-35
  - VCR Controls, U-25, U-171
  - View, U-170, U-173
- menu entry
  - Plot Over Line, U-34
  - Save Animation, U-175
  - Save Screenshot, U-175
  - Settings, U-174
  - Solid Color, U-173
  - Toolbars, U-173
  - View Settings, U-24, U-173
  - Wireframe, U-173
  - fvSchemes, U-52
- mergeLevels keyword, U-126
- mergeMeshes utility, U-92
- mergeOrSplitBaffles utility, U-92
- mergePatchPairs keyword, U-141
- mergeTolerance keyword, U-150
- mesh
  - 1-dimensional, U-133
  - 1D, U-133
  - 2-dimensional, U-133
  - 2D, U-133
  - axi-symmetric, U-133
  - basic, P-29
  - block structured, U-140
  - decomposition, U-82
  - description, U-129
  - finite volume, P-29
  - generation, U-140, U-149
  - grading, U-140, U-143
  - grading, example of, P-50
  - non-orthogonal, P-43
  - refinement, P-61
  - resolution, U-29
  - specification, U-129
  - split-hex, U-149
  - Stereolithography (STL), U-149
  - surface, U-149
  - validity constraints, U-129
- Mesh Parts window panel, U-24
- meshes tools, U-98
- meshQualityControls keyword, U-150
- meshTools
  - library, U-99
- message passing interface
  - openMPI, U-84
- method keyword, U-83
- metis
  - keyword entry, U-83
- metisDecomp model, U-100
- MGridGenGAMGAgglomeration
  - library, U-99
- MGridGen
  - keyword entry, U-125
- mhdFoam solver, P-69, U-89
- midPoint
  - keyword entry, U-119
- midPoint keyword, U-187
- midPointAndFace keyword, U-187
- min
  - tensor member function, P-23
- minArea keyword, U-158
- minDeterminant keyword, U-158
- minFaceWeight keyword, U-158
- minFlatness keyword, U-158
- minMedianAxisAngle keyword, U-157
- MINMOD differencing, P-36
- minRefinementCells keyword, U-152
- minThickness keyword, U-157
- minTriangleTwist keyword, U-158

- minTwist keyword, U-158
- minVol keyword, U-158
- minVolRatio keyword, U-158
- mirrorMesh utility, U-92
- mixed
  - boundary condition, U-138
- mixedSmagorinsky model, U-103
- mixture keyword, U-193
- mixtureAdiabaticFlameT utility, U-97
- mode keyword, U-154
- model
  - APIfunctions, U-102
  - BirdCarreau, U-104
  - CrossPowerLaw, U-104
  - DeardorffDiffStress, U-103, U-104
  - GuldersEGRLaminarFlameSpeed, U-101
  - GuldersLaminarFlameSpeed, U-101
  - HerschelBulkley, U-104
  - LRDDiffStress, U-103
  - LRR, U-102, U-103
  - LamBremhorstKE, U-102
  - LaunderGibsonRSTM, U-102, U-103
  - LaunderSharmaKE, U-102, U-103
  - LienCubicKELowRe, U-102
  - LienCubicKE, U-102
  - LienLeschzinerLowRe, U-102
  - NSRDSfunctions, U-102
  - Newtonian, U-104
  - NonlinearKEShah, U-102
  - PengRobinsonGas, U-195
  - PrandtlDelta, U-103
  - RNGkEpsilon, U-102, U-103
  - RaviPetersen, U-101
  - Smagorinsky2, U-103
  - Smagorinsky, U-103, U-104
  - SpalartAllmarasDDES, U-104
  - SpalartAllmarasIDDES, U-104
  - SpalartAllmaras, U-102–U-104
  - adiabaticPerfectFluid, U-101, U-195
  - anisotropicFilter, U-103
  - basicMultiComponentMixture, U-100
  - chemistryModel, U-102
  - chemistrySolver, U-102
  - constTransport, U-101
  - constant, U-101
  - cubeRootVolDelta, U-103
  - decompose, U-100
  - distributed, U-100
  - dynLagrangian, U-103
  - dynOneEqEddy, U-103
  - eConstThermo, U-101
  - egrMixture, U-100
  - hConstThermo, U-101
  - hPolynomialThermo, U-101
  - hePsiThermo, U-100
  - heRhoThermo, U-100
  - heheuPsiThermo, U-100
  - homogenousDynOneEqEddy, U-103, U-104
  - homogenousDynSmagorinsky, U-103
  - homogeneousMixture, U-100
  - icoPolynomial, U-101, U-195
  - incompressiblePerfectGas, U-101, U-195
  - inhomogeneousMixture, U-100
  - interfaceProperties, U-104
  - janafThermo, U-101
  - kEpsilon, U-102, U-103
  - kOmegaSSTSAS, U-103
  - kOmegaSST, U-102, U-103
  - kOmega, U-102
  - kkLOmega, U-102
  - laminar, U-102, U-103
  - laplaceFilter, U-103
  - linear, U-195
  - lowReOneEqEddy, U-104
  - maxDeltaxyz, U-103
  - metisDecomp, U-100
  - mixedSmagorinsky, U-103
  - multiComponentMixture, U-100
  - multiphaseMixtureThermo, U-192
  - oneEqEddy, U-103, U-104
  - perfectFluid, U-101, U-195
  - perfectGas, U-195
  - polynomialTransport, U-101
  - powerLaw, U-104
  - psiReactionThermo, U-100, U-192
  - psiThermo, U-192
  - psiuReactionThermo, U-100, U-192
  - ptsotchDecomp, U-100
  - pureMixture, U-100
  - qZeta, U-102
  - reactingMixture, U-100
  - realizableKE, U-102, U-103
  - reconstruct, U-100
  - rhoConst, U-101, U-195
  - rhoReactionThermo, U-100, U-192
  - rhoThermo, U-192
  - scaleSimilarity, U-103
  - scotchDecomp, U-100
  - simpleFilter, U-103
  - singleStepReactingMixture, U-100

- smoothDelta, U-103
  - specieThermo, U-101
  - spectEddyVisc, U-103
  - sutherlandTransport, U-101
  - v2f, U-103
  - vanDriestDelta, U-104
  - veryInhomogeneousMixture, U-100
  - modifyMesh utility, U-93
  - molecularMeasurements
    - library, U-99
  - molecule
    - library, U-99
  - molWeight keyword, U-196
  - moveDynamicMesh utility, U-92
  - moveEngineMesh utility, U-92
  - moveMesh utility, U-92
  - movingWallVelocity
    - boundary condition, U-139
  - MPI
    - openMPI, U-84
  - mshToFoam utility, U-91
  - mu keyword, U-193
  - multiComponentMixture model, U-100
  - multiComponentMixture keyword, U-193
  - multigrid
    - geometric-algebraic, U-125
  - multiphaseEulerFoam solver, U-87
  - multiphaseInterFoam solver, U-87
  - multiphaseMixtureThermo model, U-192
  - MUSCL
    - keyword entry, U-119
- N**
- n keyword, U-83
  - N2 keyword, U-193
  - nabla
    - operator, P-25
  - nAlphaSubCycles keyword, U-63
  - nBufferCellsNoExtrude keyword, U-157
  - nCellsBetweenLevels keyword, U-152
  - neighbour*
    - dictionary, U-132
  - neighbourPatch keyword, U-146
  - netgenNeutralToFoam utility, U-91
  - Newtonian
    - keyword entry, U-60
  - Newtonian model, U-104
  - nextWrite
    - keyword entry, U-114
  - nFaces keyword, U-132
  - nFinestSweeps keyword, U-126
  - nGrow keyword, U-157
  - nLayerIter keyword, U-157
  - nMoles keyword, U-196
  - non-orthogonal mesh, P-43
  - nonBlocking
    - keyword entry, U-80
  - none
    - keyword entry, U-117, U-125
  - NonlinearKEShik model, U-102
  - nonNewtonianIcoFoam solver, U-86
  - noWriteNow
    - keyword entry, U-114
  - nPostSweeps keyword, U-126
  - nPreSweeps keyword, U-126
  - nRelaxedIter keyword, U-157
  - nRelaxIter keyword, U-155, U-157
  - nSmoothNormals keyword, U-157
  - nSmoothPatch keyword, U-155
  - nSmoothScale keyword, U-158
  - nSmoothSurfaceNormals keyword, U-157
  - nSmoothThickness keyword, U-157
  - nSolveIter keyword, U-155
  - NSRDSfunctions model, U-102
  - nu keyword, U-199
  - null
    - keyword entry, U-185
  - numberOfSubdomains keyword, U-83
- O**
- O2 keyword, U-193
  - object keyword, U-107
  - objToVTK utility, U-92
  - ODE
    - library, U-99
  - oneEqEddy model, U-103, U-104
  - Opacity text box, U-173
  - opaqueSolid
    - library, U-101
  - OpenFOAM
    - applications, U-69
    - file format, U-106
    - libraries, U-69
  - OpenFOAM
    - library, U-98
  - OpenFOAM file syntax
    - //, U-106
  - openMPI
    - message passing interface, U-84
    - MPI, U-84
  - operator
    - scalar, P-26

vector, P-25  
 Options window, U-174  
*options* file, U-73  
 order keyword, U-83  
 Orientation Axes button, U-24  
 orientFaceZone utility, U-92  
 OSspecific  
   library, U-99  
 outer product, *see* tensor, outer product  
 outlet  
   boundary condition, P-69  
 outletInlet  
   boundary condition, U-139  
 outputControl keyword, U-179  
 outputTime  
   keyword entry, U-179  
 outside  
   keyword entry, U-154  
 owner  
   dictionary, U-131  
 oxidant keyword, U-193

## P

p field, U-23  
 P1  
   library, U-101  
 p\_rhgRefCell keyword, U-128  
 p\_rhgRefValue keyword, U-128  
 pairPatchAgglomeration  
   library, U-99  
 paraFoam, U-23, U-169  
 parallel  
   running, U-81  
 Paramters window panel, U-171  
 partialSlip  
   boundary condition, U-139  
 particleTracks utility, U-95  
 patch  
   boundary condition, U-136  
 patch  
   keyword entry, U-137, U-186  
 patchAverage utility, U-94  
 patches keyword, U-141  
 patchIntegrate utility, U-94  
 patchMap keyword, U-166  
 patchSummary utility, U-97  
 PBiCG  
   keyword entry, U-124  
 PCG  
   keyword entry, U-124  
 pdfPlot utility, U-95  
 PDRFoam solver, U-88  
 PDRMesh utility, U-93  
 Pe utility, U-94  
 PengRobinsonGas model, U-195  
 perfectFluid model, U-101, U-195  
 perfectGas model, U-195  
 permutation symbol, P-18  
 pimpleFoam solver, U-86  
 Pipeline Browser window, U-24, U-170  
 PISO  
   dictionary, U-23  
 pisoFoam solver, U-17, U-86  
 Plot Over Line  
   menu entry, U-34  
 plot3dToFoam utility, U-91  
 pointField class, P-29  
 pointField<Type> template class, P-31  
 points  
   dictionary, U-131, U-140  
 polyBoundaryMesh class, P-29  
 polyDualMesh utility, U-92  
 polyLine  
   keyword entry, U-142  
 polyMesh directory, U-105, U-131  
 polyMesh class, P-29, U-129, U-131  
 polynomialTransport model, U-101  
 polyPatch class, P-29  
 polyPatchList class, P-29  
 post-processing, U-169  
   post-processing  
     paraFoam, U-169  
 postCalc  
   library, U-98  
 postChannel utility, U-95  
 potential  
   library, U-99  
 potentialFreeSurfaceFoam solver, U-87  
 potentialFoam solver, P-44, U-86  
 pow  
   tensor member function, P-23  
 powerLaw model, U-104  
 pPrime2 utility, U-94  
 Pr keyword, U-193  
 PrandtlDelta model, U-103  
 preconditioner keyword, U-124, U-125  
 pRefCell keyword, U-23, U-128  
 pRefValue keyword, U-23, U-127  
 pressure keyword, U-51  
 pressure waves  
   in liquids, P-62

- pressureDirectedInletVelocity
    - boundary condition, U-139
  - pressureInletVelocity
    - boundary condition, U-139
  - pressureOutlet
    - boundary condition, P-63
  - pressureTransmissive
    - boundary condition, U-139
  - primitive
    - library, P-21
  - primitives tools, U-98
  - printCoeffs keyword, U-42, U-198
  - processorWeights keyword, U-82
  - probeLocations utility, U-95
  - process
    - background, U-24, U-81
    - foreground, U-24
  - processor
    - boundary condition, U-138
  - processor
    - keyword entry, U-137
  - processorN directory, U-83
  - processorWeights keyword, U-83
  - Properties window, U-171, U-172
  - Properties window panel, U-25, U-170
  - psiReactionThermo model, U-100, U-192
  - psiThermo model, U-192
  - psiuReactionThermo model, U-100, U-192
  - ptot utility, U-95
  - ptsotchDecomp model, U-100
  - pureMixture model, U-100
  - pureMixture
    - keyword entry, U-193
  - purgeWrite keyword, U-114
  - PV4FoamReader
    - library, U-169
- Q**
- Q utility, U-94
  - QUICK
    - keyword entry, U-122
  - qZeta model, U-102
- R**
- R utility, U-94
  - radiationModels
    - library, U-100
  - randomProcesses
    - library, U-99
  - RAS
    - keyword entry, U-41, U-197
  - RASModel
    - keyword entry, U-41
  - RASModel keyword, U-198
  - RaviPetersen model, U-101
  - raw
    - keyword entry, U-115, U-185
  - reactingEulerFoam solver, U-88
  - reactingFoam solver, U-88
  - reactingMixture model, U-100
  - reactingMixture
    - keyword entry, U-193
  - reactingParcelFilmFoam solver, U-89
  - reactingParcelFoam solver, U-89
  - reactionThermophysicalModels
    - library, U-100
  - realizableKE model, U-102, U-103
  - reconstruct model, U-100
  - reconstructPar utility, U-85, U-97
  - reconstructParMesh utility, U-97
  - redistributePar utility, U-97
  - refGradient keyword, U-138
  - refineHexMesh utility, U-93
  - refinementRegions keyword, U-154
  - refinementLevel utility, U-93
  - refinementRegions keyword, U-152, U-154
  - refinementSurfaces keyword, U-152, U-153
  - refineMesh utility, U-92
  - refineWallLayer utility, U-93
  - Refresh Times button, U-25, U-171
  - regions keyword, U-60
  - relative tolerance, U-124
  - relativeSizes keyword, U-157
  - relaxed keyword, U-158
  - relTo1 keyword, U-53, U-124
  - removeFaces utility, U-93
  - Render View window, U-174
  - Render View window panel, U-173, U-174
  - renumberMesh utility, U-92
  - Rescale to Data Range button, U-25
  - Reset button, U-170
  - resolveFeatureAngle keyword, U-152, U-153
  - restart, U-39
  - Reynolds number, U-17, U-21
  - rhoPorousSimpleFoam solver, U-86
  - rhoReactingBuoyantFoam solver, U-88
  - rhoCentralDyMFoam solver, U-86
  - rhoCentralFoam solver, U-86
  - rhoConst model, U-101, U-195
  - rhoPimpleFoam solver, U-86
  - rhoReactingFoam solver, U-88

rhoReactionThermo model, U-100, U-192  
 rhoSimpleFoam solver, U-86  
 rhoSimplecFoam solver, U-86  
 rhoThermo model, U-192  
 rmdepall script/alias, U-77  
 RNGkEpsilon model, U-102, U-103  
 roots keyword, U-83, U-84  
 rotateMesh utility, U-92  
 run  
   parallel, U-81  
 run directory, U-105  
 runTime  
   keyword entry, U-32, U-114  
 runTimeModifiable keyword, U-115

## S

sammToFoam utility, U-91  
 sample utility, U-95, U-184  
 sampling  
   library, U-98  
 Save Animation  
   menu entry, U-175  
 Save Screenshot  
   menu entry, U-175  
 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-162  
 scaleSimilarity model, U-103  
 scheduled  
   keyword entry, U-80  
 scientific  
   keyword entry, U-114  
 scotch  
   keyword entry, U-82, U-83  
 scotchCoeffs keyword, U-83  
 scotchDecomp model, U-100  
 script/alias  
   find, U-179  
   foamCorrectVrt, U-164  
   foamJob, U-188  
   foamLog, U-188  
   make, U-71  
   rmdepall, U-77  
   wclean, U-76  
   wmake, U-71  
 second time derivative, P-35  
 Seed window, U-175  
 selectCells utility, U-93  
 Set Ambient Color button, U-173  
 setFields utility, U-60, U-90  
 setFormat keyword, U-185  
 sets keyword, U-185  
 setSet utility, U-92  
 setsToZones utility, U-92  
 Settings  
   menu entry, U-174  
 SFCD  
   keyword entry, U-119, U-122  
 shallowWaterFoam solver, U-86  
 shape, U-143  
 SI units, U-110  
 simple  
   keyword entry, U-82, U-83  
 simpleFilter model, U-103  
 simpleFoam solver, P-53, U-86  
 simpleGrading keyword, U-143  
 simulationType keyword, U-41, U-61, U-197  
 singleCellMesh utility, U-92  
 singleStepReactingMixture model, U-100  
 singleStepReactingMixture keyword, U-193  
 skew  
   tensor member function, P-23  
 skewLinear  
   keyword entry, U-119, U-122  
 SLGThermo  
   library, U-102  
 slice class, P-29  
 slip  
   boundary condition, U-139  
 Smagorinsky model, U-103, U-104  
 Smagorinsky2 model, U-103  
 smapToFoam utility, U-93  
 smoothDelta model, U-103  
 smoother keyword, U-126  
 smoothSolver  
   keyword entry, U-124  
 snap keyword, U-150  
 snapControls keyword, U-150  
 snappyHexMesh utility  
   background mesh, U-151  
   cell removal, U-154  
   cell splitting, U-152  
   mesh layers, U-155  
   meshing process, U-149  
   snapping to surfaces, U-155  
 snappyHexMesh utility, U-91, U-149

- snappyHexMeshDict* file, U-150
- snGrad
  - fvc member function, P-35
- snGradCorrection
  - fvc member function, P-35
- snGradSchemes keyword, U-116
- Solid Color
  - menu entry, U-173
- solidChemistryModel
  - library, U-102
- solidDisplacementFoam solver, U-89
- solidDisplacementFoam solver, U-51
- solidEquilibriumDisplacementFoam solver, U-89
- solidMixtureProperties
  - library, U-102
- solidParticle
  - library, U-99
- solidProperties
  - library, U-102
- solidSpecie
  - library, U-102
- solidThermo
  - library, U-102
- solver
  - DPMFoam, U-89
  - PDRFoam, U-88
  - XiFoam, U-88
  - adjointShapeOptimizationFoam, U-86
  - blockMesh, P-45
  - boundaryFoam, U-86
  - buoyantBoussinesqPimpleFoam, U-88
  - buoyantBoussinesqSimpleFoam, U-88
  - buoyantPimpleFoam, U-88
  - buoyantSimpleFoam, U-88
  - cavitatingDyMFoam, U-87
  - cavitatingFoam, U-87
  - chemFoam, U-88
  - chtMultiRegionFoam, U-88
  - chtMultiRegionSimpleFoam, U-88
  - coalChemistryFoam, U-89
  - coldEngineFoam, U-88
  - compressibleInterDyMFoam, U-87
  - compressibleInterFoam, U-87
  - compressibleMultiphaseInterFoam, U-87
  - dnsFoam, U-88
  - driftFluxFoam, U-87
  - dsmcFoam, U-89
  - electrostaticFoam, U-89
  - engineFoam, U-88
  - financialFoam, U-90
  - fireFoam, U-88
  - icoFoam, U-17, U-21, U-22, U-24, U-86
  - icoUncoupledKinematicParcelFoam, U-89
  - interFoam, U-87
  - interMixingFoam, U-87
  - interPhaseChangeDyMFoam, U-87
  - interPhaseChangeFoam, U-87
  - laplacianFoam, U-86
  - magneticFoam, U-89
  - mdEquilibrationFoam, U-89
  - mdFoam, U-89
  - mhdFoam, P-69, U-89
  - multiphaseEulerFoam, U-87
  - multiphaseInterFoam, U-87
  - nonNewtonianIcoFoam, U-86
  - pimpleFoam, U-86
  - pisoFoam, U-17, U-86
  - potentialFoam, P-44, U-86
  - potentialFreeSurfaceFoam, U-87
  - reactingEulerFoam, U-88
  - reactingFoam, U-88
  - reactingParcelFilmFoam, U-89
  - reactingParcelFoam, U-89
  - rhoCentralDyMFoam, U-86
  - rhoCentralFoam, U-86
  - rhoPimpleFoam, U-86
  - rhoReactingFoam, U-88
  - rhoSimpleFoam, U-86
  - rhoSimplecFoam, U-86
  - rhoPorousSimpleFoam, U-86
  - rhoReactingBuoyantFoam, U-88
  - scalarTransportFoam, U-86
  - shallowWaterFoam, U-86
  - simpleFoam, P-53, U-86
  - solidDisplacementFoam, U-89
  - solidDisplacementFoam, U-51
  - solidEquilibriumDisplacementFoam, U-89
  - sonicDyMFoam, U-86
  - sonicFoam, P-59, U-87
  - sonicLiquidFoam, P-63, U-87
  - sprayFoam, U-89
  - thermoFoam, U-88
  - twoLiquidMixingFoam, U-88
  - twoPhaseEulerFoam, U-88
  - uncoupledKinematicParcelFoam, U-89
- solver keyword, U-53, U-123
- solver relative tolerance, U-124
- solver tolerance, U-124
- solvers keyword, U-123
- sonicDyMFoam solver, U-86

- sonicFoam solver, P-59, U-87
- sonicLiquidFoam solver, P-63, U-87
- source, P-35
- SpalartAllmaras model, U-102–U-104
- SpalartAllmarasDDES model, U-104
- SpalartAllmarasIDDES model, U-104
- specie
  - library, U-101
- specie keyword, U-196
- specieThermo model, U-101
- spectEddyVisc model, U-103
- spline
  - keyword entry, U-142
- spline keyword, U-141
- splitCells utility, U-93
- splitMesh utility, U-92
- splitMeshRegions utility, U-92
- spray
  - library, U-99
- sprayFoam solver, U-89
- sqr
  - tensor member function, P-23
- sqrGradGrad
  - fvc member function, P-35
- star3ToFoam utility, U-91
- star4ToFoam utility, U-91
- startFace keyword, U-132
- startFrom keyword, U-22, U-113
- starToFoam utility, U-159
- startTime
  - keyword entry, U-22, U-113
- startTime keyword, U-22, U-113
- steady flow
  - turbulent, P-50
- steadyParticleTracks utility, U-95
- steadyState
  - keyword entry, U-122
- Stereolithography (STL), U-149
- stitchMesh utility, U-92
- stl
  - keyword entry, U-185
- stopAt keyword, U-113
- strategy keyword, U-82, U-83
- streamFunction utility, U-94
- stress analysis of plate with hole, U-46
- stressComponents utility, U-94
- Style window panel, U-173
- Su
  - fvm member function, P-35
- subsetMesh utility, U-93
- summation convention, P-15
- SUPERBEE differencing, P-36
- supersonic flow, P-58
- supersonic flow over forward step, P-58
- supersonicFreeStream
  - boundary condition, U-139
- surfaceLambdaMuSmooth utility, U-96
- surface mesh, U-149
- surfaceAdd utility, U-95
- surfaceAutoPatch utility, U-95
- surfaceBooleanFeatures utility, U-95
- surfaceCheck utility, U-95
- surfaceClean utility, U-95
- surfaceCoarsen utility, U-95
- surfaceConvert utility, U-95
- surfaceFeatureConvert utility, U-95
- surfaceFeatureExtract utility, U-95, U-153
- surfaceField<Type> template class, P-31
- surfaceFilmModels
  - library, U-104
- surfaceFind utility, U-95
- surfaceFormat keyword, U-185
- surfaceHookUp utility, U-95
- surfaceInertia utility, U-96
- surfaceMesh tools, U-98
- surfaceMeshConvert utility, U-96
- surfaceMeshConvertTesting utility, U-96
- surfaceMeshExport utility, U-96
- surfaceMeshImport utility, U-96
- surfaceMeshInfo utility, U-96
- surfaceMeshTriangulate utility, U-96
- surfaceNormalFixedValue
  - boundary condition, U-139
- surfaceOrient utility, U-96
- surfacePointMerge utility, U-96
- surfaceRedistributePar utility, U-96
- surfaceRefineRedGreen utility, U-96
- surfaces keyword, U-185
- surfaceSplitByPatch utility, U-96
- surfaceSplitByTopology utility, U-96
- surfaceSplitNonManifolds utility, U-96
- surfaceSubset utility, U-96
- surfaceToPatch utility, U-96
- surfaceTransformPoints utility, U-96
- surfMesh
  - library, U-99
- SuSp
  - fvm member function, P-35
- sutherlandTransport model, U-101
- symm

tensor member function, P-23  
 symmetryPlane  
   boundary condition, P-63, U-137  
 symmetryPlane  
   keyword entry, U-137  
 symmTensorField class, P-27  
 symmTensorThirdField class, P-27  
 system directory, P-48, U-105  
 systemCall  
   library, U-99

## T

T()

  tensor member function, P-23

Tcommon keyword, U-194

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-95

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

  nth 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-91
- text box
  - Opacity, U-173
- thermalProperties*
  - dictionary, U-51
- thermodynamics keyword, U-196
- thermoFoam solver, U-88
- thermophysical
  - library, U-191
- thermophysicalFunctions
  - library, U-101
- thermophysicalProperties*
  - dictionary, U-191
- thermoType keyword, U-191
- Thigh keyword, U-194
- time
  - control, U-113
- time derivative, P-35
  - first, P-37
  - second, P-35, P-37
- time step, U-22
- timeFormat keyword, U-114
- timePrecision keyword, U-115
- timeScheme keyword, U-116
- timeStamp
  - keyword entry, U-80
- timeStampMaster
  - keyword entry, U-80
- timeStep
  - keyword entry, U-23, U-32, U-114, U-179
- Tlow keyword, U-194
- tolerance
  - solver, U-124
  - solver relative, U-124
- tolerance keyword, U-53, U-124, U-155
- Toolbars
  - menu entry, U-173
- tools
  - algorithms, U-98
  - cfTools, U-98
  - containers, U-98
  - db, U-98
  - dimensionSet, U-98
  - dimensionedTypes, U-98
  - fields, U-98
  - finiteVolume, U-98
  - fvMatrices, U-98
  - fvMesh, U-98
  - global, U-98
  - graph, U-98
  - interpolations, U-98
  - interpolation, U-98
  - matrices, U-98
  - memory, U-98
  - meshes, U-98
  - primitives, U-98
  - surfaceMesh, U-98
  - volMesh, U-98
- topoChangerFvMesh
  - library, U-99
- topoSet utility, U-93
- topoSetSource keyword, U-60
- totalPressure
  - boundary condition, U-139
- tr
  - tensor member function, P-23
- trace, *see* tensor, trace
- traction keyword, U-51
- transform
  - tensor member function, P-23
- transformPoints utility, U-93
- transport keyword, U-192, U-196
- transportProperties*
  - dictionary, U-21, U-39, U-42, U-199
- transportProperties* file, U-60
- triple inner product, P-17
- triSurface
  - library, U-99
- Ts keyword, U-193
- turbulence
  - dissipation, U-40
  - kinetic energy, U-40
  - length scale, U-41
- turbulence
  - library, U-99
- turbulence keyword, U-198
- turbulence model
  - RAS, U-40
- turbulenceProperties*
  - dictionary, U-41, U-61, U-197
- turbulent flow
  - steady, P-50
- turbulentInlet

- boundary condition, U-139
- tutorials
  - breaking of a dam, U-56
  - lid-driven cavity flow, U-17
  - stress analysis of plate with hole, U-46
- tutorials* directory, P-43, U-17
- twoLiquidMixingFoam solver, U-88
- twoPhaseEulerFoam solver, U-88
- twoPhaseProperties
  - library, U-104
- type keyword, U-135, U-192

## U

- U field, U-23
- Ucomponents utility, P-70
- UMIST
  - keyword entry, U-117
- uncompressed
  - keyword entry, U-114
- uncorrected
  - keyword entry, U-120, U-121
- uncoupledKinematicParcelFoam solver, U-89
- uniform keyword, U-187
- units
  - base, U-110
  - of measurement, P-24, U-109
  - S.I. base, P-24
  - SI, U-110
  - Système International, U-110
  - United States Customary System, U-110
  - USCS, U-110
- Update GUI button, U-171
- uprime utility, U-94
- upwind
  - keyword entry, U-119, U-122
- upwind differencing, P-36, U-63
- USCS units, U-110
- Use Parallel Projection button, U-24
- utility
  - Co, U-93
  - Lambda2, U-94
  - Mach, U-94
  - PDRMesh, U-93
  - Pe, U-94
  - Q, U-94
  - R, U-94
  - Ucomponents, P-70
  - adiabaticFlameT, U-97
  - ansysToFoam, U-91
  - applyBoundaryLayer, U-90
  - applyWallFunctionBoundaryConditions, U-90
  - attachMesh, U-92
  - autoPatch, U-92
  - autoRefineMesh, U-93
  - blockMesh, U-38, U-91, U-140
  - boxTurb, U-90
  - cfx4ToFoam, U-91, U-159
  - changeDictionary, U-90
  - checkMesh, U-92, U-160
  - chemkinToFoam, U-97
  - collapseEdges, U-93
  - combinePatchFaces, U-93
  - createBaffles, U-92
  - createPatch, U-92
  - createTurbulenceFields, U-94
  - createExternalCoupledPatchGeometry, U-90
  - datToFoam, U-91
  - decomposePar, U-82, U-83, U-97
  - deformedGeom, U-92
  - dsmcFieldsCalc, U-95
  - dsmcInitialise, U-90
  - engineCompRatio, U-95
  - engineSwirl, U-90
  - ensight74FoamExec, U-183
  - ensightFoamReader, U-93
  - enstrophy, U-93
  - equilibriumCO, U-97
  - equilibriumFlameT, U-97
  - execFlowFunctionObjects, U-95
  - expandDictionary, U-97
  - extrude2DMesh, U-91
  - extrudeMesh, U-91
  - extrudeToRegionMesh, U-91
  - faceAgglomerate, U-90
  - flattenMesh, U-92
  - flowType, U-93
  - fluent3DMeshToFoam, U-91
  - fluentMeshToFoam, U-91, U-159
  - foamCalc, U-33
  - foamDataToFluent, U-93, U-181
  - foamDebugSwitches, U-97
  - foamFormatConvert, U-97
  - foamHelp, U-97
  - foamInfoExec, U-97
  - foamListTimes, U-95
  - foamMeshToFluent, U-91, U-181
  - foamToEnightParts, U-93
  - foamToEnight, U-93
  - foamToGMV, U-93
  - foamToStarMesh, U-91
  - foamToSurface, U-91

- foamToTecplot360, U-93
- foamToTetDualMesh, U-93
- foamToVTK, U-93
- foamUpgradeCyclics, U-90
- foamUpgradeFvSolution, U-90
- foamyHexMesh, U-91
- foamyQuadMesh, U-91
- foamyHexMeshBackgroundMesh, U-91
- foamyHexMeshSurfaceSimplify, U-91
- gambitToFoam, U-91, U-159
- gmshToFoam, U-91
- ideasToFoam, U-159
- ideasUnvToFoam, U-91
- insideCells, U-92
- kivaToFoam, U-91
- mapFieldsPar, U-90
- mapFields, U-31, U-38, U-42, U-56, U-90, U-165
- mdlInitialise, U-90
- mergeMeshes, U-92
- mergeOrSplitBaffles, U-92
- mirrorMesh, U-92
- mixtureAdiabaticFlameT, U-97
- modifyMesh, U-93
- moveDynamicMesh, U-92
- moveEngineMesh, U-92
- moveMesh, U-92
- mshToFoam, U-91
- netgenNeutralToFoam, U-91
- objToVTK, U-92
- orientFaceZone, U-92
- pPrime2, U-94
- particleTracks, U-95
- patchAverage, U-94
- patchIntegrate, U-94
- patchSummary, U-97
- pdfPlot, U-95
- plot3dToFoam, U-91
- polyDualMesh, U-92
- postChannel, U-95
- probeLocations, U-95
- ptot, U-95
- reconstructParMesh, U-97
- reconstructPar, U-85, U-97
- redistributePar, U-97
- refineHexMesh, U-93
- refineMesh, U-92
- refineWallLayer, U-93
- refinementLevel, U-93
- removeFaces, U-93
- renumberMesh, U-92
- rotateMesh, U-92
- sammToFoam, U-91
- sample, U-95, U-184
- scalePoints, U-162
- selectCells, U-93
- setFields, U-60, U-90
- setSet, U-92
- setsToZones, U-92
- singleCellMesh, U-92
- smapToFoam, U-93
- snappyHexMesh, U-91, U-149
- splitCells, U-93
- splitMeshRegions, U-92
- splitMesh, U-92
- star3ToFoam, U-91
- star4ToFoam, U-91
- starToFoam, U-159
- steadyParticleTracks, U-95
- stitchMesh, U-92
- streamFunction, U-94
- stressComponents, U-94
- subsetMesh, U-93
- surfaceLambdaMuSmooth, U-96
- surfaceAdd, U-95
- surfaceAutoPatch, U-95
- surfaceBooleanFeatures, U-95
- surfaceCheck, U-95
- surfaceClean, U-95
- surfaceCoarsen, U-95
- surfaceConvert, U-95
- surfaceFeatureConvert, U-95
- surfaceFeatureExtract, U-95, U-153
- surfaceFind, U-95
- surfaceHookUp, U-95
- surfaceInertia, U-96
- surfaceMeshConvertTesting, U-96
- surfaceMeshConvert, U-96
- surfaceMeshExport, U-96
- surfaceMeshImport, U-96
- surfaceMeshInfo, U-96
- surfaceMeshTriangulate, U-96
- surfaceOrient, U-96
- surfacePointMerge, U-96
- surfaceRedistributePar, U-96
- surfaceRefineRedGreen, U-96
- surfaceSplitByPatch, U-96
- surfaceSplitByTopology, U-96
- surfaceSplitNonManifolds, U-96
- surfaceSubset, U-96

surfaceToPatch, U-96  
 surfaceTransformPoints, U-96  
 temporalInterpolate, U-95  
 tetgenToFoam, U-91  
 topoSet, U-93  
 transformPoints, U-93  
 uprime, U-94  
 viewFactorsGen, U-90  
 vorticity, U-94  
 vtkUnstructuredToFoam, U-92  
 wallFunctionTable, U-90  
 wallGradU, U-94  
 wallHeatFlux, U-94  
 wallShearStress, U-94  
 wdot, U-95  
 writeCellCentres, U-95  
 writeMeshObj, U-92  
 yPlus, U-94  
 zipUpMesh, U-93  
 utilityFunctionObjects  
   library, U-99

## V

v2f model, U-103  
 value keyword, U-21, U-138  
 valueFraction keyword, U-138  
 van Leer differencing, P-36  
 vanDriestDelta model, U-104  
 vanLeer  
   keyword entry, U-119  
 VCR Controls menu, U-25, U-171  
 vector, P-14  
   operator, P-25  
   unit, P-18  
 vector class, P-22, U-109  
 vector product, *see* tensor, vector cross product  
 vectorField class, P-27  
 version keyword, U-107  
 vertices keyword, U-20, U-141  
 veryInhomogeneousMixture model, U-100  
 veryInhomogeneousMixture keyword, U-193  
 View menu, U-170, U-173  
 View Render window panel, U-24  
 View Settings  
   menu entry, U-24, U-173  
 viewFactor  
   library, U-101  
 viewFactorsGen utility, U-90  
 viscosity  
   kinematic, U-22, U-42  
 volField<Type> template class, P-31  
 volMesh tools, U-98  
 vorticity utility, U-94  
 vtk  
   keyword entry, U-185  
 vtkPV4Foam  
   library, U-169  
 vtkUnstructuredToFoam utility, U-92

## W

wall  
   boundary condition, P-63, P-69, U-59, U-137  
 wall  
   keyword entry, U-137  
 wallFunctionTable utility, U-90  
 wallGradU utility, U-94  
 wallHeatFlux utility, U-94  
 Wallis  
   library, U-101  
 wallShearStress utility, U-94  
 wclean script/alias, U-76  
 wdot utility, U-95  
 wedge  
   boundary condition, U-133, U-137, U-148  
 wedge  
   keyword entry, U-137  
 window  
   *Color Legend*, U-29  
   *Options*, U-174  
   *Pipeline Browser*, U-24, U-170  
   *Properties*, U-171, U-172  
   *Render View*, U-174  
   *Seed*, U-175  
 window panel  
   *Animations*, U-174  
   *Annotation*, U-24  
   *Charts*, U-174  
   *Color Legend*, U-173  
   *Color Scale*, U-173  
   *Colors*, U-174  
   *Display*, U-24, U-25, U-170, U-172  
   *General*, U-174  
   *Information*, U-170  
   *Mesh Parts*, U-24  
   *Parameters*, U-171  
   *Properties*, U-25, U-170  
   *Render View*, U-173, U-174  
   *Style*, U-173  
   *View Render*, U-24  
 Wireframe  
   menu entry, U-173  
 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-95  
writeCompression keyword, U-114  
writeControl  
keyword entry, U-114  
writeControl keyword, U-22, U-62, U-114  
writeFormat keyword, U-55, U-114  
writeInterval  
keyword entry, U-179  
writeInterval keyword, U-23, U-32, U-114  
writeMeshObj utility, U-92  
writeNow  
keyword entry, U-113  
writePrecision keyword, U-114

## X

x  
keyword entry, U-187  
XiFoam solver, U-88  
xmgr  
keyword entry, U-115, U-185  
xyz  
keyword entry, U-187

## Y

y  
keyword entry, U-187  
yPlus utility, U-94

## Z

z  
keyword entry, U-187  
zeroGradient  
boundary condition, U-138  
zipUpMesh utility, U-93