



User Guide

version 5.0

24th July 2017

<http://openfoam.org>

Copyright © 2011-2017 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^AT_EX.

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 HEREIN 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 (“syncing”) 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.

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 CD-Adapco.

UNIX is a registered trademark of The Open Group.

Contents

Copyright Notice	U-2
1. Definitions	U-2
2. Fair Dealing Rights.	U-3
3. License Grant.	U-3
4. Restrictions.	U-4
5. Representations, Warranties and Disclaimer	U-5
6. Limitation on Liability.	U-5
7. Termination	U-5
8. Miscellaneous	U-6
Trademarks	U-7
Contents	U-9
1 Introduction	U-17
2 Tutorials	U-19
2.1 Lid-driven cavity flow	U-20
2.1.1 Pre-processing	U-20
2.1.1.1 Mesh generation	U-21
2.1.1.2 Boundary and initial conditions	U-23
2.1.1.3 Physical properties	U-24
2.1.1.4 Control	U-24
2.1.1.5 Discretisation and linear-solver settings	U-25
2.1.2 Viewing the mesh	U-26
2.1.3 Running an application	U-28
2.1.4 Post-processing	U-28
2.1.4.1 Colouring surfaces	U-28
2.1.4.2 Cutting plane (slice)	U-31
2.1.4.3 Contours	U-31
2.1.4.4 Vector plots	U-31
2.1.4.5 Streamline plots	U-33
2.1.5 Increasing the mesh resolution	U-33
2.1.5.1 Creating a new case using an existing case	U-33
2.1.5.2 Creating the finer mesh	U-35
2.1.5.3 Mapping the coarse mesh results onto the fine mesh . .	U-35
2.1.5.4 Control adjustments	U-36
2.1.5.5 Running the code as a background process	U-36

2.1.5.6	Vector plot with the refined mesh	U-36
2.1.5.7	Plotting graphs	U-37
2.1.6	Introducing mesh grading	U-39
2.1.6.1	Creating the graded mesh	U-39
2.1.6.2	Changing time and time step	U-41
2.1.6.3	Mapping fields	U-42
2.1.7	Increasing the Reynolds number	U-42
2.1.7.1	Pre-processing	U-42
2.1.7.2	Running the code	U-43
2.1.8	High Reynolds number flow	U-43
2.1.8.1	Pre-processing	U-44
2.1.8.2	Running the code	U-46
2.1.9	Changing the case geometry	U-46
2.1.10	Post-processing the modified geometry	U-50
2.2	Stress analysis of a plate with a hole	U-50
2.2.1	Mesh generation	U-51
2.2.1.1	Boundary and initial conditions	U-54
2.2.1.2	Mechanical properties	U-55
2.2.1.3	Thermal properties	U-55
2.2.1.4	Control	U-55
2.2.1.5	Discretisation schemes and linear-solver control	U-56
2.2.2	Running the code	U-58
2.2.3	Post-processing	U-58
2.2.4	Exercises	U-59
2.2.4.1	Increasing mesh resolution	U-59
2.2.4.2	Introducing mesh grading	U-60
2.2.4.3	Changing the plate size	U-60
2.3	Breaking of a dam	U-60
2.3.1	Mesh generation	U-61
2.3.2	Boundary conditions	U-63
2.3.3	Setting initial field	U-63
2.3.4	Fluid properties	U-64
2.3.5	Turbulence modelling	U-65
2.3.6	Time step control	U-66
2.3.7	Discretisation schemes	U-67
2.3.8	Linear-solver control	U-67
2.3.9	Running the code	U-68
2.3.10	Post-processing	U-68
2.3.11	Running in parallel	U-68
2.3.12	Post-processing a case run in parallel	U-71
3	Applications and libraries	U-73
3.1	The programming language of OpenFOAM	U-73
3.1.1	Language in general	U-73
3.1.2	Object-orientation and C++	U-74
3.1.3	Equation representation	U-74
3.1.4	Solver codes	U-75
3.2	Compiling applications and libraries	U-75

3.2.1	Header <i>H</i> files	U-75
3.2.2	Compiling with <code>wmake</code>	U-77
3.2.2.1	Including headers	U-77
3.2.2.2	Linking to libraries	U-78
3.2.2.3	Source files to be compiled	U-79
3.2.2.4	Running <code>wmake</code>	U-79
3.2.2.5	<code>wmake</code> environment variables	U-79
3.2.3	Removing dependency lists: <code>wclean</code>	U-79
3.2.4	Compiling libraries	U-81
3.2.5	Compilation example: the <code>pisoFoam</code> application	U-81
3.2.6	Debug messaging and optimisation switches	U-84
3.2.7	Linking new user-defined libraries to existing applications	U-85
3.3	Running applications	U-85
3.4	Running applications in parallel	U-86
3.4.1	Decomposition of mesh and initial field data	U-86
3.4.2	File input/output in parallel	U-87
3.4.2.1	Selecting the file handler	U-89
3.4.2.2	Updating existing files	U-89
3.4.2.3	Threading support	U-89
3.4.3	Running a decomposed case	U-90
3.4.4	Distributing data across several disks	U-90
3.4.5	Post-processing parallel processed cases	U-91
3.4.5.1	Reconstructing mesh and data	U-91
3.4.5.2	Post-processing decomposed cases	U-91
3.5	Standard solvers	U-91
3.5.1	'Basic' CFD codes	U-92
3.5.2	Incompressible flow	U-92
3.5.3	Compressible flow	U-92
3.5.4	Multiphase flow	U-93
3.5.5	Direct numerical simulation (DNS)	U-94
3.5.6	Combustion	U-95
3.5.7	Heat transfer and buoyancy-driven flows	U-95
3.5.8	Particle-tracking flows	U-96
3.5.9	Discrete methods	U-97
3.5.10	Electromagnetics	U-97
3.5.11	Stress analysis of solids	U-97
3.5.12	Finance	U-97
3.6	Standard utilities	U-97
3.6.1	Pre-processing	U-97
3.6.2	Mesh generation	U-98
3.6.3	Mesh conversion	U-98
3.6.4	Mesh manipulation	U-99
3.6.5	Other mesh tools	U-101
3.6.6	Post-processing	U-101
3.6.7	Post-processing data converters	U-102
3.6.8	Surface mesh (e.g. OBJ/STL) tools	U-102
3.6.9	Parallel processing	U-103

3.6.10 Thermophysical-related utilities	U-104
3.6.11 Miscellaneous utilities	U-104
4 OpenFOAM cases	U-105
4.1 File structure of OpenFOAM cases	U-105
4.2 Basic input/output file format	U-106
4.2.1 General syntax rules	U-106
4.2.2 Dictionaries	U-107
4.2.3 The data file header	U-107
4.2.4 Lists	U-108
4.2.5 Scalars, vectors and tensors	U-109
4.2.6 Dimensional units	U-109
4.2.7 Dimensioned types	U-110
4.2.8 Fields	U-110
4.2.9 Macro expansion	U-111
4.2.10 Including files	U-112
4.2.11 Regular expressions	U-113
4.2.12 Keyword ordering	U-114
4.2.13 Inline calculations and code	U-114
4.3 Time and data input/output control	U-115
4.3.1 Time control	U-116
4.3.2 Data writing	U-116
4.3.3 Other settings	U-117
4.4 Numerical schemes	U-118
4.4.1 Time schemes	U-120
4.4.2 Gradient schemes	U-120
4.4.3 Divergence schemes	U-121
4.4.4 Surface normal gradient schemes	U-123
4.4.5 Laplacian schemes	U-124
4.4.6 Interpolation schemes	U-125
4.5 Solution and algorithm control	U-125
4.5.1 Linear solver control	U-126
4.5.1.1 Solution tolerances	U-127
4.5.1.2 Preconditioned conjugate gradient solvers	U-128
4.5.1.3 Smooth solvers	U-128
4.5.1.4 Geometric-algebraic multi-grid solvers	U-129
4.5.2 Solution under-relaxation	U-130
4.5.3 PISO, SIMPLE and PIMPLE algorithms	U-131
4.5.4 Pressure referencing	U-131
4.5.5 Other parameters	U-131
4.6 Case management tools	U-132
4.6.1 File management scripts	U-132
4.6.2 The <code>foamDictionary</code> tool	U-132
5 Mesh generation and conversion	U-135
5.1 Mesh description	U-135
5.1.1 Mesh specification and validity constraints	U-135
5.1.1.1 Points	U-135

5.1.1.2	Faces	U-136
5.1.1.3	Cells	U-136
5.1.1.4	Boundary	U-137
5.1.2	The <code>polyMesh</code> description	U-137
5.1.3	Cell shapes	U-138
5.1.4	1- and 2-dimensional and axi-symmetric problems	U-138
5.2	Boundaries	U-138
5.2.1	Geometric (constraint) patch types	U-141
5.2.2	Basic boundary conditions	U-142
5.2.3	Derived types	U-143
5.2.3.1	The inlet/outlet condition	U-143
5.2.3.2	Entrainment boundary conditions	U-144
5.2.3.3	Fixed flux pressure	U-145
5.2.3.4	Time-varying boundary conditions	U-146
5.3	Mesh generation with the <code>blockMesh</code> utility	U-147
5.3.1	Writing a <code>blockMeshDict</code> file	U-148
5.3.1.1	The <code>vertices</code>	U-149
5.3.1.2	The <code>edges</code>	U-150
5.3.1.3	The <code>blocks</code>	U-150
5.3.1.4	Multi-grading of a block	U-151
5.3.1.5	The <code>boundary</code>	U-153
5.3.2	Multiple blocks	U-154
5.3.3	Projection of vertices, edges and faces	U-156
5.3.4	Naming vertices, edges, faces and blocks	U-157
5.3.5	Creating blocks with fewer than 8 vertices	U-157
5.3.6	Running <code>blockMesh</code>	U-157
5.4	Mesh generation with the <code>snappyHexMesh</code> utility	U-158
5.4.1	The mesh generation process of <code>snappyHexMesh</code>	U-159
5.4.2	Creating the background hex mesh	U-160
5.4.3	Cell splitting at feature edges and surfaces	U-160
5.4.4	Cell removal	U-162
5.4.5	Cell splitting in specified regions	U-163
5.4.6	Snapping to surfaces	U-163
5.4.7	Mesh layers	U-164
5.4.8	Mesh quality controls	U-167
5.5	Mesh conversion	U-168
5.5.1	<code>fluentMeshToFoam</code>	U-168
5.5.2	<code>starToFoam</code>	U-169
5.5.2.1	General advice on conversion	U-169
5.5.2.2	Eliminating extraneous data	U-169
5.5.2.3	Removing default boundary conditions	U-170
5.5.2.4	Renumbering the model	U-171
5.5.2.5	Writing out the mesh data	U-171
5.5.2.6	Problems with the <code>.vrt</code> file	U-172
5.5.2.7	Converting the mesh to OpenFOAM format	U-173
5.5.3	<code>gambitToFoam</code>	U-173
5.5.4	<code>ideasToFoam</code>	U-173

5.5.5	<code>cfx4ToFoam</code>	U-173
5.6	Mapping fields between different geometries	U-174
5.6.1	Mapping consistent fields	U-174
5.6.2	Mapping inconsistent fields	U-174
5.6.3	Mapping parallel cases	U-175
6	Post-processing	U-177
6.1	ParaView/paraFoam graphical user interface (GUI)	U-177
6.1.1	Overview of ParaView/paraFoam	U-177
6.1.2	The Parameters panel	U-179
6.1.3	The Display panel	U-180
6.1.4	The button toolbars	U-181
6.1.5	Manipulating the view	U-181
6.1.5.1	View settings	U-181
6.1.5.2	General settings	U-182
6.1.6	Contour plots	U-182
6.1.6.1	Introducing a cutting plane	U-182
6.1.7	Vector plots	U-182
6.1.7.1	Plotting at cell centres	U-183
6.1.8	Streamlines	U-183
6.1.9	Image output	U-183
6.1.10	Animation output	U-183
6.2	Post-processing command line interface (CLI)	U-184
6.2.1	Post-processing functionality	U-184
6.2.1.1	Field calculation	U-185
6.2.1.2	Flow rate calculation	U-186
6.2.1.3	Forces and force coefficients	U-186
6.2.1.4	Sampling for graph plotting	U-186
6.2.1.5	Lagrangian data	U-187
6.2.1.6	Monitoring minima and maxima	U-187
6.2.1.7	Numerical data	U-187
6.2.1.8	Pressure tools	U-187
6.2.1.9	Probes	U-187
6.2.1.10	'Pluggable' solvers	U-188
6.2.1.11	Visualisation tools	U-188
6.2.2	Run-time data processing	U-188
6.2.3	The <code>postProcess</code> utility	U-189
6.2.4	Solver post-processing	U-190
6.3	Sampling and monitoring data	U-191
6.3.1	Probing data	U-191
6.3.2	Sampling for graphs	U-192
6.3.3	Sampling for visualisation	U-193
6.3.4	Live monitoring of data	U-195
6.4	Third-Party post-processing	U-195
6.4.1	Post-processing with EnSight	U-196
6.4.1.1	Converting data to EnSight format	U-196
6.4.1.2	The <code>ensightFoamReader</code> reader module	U-197

7 Models and physical properties	U-199
7.1 Thermophysical models	U-199
7.1.1 Thermophysical and mixture models	U-200
7.1.2 Transport model	U-201
7.1.3 Thermodynamic models	U-202
7.1.4 Composition of each constituent	U-202
7.1.5 Equation of state	U-203
7.1.6 Selection of energy variable	U-204
7.1.7 Thermophysical property data	U-204
7.2 Turbulence models	U-205
7.2.1 Reynolds-averaged simulation (RAS) modelling	U-206
7.2.1.1 Incompressible RAS turbulence models	U-206
7.2.1.2 Compressible RAS turbulence models	U-207
7.2.2 Large eddy simulation (LES) modelling	U-208
7.2.2.1 Incompressible LES turbulence models	U-208
7.2.2.2 Compressible LES turbulence models	U-209
7.2.3 Model coefficients	U-210
7.2.4 Wall functions	U-210
7.3 Transport/rheology models	U-211
7.3.1 Newtonian model	U-211
7.3.2 Bird-Carreau model	U-211
7.3.3 Cross Power Law model	U-211
7.3.4 Power Law model	U-212
7.3.5 Herschel-Bulkley model	U-212
7.3.6 Casson model	U-212
7.3.7 General strain-rate function	U-213

Chapter 1

Introduction

This guide accompanies the release of version 5.0 of the Open Source Field Operation and Manipulation (OpenFOAM) C++ libraries. It provides a description of the basic operation of OpenFOAM, first through a set of tutorial exercises in chapter 2 and later by a more detailed description of the individual components that make up OpenFOAM.

OpenFOAM is a framework for developing *application* executables that use packaged functionality contained within a collection of approximately 100 *C+ libraries*. OpenFOAM is shipped with approximately 250 pre-built applications that fall into two categories: *solvers*, that are each designed to solve a specific problem in fluid (or continuum) mechanics; and *utilities*, that are designed to perform tasks that involve data manipulation. The solvers in OpenFOAM cover a wide range of problems in fluid dynamics, as described in chapter 3.

Users can extend the collection of solvers, utilities and libraries in OpenFOAM, using some pre-requisite knowledge of the underlying method, physics and programming techniques involved.

OpenFOAM is supplied with pre- and post-processing environments. The interface to the pre- and post-processing are themselves OpenFOAM utilities, thereby ensuring consistent data handling across all environments. The overall structure of OpenFOAM is shown in Figure 1.1. The pre-processing and running of OpenFOAM cases is described in chapter 4.

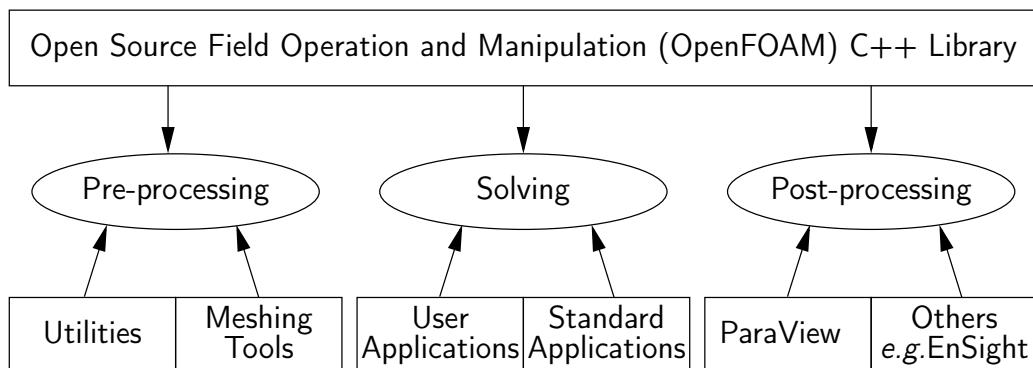


Figure 1.1: Overview of OpenFOAM structure.

In chapter 5, we cover both the generation of meshes using the mesh generator supplied with OpenFOAM and conversion of mesh data generated by third-party products. Post-processing is described in chapter 6 and some aspects of physical modelling, *e.g.* transport and thermophysical modelling, are described in chapter 7.

Chapter 2

Tutorials

In this chapter we shall describe in detail the process of setup, simulation and post-processing for some OpenFOAM test cases, with the principal aim of introducing a user to the basic procedures of running OpenFOAM. The `$FOAM_TUTORIALS` directory contains many more cases that demonstrate the use of all the solvers and many utilities supplied with OpenFOAM.

Before attempting to run the tutorials, the user must first make sure that OpenFOAM is installed correctly. Cases in the tutorials will be copied into the so-called `run` directory, an OpenFOAM project directory in the user's file system at `$HOME/OpenFOAM/<USER>/run` where `<USER>` is the account login name. The `run` directory is represented by the `$FOAM_RUN` environment variable enabling the user to check its existence conveniently by typing

```
ls $FOAM_RUN
```

If a message is returned saying no such directory exists, the user should create the directory by typing

```
mkdir -p $FOAM_RUN
```

The tutorial cases describe the use of the `blockMesh` pre-processing tool, case setup and running OpenFOAM solvers and post-processing using `ParaView`. Those users with access to third-party post-processing tools supported in OpenFOAM have an option: either they can follow the tutorials using `ParaView`; or refer to the description of the use of the third-party product in chapter 6 when post-processing is required.

Copies of all tutorials are available from the `tutorials` directory of the OpenFOAM installation. The tutorials are organised into a set of directories according to the type of flow and then subdirectories according to solver. For example, all the `icoFoam` cases are stored within a subdirectory `incompressible/icoFoam`, where `incompressible` indicates the type of flow. The user can copy cases from the `tutorials` directory into their local `run` directory as needed. For example to run the `pitzDaily` tutorial case for the `simpleFoam` solver, the user can copy it to the `run` directory by typing:

```
cd $FOAM_RUN
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
```

2.1 Lid-driven cavity flow

This tutorial will describe how to pre-process, run and post-process a case involving isothermal, incompressible flow in a two-dimensional square domain. The geometry is shown in Figure 2.1 in which all the boundaries of the square are walls. The top wall moves in the x -direction at a speed of 1 m/s while the other 3 are stationary. Initially, the flow will be assumed laminar and will be solved on a uniform mesh using the `icoFoam` solver for laminar, isothermal, incompressible flow. During the course of the tutorial, the effect of increased mesh resolution and mesh grading towards the walls will be investigated. Finally, the flow Reynolds number will be increased and the `pisoFoam` solver will be used for turbulent, isothermal, incompressible flow.

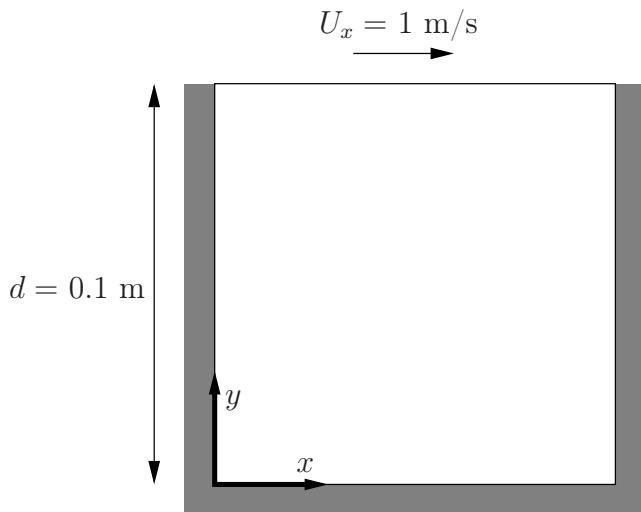


Figure 2.1: Geometry of the lid driven cavity.

2.1.1 Pre-processing

Cases are setup in OpenFOAM by editing case files. Users should select an editor of choice with which to do this, such as `emacs`, `vi`, `gedit`, `nedit`, etc. Editing files is possible in OpenFOAM because the I/O uses a dictionary format with keywords that convey sufficient meaning to be understood by the users.

A case being simulated involves data for mesh, fields, properties, control parameters, etc. As described in section 4.1, in OpenFOAM this data is stored in a set of files within a case directory rather than in a single case file, as in many other CFD packages. The case directory is given a suitably descriptive name. This tutorial consists of a set of cases located in `$FOAM_TUTORIALS/incompressible/icoFoam/cavity`, the first of which is simply named `cavity`. As a first step, the user should copy the `cavity` case directory to their `run` directory.

```
cd $FOAM_RUN
cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity/cavity .
cd cavity
```

2.1.1.1 Mesh generation

OpenFOAM always operates in a 3 dimensional Cartesian coordinate system and all geometries are generated in 3 dimensions. OpenFOAM solves the case in 3 dimensions by default but can be instructed to solve in 2 dimensions by specifying a ‘special’ `empty` boundary condition on boundaries normal to the (3rd) dimension for which no solution is required.

The `cavity` domain consists of a square of side length $d = 0.1$ m in the x - y plane. A uniform mesh of 20 by 20 cells will be used initially. The block structure is shown in Figure 2.2. The mesh generator supplied with OpenFOAM, `blockMesh`, generates meshes

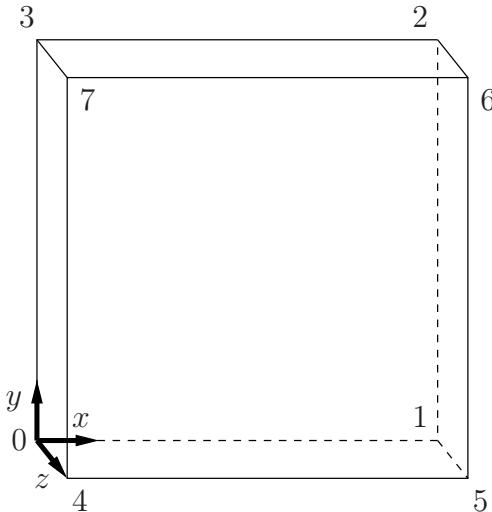


Figure 2.2: Block structure of the mesh for the cavity.

from a description specified in an input dictionary, `blockMeshDict` located in the `system` (or `constant/polyMesh`) directory for a given case. The `blockMeshDict` entries for this case are as follows:

```

1  /*----- C++ -----*/
2  =====
3  | \ \ / Field          | OpenFOAM: The Open Source CFD Toolbox
4  | \ \ / Operation      | Version: 5.0
5  | \ \ / And            | Web: www.OpenFOAM.org
6  | \ \ / Manipulation   |
7  \*-----*/
8  FoamFile
9  {
10    version    2.0;
11    format     ascii;
12    class      dictionary;
13    object     blockMeshDict;
14  } // * * * * *
15  convertToMeters 0.1;
16
17  vertices
18  (
19    (0 0 0)
20    (1 0 0)
21    (1 1 0)
22    (0 1 0)
23    (0 0 0.1)
24    (1 0 0.1)
25    (1 1 0.1)
26    (0 1 0.1)
27  );
28
29  blocks
30  (
31

```

```

33     hex (0 1 2 3 4 5 6 7) (20 20 1) simpleGrading (1 1 1)
34   );
35
36   edges
37   (
38   );
39
40   boundary
41   (
42     movingWall
43     {
44       type wall;
45       faces
46       (
47         (3 7 6 2)
48       );
49     }
50     fixedWalls
51     {
52       type wall;
53       faces
54       (
55         (0 4 7 3)
56         (2 6 5 1)
57         (1 5 4 0)
58       );
59     }
60     frontAndBack
61     {
62       type empty;
63       faces
64       (
65         (0 3 2 1)
66         (4 5 6 7)
67       );
68     }
69   );
70
71   mergePatchPairs
72   (
73   );
74 // ****

```

The file first contains header information in the form of a banner (lines 1-7), then file information contained in a *FoamFile* sub-dictionary, delimited by curly braces `{...}`.

For the remainder of the manual:

For the sake of clarity and to save space, file headers, including the banner and *FoamFile* sub-dictionary, will be removed from verbatim quoting of case files

The file first specifies coordinates of the block *vertices*; it then defines the *blocks* (here, only 1) from the vertex labels and the number of cells within it; and finally, it defines the boundary patches. The user is encouraged to consult section 5.3 to understand the meaning of the entries in the *blockMeshDict* file.

The mesh is generated by running *blockMesh* on this *blockMeshDict* file. From within the case directory, this is done, simply by typing in the terminal:

```
blockMesh
```

The running status of *blockMesh* is reported in the terminal window. Any mistakes in the *blockMeshDict* file are picked up by *blockMesh* and the resulting error message directs the user to the line in the file where the problem occurred. There should be no error messages at this stage.

2.1.1.2 Boundary and initial conditions

Once the mesh generation is complete, the user can look at this initial fields set up for this case. The case is set up to start at time $t = 0$ s, so the initial field data is stored in a θ sub-directory of the **cavity** directory. The θ sub-directory contains 2 files, **p** and **U**, one for each of the pressure (**p**) and velocity (**U**) fields whose initial values and boundary conditions must be set. Let us examine file **p**:

```

17 dimensions      [0 2 -2 0 0 0];
18 internalField   uniform 0;
19
20 boundaryField
21 {
22     movingWall
23     {
24         type          zeroGradient;
25     }
26
27     fixedWalls
28     {
29         type          zeroGradient;
30     }
31
32     frontAndBack
33     {
34         type          empty;
35     }
36 }
37
38 // ****
39

```

There are 3 principal entries in field data files:

dimensions specifies the dimensions of the field, here *kinematic* pressure, *i.e.* $\text{m}^2 \text{s}^{-2}$ (see section 4.2.6 for more information);

internalField the internal field data which can be **uniform**, described by a single value; or **nonuniform**, where all the values of the field must be specified (see section 4.2.8 for more information);

boundaryField the boundary field data that includes boundary conditions and data for all the boundary patches (see section 4.2.8 for more information).

For this case **cavity**, the boundary consists of walls only, split into 2 patches named: (1) **fixedWalls** for the fixed sides and base of the cavity; (2) **movingWall** for the moving top of the cavity. As walls, both are given a **zeroGradient** boundary condition for **p**, meaning “the normal gradient of pressure is zero”. The **frontAndBack** patch represents the front and back planes of the 2D case and therefore must be set as **empty**.

In this case, as in most we encounter, the initial fields are set to be uniform. Here the pressure is kinematic, and as an incompressible case, its absolute value is not relevant, so is set to **uniform 0** for convenience.

The user can similarly examine the velocity field in the **0/U** file. The **dimensions** are those expected for velocity, the internal field is initialised as uniform zero, which in the case of velocity must be expressed by 3 vector components, *i.e.* **uniform (0 0 0)** (see section 4.2.5 for more information).

The boundary field for velocity requires the same boundary condition for the **frontAndBack** patch. The other patches are walls: a no-slip condition is assumed on the **fixedWalls**, hence a **noSlip** condition. The top surface moves at a speed of 1 m/s in the x -direction so requires a **fixedValue** condition with value of **uniform (1 0 0)**.

2.1.1.3 Physical properties

The physical properties for the case are stored in dictionaries whose names are given the suffix `Properties`, located in the `Dictionaries` directory tree. For an `icoFoam` case, the only property that must be specified is the kinematic viscosity which is stored from the `transportProperties` dictionary. The user can check that the kinematic viscosity is set correctly by opening the `transportProperties` dictionary to view/edit its entries. The keyword for kinematic viscosity is `nu`, the phonetic label for the Greek symbol ν by which it is represented in equations. Initially this case will be run with a Reynolds number of 10, where the Reynolds number is defined as:

$$Re = \frac{d|\mathbf{U}|}{\nu} \quad (2.1)$$

where d and $|\mathbf{U}|$ are the characteristic length and velocity respectively and ν is the kinematic viscosity. Here $d = 0.1$ m, $|\mathbf{U}| = 1$ m/s, so that for $Re = 10$, $\nu = 0.01$ m²s⁻¹. The correct file entry for kinematic viscosity is thus specified below:

```
17   nu           [0 2 -1 0 0 0] 0.01;
18
19 // ****
```

2.1.1.4 Control

Input data relating to the control of time and reading and writing of the solution data are read in from the `controlDict` dictionary. The user should view this file; as a case control file, it is located in the `system` directory.

The start/stop times and the time step for the run must be set. OpenFOAM offers great flexibility with time control which is described in full in section 4.3. In this tutorial we wish to start the run at time $t = 0$ which means that OpenFOAM needs to read field data from a directory named `0` — see section 4.1 for more information of the case file structure. Therefore we set the `startFrom` keyword to `startTime` and then specify the `startTime` keyword to be 0.

For the end time, we wish to reach the steady state solution where the flow is circulating around the cavity. As a general rule, the fluid should pass through the domain 10 times to reach steady state in laminar flow. In this case the flow does not pass through this domain as there is no inlet or outlet, so instead the end time can be set to the time taken for the lid to travel ten times across the cavity, *i.e.* 1 s; in fact, with hindsight, we discover that 0.5 s is sufficient so we shall adopt this value. To specify this end time, we must specify the `stopAt` keyword as `endTime` and then set the `endTime` keyword to 0.5.

Now we need to set the time step, represented by the keyword `deltaT`. To achieve temporal accuracy and numerical stability when running `icoFoam`, a Courant number of less than 1 is required. The Courant number is defined for one cell as:

$$Co = \frac{\delta t |\mathbf{U}|}{\delta x} \quad (2.2)$$

where δt is the time step, $|\mathbf{U}|$ is the magnitude of the velocity through that cell and δx is the cell size in the direction of the velocity. The flow velocity varies across the domain and we must ensure $Co < 1$ everywhere. We therefore choose δt based on the worst case: the maximum Co corresponding to the combined effect of a large flow velocity and small cell

size. Here, the cell size is fixed across the domain so the maximum Co will occur next to the lid where the velocity approaches 1 m s^{-1} . The cell size is:

$$\delta x = \frac{d}{n} = \frac{0.1}{20} = 0.005 \text{ m} \quad (2.3)$$

Therefore to achieve a Courant number less than or equal to 1 throughout the domain the time step `deltaT` must be set to less than or equal to:

$$\delta t = \frac{Co \delta x}{|\mathbf{U}|} = \frac{1 \times 0.005}{1} = 0.005 \text{ s} \quad (2.4)$$

As the simulation progresses we wish to write results at certain intervals of time that we can later view with a post-processing package. The `writeControl` keyword presents several options for setting the time at which the results are written; here we select the `timeStep` option which specifies that results are written every n th time step where the value n is specified under the `writeInterval` keyword. Let us decide that we wish to write our results at times $0.1, 0.2, \dots, 0.5$ s. With a time step of 0.005 s, we therefore need to output results at every 20th time step and so we set `writeInterval` to 20.

OpenFOAM creates a new directory *named after the current time*, e.g. 0.1 s, on each occasion that it writes a set of data, as discussed in full in section 4.1. In the `icoFoam` solver, it writes out the results for each field, \mathbf{U} and p , into the time directories. For this case, the entries in the `controlDict` are shown below:

```

17 application      icoFoam;
18 startFrom       startTime;
19 startTime        0;
20 stopAt          endTime;
21 endTime          0.5;
22 deltaT          0.005;
23 writeControl    timeStep;
24 writeInterval   20;
25 purgeWrite      0;
26 writeFormat     ascii;
27 writePrecision  6;
28 writeCompression off;
29 timeFormat      general;
30 timePrecision   6;
31 runTimeModifiable true;
32 // ****
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47

```

2.1.1.5 Discretisation and linear-solver settings

The user specifies the choice of finite volume discretisation schemes in the `fvSchemes` dictionary in the `system` directory. The specification of the linear equation solvers and tolerances and other algorithm controls is made in the `fvSolution` dictionary, similarly in the `system` directory. The user is free to view these dictionaries but we do not need to discuss all their entries at this stage except for `pRefCell` and `pRefValue` in the `PISO` sub-dictionary of the

`fvSolution` dictionary. In a closed incompressible system such as the cavity, pressure is relative: it is the pressure range that matters not the absolute values. In cases such as this, the solver sets a reference level by `pRefValue` in cell `pRefCell`. In this example both are set to 0. Changing either of these values will change the absolute pressure field, but not, of course, the relative pressures or velocity field.

2.1.2 Viewing the mesh

Before the case is run it is a good idea to view the mesh to check for any errors. The mesh is viewed in `ParaView`, the post-processing tool supplied with OpenFOAM. The `ParaView` post-processing is conveniently launched on OpenFOAM case data by executing the `paraFoam` script from within the case directory.

Any `UNIX/Linux` executable can be run in two ways: as a foreground process, *i.e.* one in which the shell waits until the command has finished before giving a command prompt; as a background process, which allows the shell to accept additional commands while it is still running. Since it is convenient to keep `ParaView` open while running other commands from the terminal, we will launch it in the background using the `&` operator by typing

```
paraFoam &
```

Alternatively, it can be launched from another directory location with an optional `-case` argument giving the case directory, *e.g.*

```
paraFoam -case $FOAM_RUN/cavity &
```

This launches the `ParaView` window as shown in Figure 6.1. In the Pipeline Browser, the user can see that `ParaView` has opened `cavity.OpenFOAM`, the module for the `cavity` case. **Before clicking the Apply button**, the user needs to select some geometry from the Mesh Parts panel. Because the case is small, it is easiest to select all the data by checking the box adjacent to the Mesh Parts panel title, which automatically checks all individual components within the respective panel. The user should then click the **Apply** button to load the geometry into `ParaView`.

The user should then scroll down to the Display panel that controls the visual representation of the selected module. Within the Display panel the user should do the following as shown in Figure 2.3:

1. in the **Coloring** section, select **Solid Color**;
2. click **Edit** (in **Coloring**) and select an appropriate colour *e.g.* black (for a white background);
3. select **Wireframe** from the **Representation** menu. The background colour can be set in the **View Render** panel below the **Display** panel in the **Properties** window.

Especially the first time the user starts `ParaView`, **it is recommended** that they manipulate the view as described in section 6.1.5. In particular, since this is a 2D case, it is recommended that **Use Parallel Projection** is selected near the bottom of the **View Render** panel, available only with the **Advanced Properties** gearwheel button pressed at the top of the **Properties** window, next to the search box. **View Settings** window selected from the **Edit** menu. The **Orientation Axes** can be toggled on and off in the **Annotation** window or moved by drag and drop with the mouse.

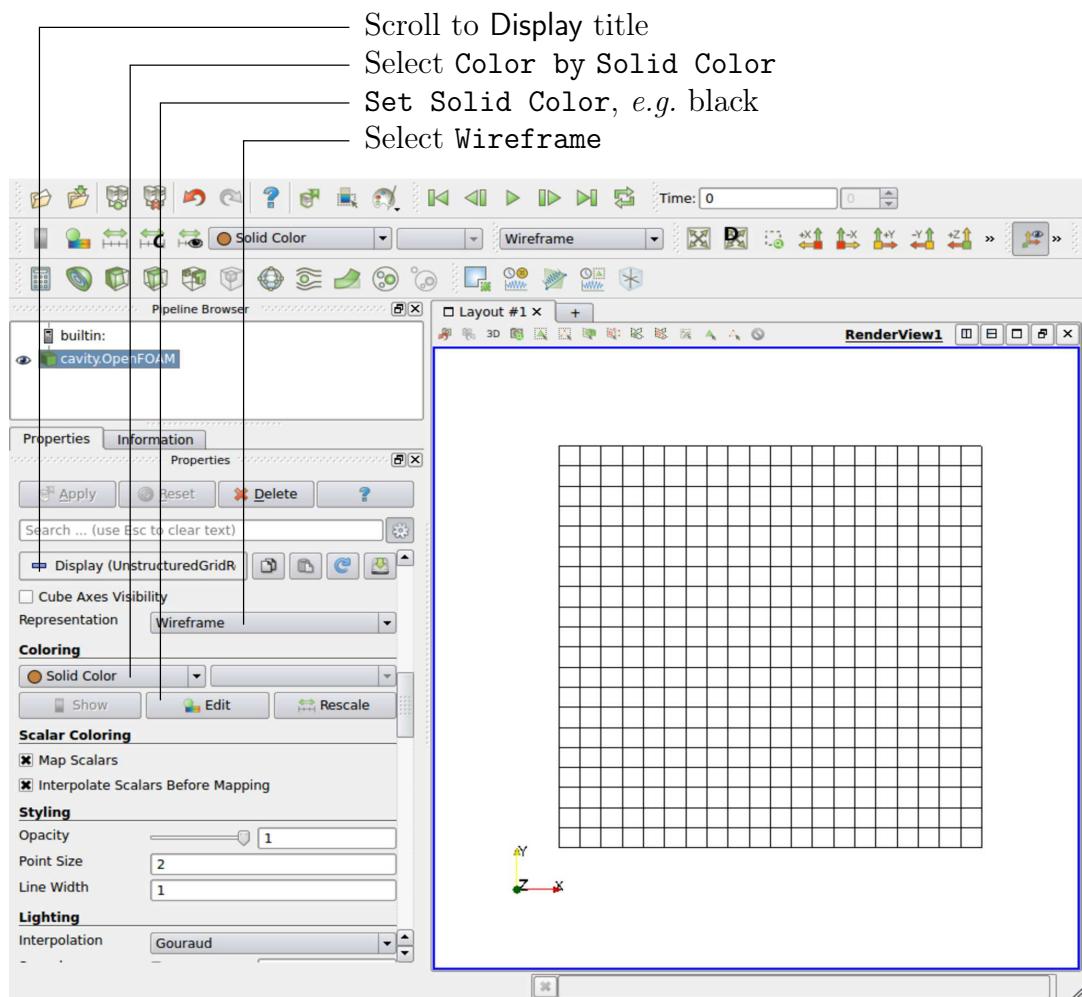


Figure 2.3: Viewing the mesh in paraFoam.

2.1.3 Running an application

Like any UNIX/Linux executable, OpenFOAM applications can be run either in the foreground or background. On this occasion, we will run `icoFoam` in the foreground. The `icoFoam` solver is executed either by entering the case directory and typing

```
icoFoam
```

at the command prompt, or with the optional `-case` argument giving the case directory, *e.g.*

```
icoFoam -case $FOAM_RUN/cavity
```

The progress of the job is written to the terminal window. It tells the user the current time, maximum Courant number, initial and final residuals for all fields.

2.1.4 Post-processing

As soon as results are written to time directories, they can be viewed using `paraFoam`. Return to the `paraFoam` window and select the `Properties` panel for the `cavity.OpenFOAM` case module. If the correct window panels for the case module do not seem to be present at any time, please ensure that: `cavity.OpenFOAM` is highlighted in blue; `eye` button alongside it is switched on to show the graphics are enabled;

To prepare `paraFoam` to display the data of interest, we must first load the data at the required run time of 0.5 s. If the case was run while `ParaView` was open, the output data in time directories will not be automatically loaded within `ParaView`. To load the data the user should click `Refresh Times` at the **top** `Properties` window (scroll up the panel if necessary). The time data will be loaded into `ParaView`.

In order to view the solution at $t = 0.5$ s, the user can use the `VCR Controls` or `Current Time Controls` to change the current time to 0.5. These are located in the toolbars at the top of the `ParaView` window, as shown in Figure 6.4.

2.1.4.1 Colouring surfaces

To view pressure, the user should go to the `Display` panel since it controls the visual representation of the selected module. To make a simple plot of pressure, the user should select the following, as described in detail in Figure 2.4:

1. select `Surface` from the `Representation` menu;
2. select  in `Coloring`
3. click the `Rescale` button to set the colour scale to the data range, if necessary.

The pressure field should appear as shown in Figure 2.5, with a region of low pressure at the top left of the cavity and one of high pressure at the top right of the cavity.

With the point icon () the pressure field is interpolated across each cell to give a continuous appearance. Instead if the user selects the cell icon, , from the `Coloring` menu, a single value for pressure will be attributed to each cell so that each cell will be denoted by a single colour with no grading.

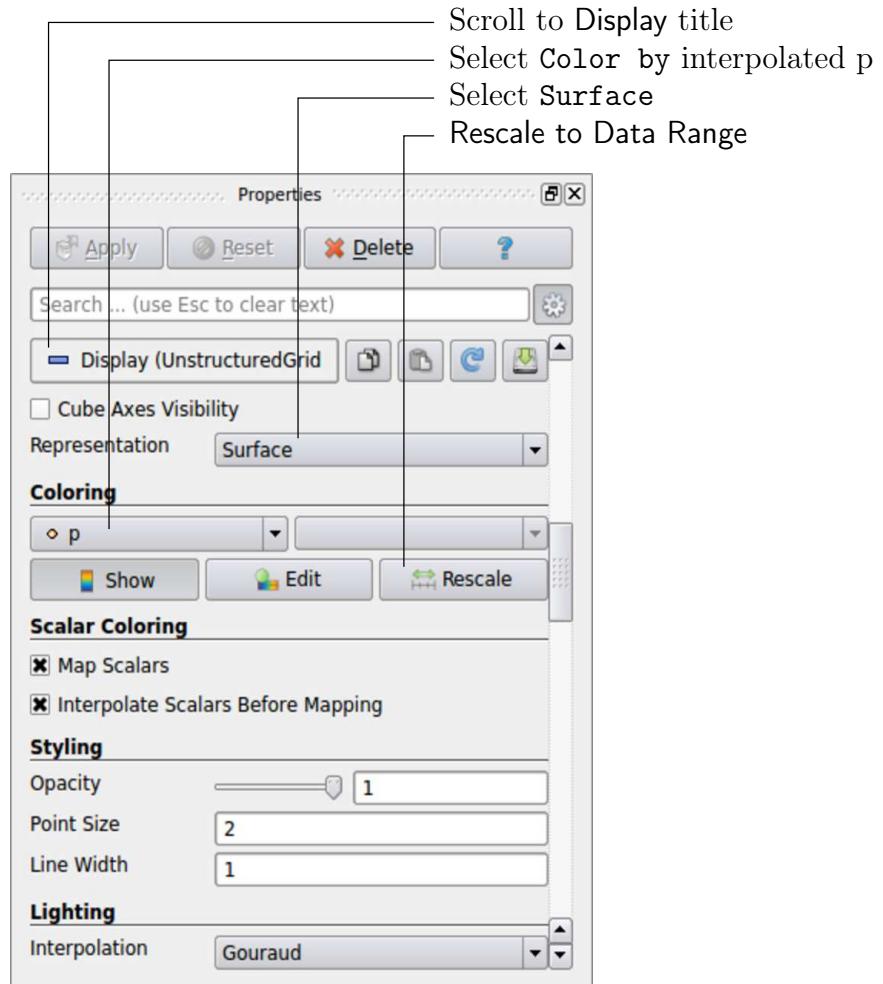


Figure 2.4: Displaying pressure contours for the cavity case.

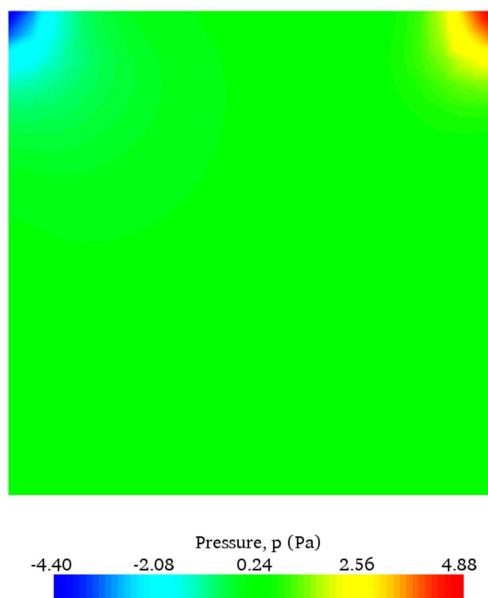


Figure 2.5: Pressures in the cavity case.

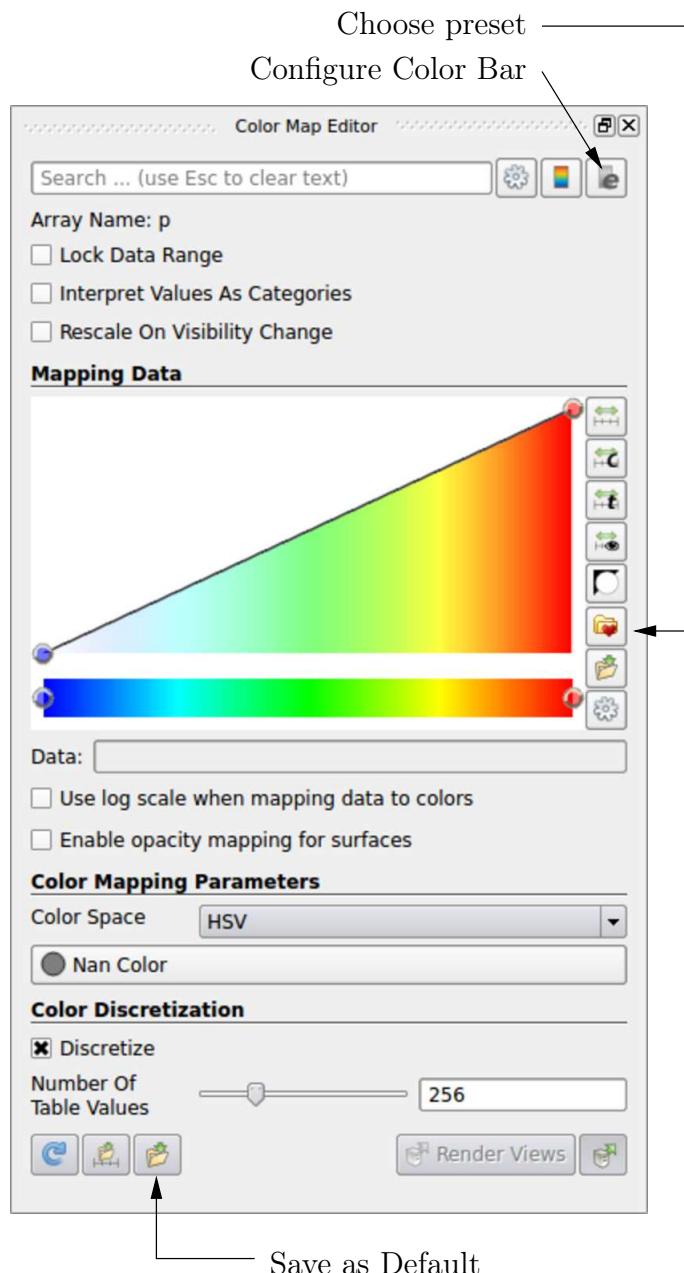


Figure 2.6: Color Map Editor.

A colour legend can be added by either by clicking the Toggle Color Legend Visibility button in the Active Variable Controls toolbar or the Show button in the Coloring section of the Display panel. The legend can be located in the image window by drag and drop with the mouse. The Edit button, either in the Active Variable Controls toolbar or in the Coloring panel of the Display panel, opens the Color Map Editor window, as shown in Figure 2.6, where the user can set a range of attributes of the colour scale and the color bar. In particular, ParaView defaults to using a colour scale of blue to white to red rather than the more common blue to green to red (rainbow). Therefore *the first time* that the user executes ParaView, they may wish to change the colour scale. This can be done by selecting the Choose Preset button (with the heart icon) in the Color Scale Editor and selecting Blue to Red Rainbow. After clicking the OK confirmation button, the user can click the Save as Default button at the bottom of the panel (disk drive symbol) so that ParaView will always

adopt this type of colour bar.

The user can also edit the color legend properties, such as text size, font selection and numbering format for the scale, by clicking the **Edit Color Legend Properties** to the far right of the search bar, as shown in Figure 2.6.

2.1.4.2 Cutting plane (slice)

If the user rotates the image, by holding down the left mouse button in the image window and moving the cursor, they can see that they have now coloured the complete geometry surface by the pressure. In order to produce a genuine 2-dimensional contour plot the user should first create a cutting plane, or ‘slice’. With the `cavity.OpenFOAM` module highlighted in the **Pipeline Browser**, the user should select the **Slice** filter from the **Filters** menu in the top menu of **ParaView** (accessible at the top of the screen on some systems). The **Slice** filter can be initially found in the **Common** sub-menu, but once selected, it **moves** to the **Recent** sub-menu, **disappearing** from the the **Common** sub-menu. The cutting plane should be centred at $(0.05, 0.05, 0.005)$ and its normal should be set to $(0, 0, 1)$ (click the **Z Normal** button).

2.1.4.3 Contours

Having generated the cutting plane, contours can be created using by applying the **Contour** filter. With the **Slice** module highlighted in the **Pipeline Browser**, the user should select the **Contour** filter. In the **Properties** panel, the user should select pressure from the **Contour By** menu. Under **Isosurfaces**, the user could delete the default value with the minus button, then add a range of 10 values. The contours can be displayed with a **Wireframe** representation if the **Coloring** is solid or by a field, *e.g.* pressure.

2.1.4.4 Vector plots

Before we start to plot the vectors of the flow velocity, it may be useful to remove other modules that have been created, *e.g.* using the **Slice** and **Contour** filters described above. These can: either be deleted entirely, by highlighting the relevant module in the **Pipeline Browser** and clicking **Delete** in their respective **Properties** panel; or, be disabled by toggling the **eye** button for the relevant module in the **Pipeline Browser**.

We now wish to generate a vector glyph for velocity at the centre of each cell. We first need to filter the data to cell centres as described in section 6.1.7.1. With the `cavity.OpenFOAM` module highlighted in the **Pipeline Browser**, the user should select **Cell Centers** from the **Filter->Alphabetical** menu and then click **Apply**.

With these **Centers** highlighted in the **Pipeline Browser**, the user should then select **Glyph** from the **Filter->Common** menu. The **Properties** window panel should appear as shown in Figure 2.7. Note that newly selected filters are moved to the **Filter->Recent** menu and are unavailable in the menus from where they were originally selected. In the resulting **Properties** panel, the velocity field, **U**, must be selected from the **vectors** menu. The user should set the **Scale Mode** for the glyphs to be **off**, with **Set Scale Factor** set to **0.005**. On clicking **Apply**, the glyphs appear but, probably as a single colour, *e.g.* white. The user should colour the glyphs by velocity magnitude which, as usual, is controlled by setting **Color by U** in the **Display** panel. The user can also select **Show Color Legend** in **Edit Color Map**. The output is shown in Figure 2.8, in which uppercase Times Roman fonts are selected

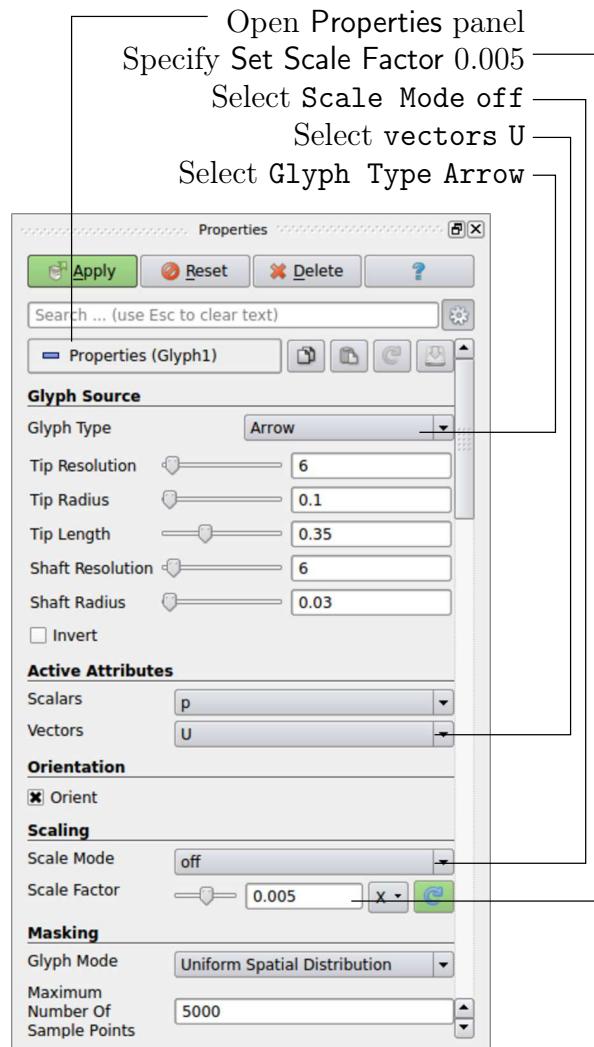


Figure 2.7: Properties panel for the Glyph filter.

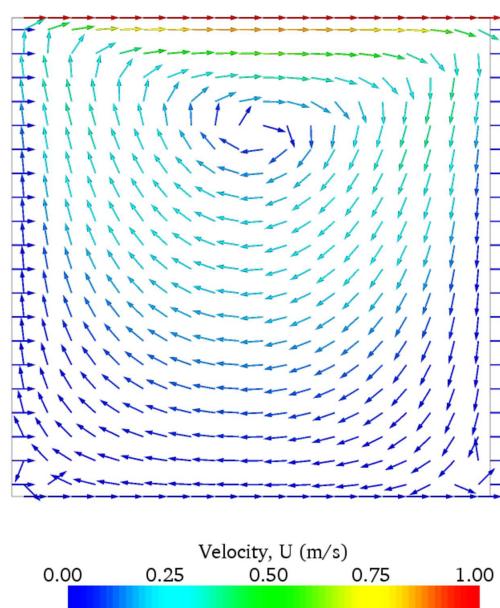


Figure 2.8: Velocities in the cavity case.

for the **Color Legend** headings and the labels are specified to 2 fixed significant figures by deselecting **Automatic Label Format** and entering `%-#6.2f` in the **Label Format** text box. The background colour is set to white in the **General** panel of **View Settings** as described in section 6.1.5.1.

Note that at the left and right walls, glyphs appear to indicate flow through the walls. However, it is clear that, while the flow direction is normal to the wall, its magnitude is 0. This slightly confusing situation is caused by ParaView choosing to orientate the glyphs in the *x*-direction when the **glyph scaling off** and the velocity magnitude is 0.

2.1.4.5 Streamline plots

Again, before the user continues to post-process in ParaView, they should disable modules such as those for the vector plot described above. We now wish to plot streamlines of velocity as described in section 6.1.8. With the `cavity.OpenFOAM` module highlighted in the Pipeline Browser, the user should then select **Stream Tracer** from the **Filter** menu and then click **Apply**. The Properties window panel should appear as shown in Figure 2.9. The **Seed points** should be specified along a **High Resolution Line Source** running vertically through the centre of the geometry, *i.e.* from $(0.05, 0, 0.005)$ to $(0.05, 0.1, 0.005)$. For the image in this guide we used: a point **Resolution** of 21; **Maximum Step Length** of 0.5; **Initial Step Length** of 0.2; and, **Integration Direction BOTH**. The **Runge-Kutta 4/5 IntegratorType** was used with default parameters.

On clicking **Apply** the tracer is generated. The user should then select **Tube** from the **Filter** menu to produce high quality streamline images. For the image in this report, we used: **Num. sides** 6; **Radius** 0.0003; and, **Radius factor** 10. The streamtubes are coloured by velocity magnitude. On clicking **Apply** the image in Figure 2.10 should be produced.

2.1.5 Increasing the mesh resolution

The mesh resolution will now be increased by a factor of two in each direction. The results from the coarser mesh will be mapped onto the finer mesh to use as initial conditions for the problem. The solution from the finer mesh will then be compared with those from the coarser mesh.

2.1.5.1 Creating a new case using an existing case

We now wish to create a new case named `cavityFine` that is created from `cavity`. The user should therefore clone the `cavity` case and edit the necessary files. First the user should go to the `run` directory, by typing

```
cd $FOAM_RUN
```

Note that there is also a convenient alias, named `run`, that reproduces the command above to change directory to `$FOAM_RUN`, simply by typing `run`.

The `cavityFine` case can be created by making a new case directory and copying the relevant directories from the `cavity` case.

```
mkdir cavityFine
cp -r cavity/constant cavityFine
cp -r cavity/system cavityFine
```

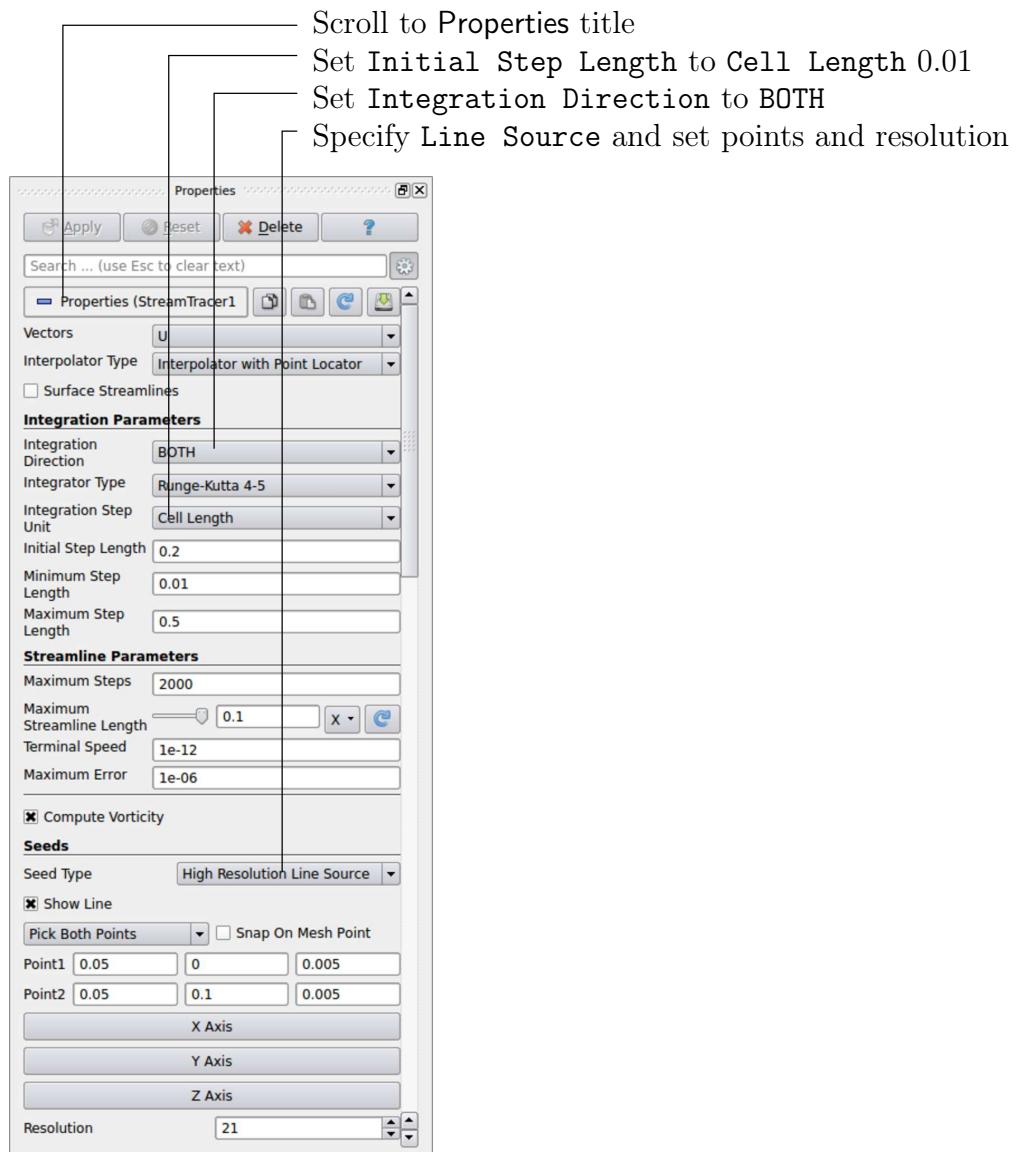


Figure 2.9: Properties panel for the Stream Tracer filter.

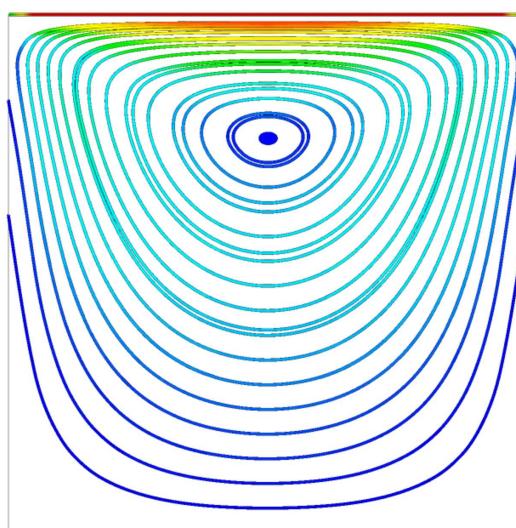


Figure 2.10: Streamlines in the cavity case.

The user can then prepare to run the new case by changing into the case directory.

```
cd cavityFine
```

2.1.5.2 Creating the finer mesh

We now wish to increase the number of cells in the mesh by using `blockMesh`. The user should open the `blockMeshDict` file in the `system` directory in an editor and edit the block specification. The blocks are specified in a list under the `blocks` keyword. The syntax of the block definitions is described fully in section 5.3.1.3; at this stage it is sufficient to know that following `hex` is first the list of vertices in the block, then a list (or vector) of numbers of cells in each direction. This was originally set to `(20 20 1)` for the `cavity` case. The user should now change this to `(40 40 1)` and save the file. The new refined mesh should then be created by running `blockMesh` as before.

2.1.5.3 Mapping the coarse mesh results onto the fine mesh

The `mapFields` utility maps one or more fields relating to a given geometry onto the corresponding fields for another geometry. In our example, the fields are deemed ‘consistent’ because the geometry and the boundary types, or conditions, of both source and target fields are identical. We use the `-consistent` command line option when executing `mapFields` in this example.

The field data that `mapFields` maps is read from the time directory specified by `startFrom` and `startTime` in the `controlDict` of the target case, *i.e.* those **into which** the results are being mapped. In this example, we wish to map the final results of the coarser mesh from case `cavity` onto the finer mesh of case `cavityFine`. Therefore, since these results are stored in the `0.5` directory of `cavity`, the `startTime` should be set to `0.5` s in the `controlDict` dictionary and `startFrom` should be set to `startTime`.

The case is ready to run `mapFields`. Typing `mapFields -help` quickly shows that `mapFields` requires the source case directory as an argument. We are using the `-consistent` option, so the utility is executed from within the `cavityFine` directory by

```
mapFields ../cavity -consistent
```

The utility should run with output to the terminal including:

```
Source: ".." "cavity"
Target: "." "cavityFine"

Create databases as time

Source time: 0.5
Target time: 0.5

Create meshes

Source mesh size: 400    Target mesh size: 1600

Consistently creating and mapping fields for time 0.5

    interpolating p
    interpolating U

End
```

2.1.5.4 Control adjustments

To maintain a Courant number of less than 1, as discussed in section 2.1.1.4, the time step must now be halved since the size of all cells has halved. Therefore `deltaT` should be set to to 0.0025 s in the `controlDict` dictionary. Field data is currently written out at an interval of a fixed number of time steps. Here we demonstrate how to specify data output at fixed intervals of time. Under the `writeControl` keyword in `controlDict`, instead of requesting output by a fixed number of time steps with the `timeStep` entry, a fixed amount of run time can be specified between the writing of results using the `runTime` entry. In this case the user should specify output every 0.1 and therefore should set `writeInterval` to 0.1 and `writeControl` to `runTime`. Finally, since the case is starting with a the solution obtained on the coarse mesh we only need to run it for a short period to achieve reasonable convergence to steady-state. Therefore the `endTime` should be set to 0.7 s. Make sure these settings are correct and then save the file.

2.1.5.5 Running the code as a background process

The user should experience running `icoFoam` as a background process, redirecting the terminal output to a `log` file that can be viewed later. From the `cavityFine` directory, the user should execute:

```
icoFoam > log &
cat log
```

2.1.5.6 Vector plot with the refined mesh

The user can open multiple cases simultaneously in ParaView; essentially because each new case is simply another module that appears in the Pipeline Browser. There is an inconvenience when opening a new OpenFOAM case in ParaView because it expects that case data is stored in a single file which has a file extension that enables it to establish the format. However, OpenFOAM stores case data in multiple files without an extension in the name, within a specific directory structure. The ParaView reader module works on the basis that, when opening case data in OpenFOAM format, it is passed a dummy (empty) file with the `.OpenFOAM` extension that resides in the case directory. The `paraFoam` script automatically creates this file — hence, the `cavity` case module is called `cavity.OpenFOAM`.

If the user wishes to open a second case directly from within ParaView, they need to create such a dummy file. They can do this ‘by hand’ or, more simply, use the `paraFoam` script with the option `-touch`. For the `cavityFine` example, that involves executing from the case directory:

```
paraFoam -touch
```

Now the `cavityFine` case can be loaded into ParaView by selecting `Open` from the `File` menu, and having navigated to the `cavityFine` directory, opening `cavityFine.OpenFOAM`. The user can now make a vector plot of the results from the refined mesh in ParaView. The plot can be compared with the `cavity` case by enabling glyph images for both case simultaneously.

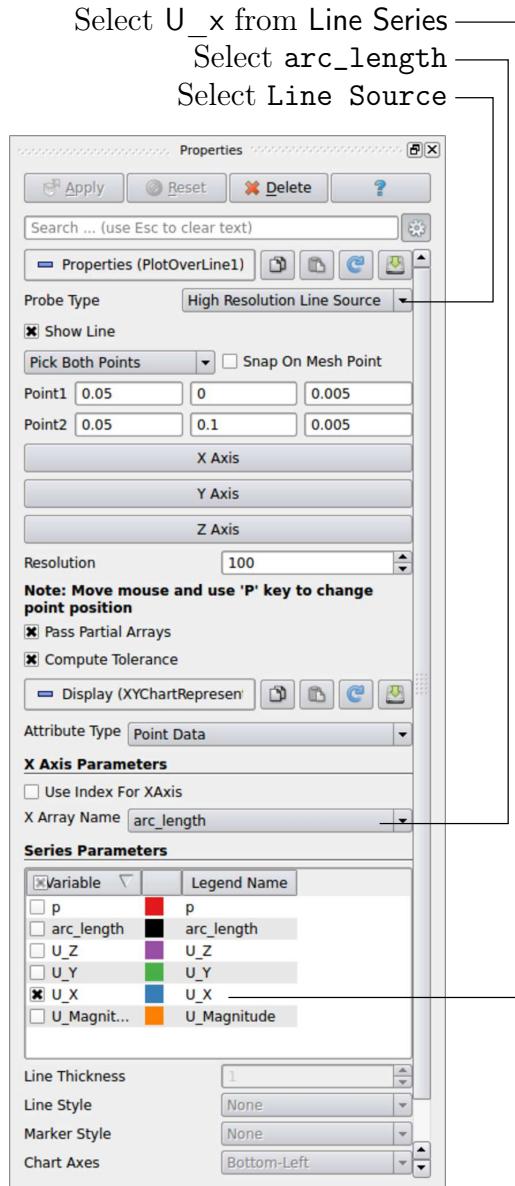


Figure 2.11: Selecting fields for graph plotting.

2.1.5.7 Plotting graphs

The user may wish to visualise the results by extracting some scalar measure of velocity and plotting 2-dimensional graphs along lines through the domain. OpenFOAM is well equipped for this kind of data manipulation. There are numerous utilities that do specialised data manipulations, and the `postProcess` utility that includes a broad range of generic post-processing functionality. The functions within `postProcess` can be listed by typing:

```
postProcess -list
```

The `components` and `mag` functions provide useful scalar measures of velocity. When the `components` function is executed on a case, *e.g.* `cavity`, it reads in the velocity vector field from each time directory and, in the corresponding time directories, writes scalar fields `Ux`, `Uy` and `Uz` representing the *x*, *y* and *z* components of velocity.

The user can run `postProcess` with the `components` function on both `cavity` and `cavityFine` cases. For example, for the `cavity` case the user should go into the `cavity` directory and execute `postProcess` as follows:

```
cd $FOAM_RUN/cavity
postProcess -func "components(U)"
```

The individual components can be plotted as a graph in ParaView. It is quick, convenient and has reasonably good control over labelling and formatting, so the printed output is a fairly good standard. However, to produce graphs for publication, users may prefer to write raw data and plot it with a dedicated graphing tool, such as `gnuplot` or `Grace/xmgr`. To do this, we recommend using the sampling tools, described in section 6.3.2 and section 2.2.3.

Before commencing plotting, the user needs to load the newly generated `Ux`, `Uy` and `Uz` fields into ParaView. To do this, the user should click the **Refresh Times** at the top of the **Properties** panel for the `cavity.OpenFOAM` module which will cause the new fields to be loaded into ParaView and appear in the **Volume Fields** window. Ensure the new fields are selected and the changes are applied, *i.e.* click **Apply** again if necessary. *Also,* data is interpolated incorrectly at boundaries if the boundary regions are selected in the **Mesh Parts** panel. Therefore the user should *deselect the patches* in the **Mesh Parts** panel, *i.e.* `movingWall`, `fixedWall` and `frontAndBack`, and apply the changes.

Now, in order to display a graph in ParaView the user should select the module of interest, *e.g.* `cavity.OpenFOAM` and apply the **Plot Over Line** filter from the **Filter->Data Analysis** menu. This opens up a new **XY Plot** window below or beside the existing 3D View window. A **PlotOverLine** module is created in which the user can specify the end points of the line in the **Properties** panel. In this example, the user should position the line vertically up the centre of the domain, *i.e.* from $(0.05, 0, 0.005)$ to $(0.05, 0.1, 0.005)$, in the **Point1** and **Point2** text boxes. The **Resolution** can be set to 100.

On clicking **Apply**, a graph is generated in the **XY Plot** window. In the **Display** panel, the user should set **Attribute Mode** to **Point Data**. The **Use Data Array** option can be selected for the **X Axis Data**, taking the `arc_length` option so that the x-axis of the graph represents distance from the base of the cavity.

The user can choose the fields to be displayed in the **Line Series** panel of the **Display** window. From the list of scalar fields to be displayed, it can be seen that the magnitude and components of vector fields are available by default, *e.g.* displayed as `U_X`, so that it was not necessary to create `Ux` using `foamCalc`. Nevertheless, the user should deselect all series except `Ux` (*or* `U_x`). A square colour box in the adjacent column to the selected series indicates the line colour. The user can edit this most easily by a double click of the mouse over that selection.

In order to format the graph, the user should modify the settings below the **Line Series** panel, namely **Line Color**, **Line Thickness**, **Line Style**, **Marker Style** and **Chart Axes**.

Also the user can click one of the buttons above the top left corner of the **XY Plot**. The third button, for example, allows the user to control **View Settings** in which the user can set title and legend for each axis, for example. Also, the user can set font, colour and alignment of the axes titles, and has several options for axis range and labels in linear or logarithmic scales.

Figure 2.12 is a graph produced using ParaView. The user can produce a graph however he/she wishes. For information, the graph in Figure 2.12 was produced with the options for axes of: **Standard** type of **Notation**; **Specify Axis Range** selected; titles in **Sans Serif 12**

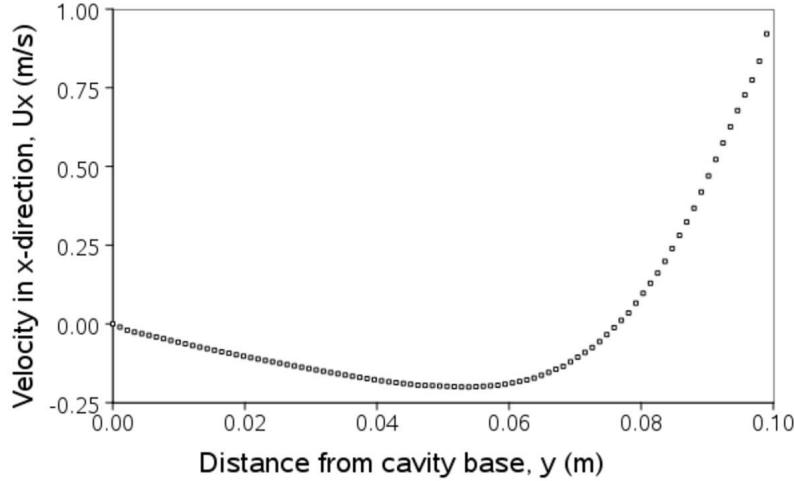


Figure 2.12: Plotting graphs in paraFoam.

font. The graph is displayed as a set of points rather than a line by activating the `Enable Line Series` button in the `Display` window. Note: if this button appears to be inactive by being “greyed out”, it can be made active by selecting and deselecting the sets of variables in the `Line Series` panel. Once the `Enable Line Series` button is selected, the `Line Style` and `Marker Style` can be adjusted to the user’s preference.

2.1.6 Introducing mesh grading

The error in any solution will be more pronounced in regions where the form of the true solution differ widely from the form assumed in the chosen numerical schemes. For example a numerical scheme based on linear variations of variables over cells can only generate an exact solution if the true solution is itself linear in form. The error is largest in regions where the true solution deviates greatest from linear form, *i.e.* where the change in gradient is largest. Error decreases with cell size.

It is useful to have an intuitive appreciation of the form of the solution before setting up any problem. It is then possible to anticipate where the errors will be largest and to grade the mesh so that the smallest cells are in these regions. In the `cavity` case the large variations in velocity can be expected near a wall and so in this part of the tutorial the mesh will be graded to be smaller in this region. By using the same number of cells, greater accuracy can be achieved without a significant increase in computational cost.

A mesh of 20×20 cells with grading towards the walls will be created for the lid-driven cavity problem and the results from the finer mesh of section 2.1.5.2 will then be mapped onto the graded mesh to use as an initial condition. The results from the graded mesh will be compared with those from the previous meshes. Since the changes to the `blockMeshDict` dictionary are fairly substantial, the case used for this part of the tutorial, `cavityGrade`, is supplied in the `$FOAM_TUTORIALS/incompressible/icoFoam/cavity` directory. The user should copy the `cavityGrade` case into the `run` directory, then follow the steps below.

2.1.6.1 Creating the graded mesh

The mesh now needs 4 blocks as different mesh grading is needed on the left and right and top and bottom of the domain. The block structure for this mesh is shown in Figure 2.13.

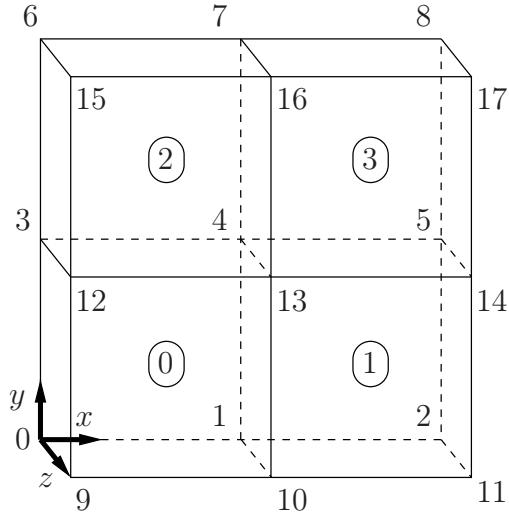


Figure 2.13: Block structure of the graded mesh for the cavity (block numbers encircled).

The user can view the *blockMeshDict* file in the *system* subdirectory of *cavityGrade*; for completeness the key elements of the *blockMeshDict* file are also reproduced below. Each block now has 10 cells in the *x* and *y* directions and the ratio between largest and smallest cells is 2.

```

17 convertToMeters 0.1;
18
19 vertices
20 (
21     (0 0 0)
22     (0.5 0 0)
23     (1 0 0)
24     (0 0.5 0)
25     (0.5 0.5 0)
26     (1 0.5 0)
27     (0 1 0)
28     (0.5 1 0)
29     (1 1 0)
30     (0 0 0.1)
31     (0.5 0 0.1)
32     (1 0 0.1)
33     (0 0.5 0.1)
34     (0.5 0.5 0.1)
35     (1 0.5 0.1)
36     (0 1 0.1)
37     (0.5 1 0.1)
38     (1 1 0.1)
39 );
40
41 blocks
42 (
43     hex (0 1 4 3 9 10 13 12) (10 10 1) simpleGrading (2 2 1)
44     hex (1 2 5 4 10 11 14 13) (10 10 1) simpleGrading (0.5 2 1)
45     hex (3 4 7 6 12 13 16 15) (10 10 1) simpleGrading (2 0.5 1)
46     hex (4 5 8 7 13 14 17 16) (10 10 1) simpleGrading (0.5 0.5 1)
47 );
48
49 edges
50 (
51 );
52
53 boundary
54 (
55     movingWall
56     {
57         type wall;
58         faces
59         (
60             (6 15 16 7)
61             (7 16 17 8)

```

```

62     );
63 }
64 fixedWalls
65 {
66     type wall;
67     faces
68     (
69         (3 12 15 6)
70         (0 9 12 3)
71         (0 1 10 9)
72         (1 2 11 10)
73         (2 5 14 11)
74         (5 8 17 14)
75     );
76 }
77 frontAndBack
78 {
79     type empty;
80     faces
81     (
82         (0 3 4 1)
83         (1 4 5 2)
84         (3 6 7 4)
85         (4 7 8 5)
86         (9 10 13 12)
87         (10 11 14 13)
88         (12 13 16 15)
89         (13 14 17 16)
90     );
91 }
92 );
93
94 mergePatchPairs
95 (
96 );
97 // ****

```

Once familiar with the *blockMeshDict* file for this case, the user can execute **blockMesh** from the command line. The graded mesh can be viewed as before using **paraFoam** as described in section 2.1.2.

2.1.6.2 Changing time and time step

The highest velocities and smallest cells are next to the lid, therefore the highest Courant number will be generated next to the lid, for reasons given in section 2.1.1.4. It is therefore useful to estimate the size of the cells next to the lid to calculate an appropriate time step for this case.

When a nonuniform mesh grading is used, **blockMesh** calculates the cell sizes using a geometric progression. Along a length l , if n cells are requested with a ratio of R between the last and first cells, the size of the smallest cell, δx_s , is given by:

$$\delta x_s = l \frac{r - 1}{\alpha r - 1} \quad (2.5)$$

where r is the ratio between one cell size and the next which is given by:

$$r = R^{\frac{1}{n-1}} \quad (2.6)$$

and

$$\alpha = \begin{cases} R & \text{for } R > 1, \\ 1 - r^{-n} + r^{-1} & \text{for } R < 1. \end{cases} \quad (2.7)$$

For the **cavityGrade** case the number of cells in each direction in a block is 10, the ratio between largest and smallest cells is 2 and the block height and width is 0.05 m. Therefore

the smallest cell length is 3.45 mm. From Equation 2.2, the time step should be less than 3.45 ms to maintain a Courant of less than 1. To ensure that results are written out at convenient time intervals, the time step `deltaT` should be reduced to 2.5 ms and the `writeInterval` set to 40 so that results are written out every 0.1 s. These settings can be viewed in the `cavityGrade/system/controlDict` file.

The `startTime` needs to be set to that of the final conditions of the case `cavityFine`, *i.e.* 0.7. Since `cavity` and `cavityFine` converged well within the prescribed run time, we can set the run time for case `cavityGrade` to 0.1 s, *i.e.* the `endTime` should be 0.8.

2.1.6.3 Mapping fields

As in section 2.1.5.3, use `mapFields` to map the final results from case `cavityFine` onto the mesh for case `cavityGrade`. Enter the `cavityGrade` directory and execute `mapFields` by:

```
cd $FOAM_RUN/cavityGrade
mapFields ../cavityFine -consistent
```

Now run `icoFoam` from the case directory and monitor the run time information. View the converged results for this case and compare with other results using post-processing tools described previously in section 2.1.5.6 and section 2.1.5.7.

2.1.7 Increasing the Reynolds number

The cases solved so far have had a Reynolds number of 10. This is very low and leads to a stable solution quickly with only small secondary vortices at the bottom corners of the cavity. We will now increase the Reynolds number to 100, at which point the solution takes a noticeably longer time to converge. The coarsest mesh in case `cavity` will be used initially. The user should clone the `cavity` case and name it `cavityHighRe`. Rather than copying individual directories (`system`, `constant`, etc.) as described previously, the `foamCloneCase` can be used, which copies the relevant directories in one step. By default the `0` time directory is copied, but here the user can use the `-latestTime` option to copy the latest time directory, `0.5`, which can be used as the initial field data for our simulation. The example also uses the `run` alias as a quick way to change to the `run` directory.

```
run
foamCloneCase -latestTime cavity cavityHighRe
cd cavityHighRe
```

2.1.7.1 Pre-processing

Go into the `cavityHighRe` case and edit the `transportProperties` dictionary in the `constant` directory. Since the Reynolds number is required to be increased by a factor of 10, decrease the kinematic viscosity by a factor of 10, *i.e.* to $1 \times 10^{-3} \text{ m}^2 \text{s}^{-1}$. We now run this case by restarting from the solution at the end of the `cavity` case run. To do this we can use the option of setting the `startFrom` keyword to `latestTime` so that `icoFoam` takes as its initial data the values stored in the directory corresponding to the most recent time, *i.e.* `0.5`. The `endTime` should be set to 2 s.

2.1.7.2 Running the code

Run `icoFoam` for this case from the case directory and view the run time information. When running a job in the background, the following UNIX commands can be useful:

`nohup` enables a command to keep running after the user who issues the command has logged out;

`nice` changes the priority of the job in the kernel's scheduler; a niceness of -20 is the highest priority and 19 is the lowest priority.

This is useful, for example, if a user wishes to set a case running on a remote machine and does not wish to monitor it heavily, in which case they may wish to give it low priority on the machine. In that case the `nohup` command allows the user to log out of a remote machine he/she is running on and the job continues running, while `nice` can set the priority to 19. For our case of interest, we can execute the command in this manner as follows:

```
nohup nice -n 19 icoFoam > log &
cat log
```

In previous runs you may have noticed that `icoFoam` stops solving for velocity U quite quickly but continues solving for pressure p for a lot longer or until the end of the run. In practice, once `icoFoam` stops solving for U and the initial residual of p is less than the tolerance set in the `fvSolution` dictionary (typically 10^{-6}), the run has effectively converged and can be stopped once the field data has been written out to a time directory. For example, at convergence a sample of the `log` file from the run on the `cavityHighRe` case appears as follows in which the velocity has already converged after 1.395 s and initial pressure residuals are small; `No Iterations 0` indicates that the solution of U has stopped:

```
1 Time = 1.43
2
3 Courant Number mean: 0.221921 max: 0.839902
4 smoothSolver: Solving for Ux, Initial residual = 8.73381e-06, Final residual = 8.73381e-06, No Iterations 0
5 smoothSolver: Solving for Uy, Initial residual = 9.89679e-06, Final residual = 9.89679e-06, No Iterations 0
6 DICPCG: Solving for p, Initial residual = 3.67506e-06, Final residual = 8.62986e-07, No Iterations 4
7 time step continuity errors : sum local = 6.57947e-09, global = -6.6679e-19, cumulative = -6.2539e-18
8 DICPCG: Solving for p, Initial residual = 2.60898e-06, Final residual = 7.92532e-07, No Iterations 3
9 time step continuity errors : sum local = 6.26199e-09, global = -1.02984e-18, cumulative = -7.28374e-18
10 ExecutionTime = 0.37 s ClockTime = 0 s
11
12 Time = 1.435
13
14 Courant Number mean: 0.221923 max: 0.839903
15 smoothSolver: Solving for Ux, Initial residual = 8.53935e-06, Final residual = 8.53935e-06, No Iterations 0
16 smoothSolver: Solving for Uy, Initial residual = 9.71405e-06, Final residual = 9.71405e-06, No Iterations 0
17 DICPCG: Solving for p, Initial residual = 4.0223e-06, Final residual = 9.89693e-07, No Iterations 3
18 time step continuity errors : sum local = 8.15199e-09, global = 5.33614e-19, cumulative = -6.75012e-18
19 DICPCG: Solving for p, Initial residual = 2.38807e-06, Final residual = 8.44595e-07, No Iterations 3
20 time step continuity errors : sum local = 7.48751e-09, global = -4.42707e-19, cumulative = -7.19283e-18
21 ExecutionTime = 0.37 s ClockTime = 0 s
```

2.1.8 High Reynolds number flow

View the results in `paraFoam` and display the velocity vectors. The secondary vortices in the corners have increased in size. The user can then increase the Reynolds number further by decreasing the viscosity and then rerun the case. The number of vortices increases so the mesh resolution around them will need to increase in order to resolve the more complicated flow patterns. In addition, as the Reynolds number increases the time to convergence increases. The user should monitor residuals and extend the `endTime` accordingly to ensure convergence.

The need to increase spatial and temporal resolution then becomes impractical as the flow moves into the turbulent regime, where problems of solution stability may also occur. Of course, many engineering problems have very high Reynolds numbers and it is infeasible to bear the huge cost of solving the turbulent behaviour directly. Instead Reynolds-averaged simulation (RAS) turbulence models are used to solve for the mean flow behaviour and calculate the statistics of the fluctuations. The standard $k - \varepsilon$ model with wall functions will be used in this tutorial to solve the lid-driven cavity case with a Reynolds number of 10^4 . Two extra variables are solved for: k , the turbulent kinetic energy; and, ε , the turbulent dissipation rate. The additional equations and models for turbulent flow are implemented into a OpenFOAM solver called **pisoFoam**.

2.1.8.1 Pre-processing

Go back to the run directory and copy the **cavity** case in the `$FOAM_RUN/tutorials/incompressible/pisoFoam/RAS` directory (N.B: the **pisoFoam/RAS** directory), renaming it **cavityRAS** to avoid a clash with the existing **cavity** tutorial. Go into the new case directory.

```
run
cp -r $FOAM_TUTORIALS/incompressible/pisoFoam/RAS/cavity cavityRAS
cd cavityRAS
```

Generate the mesh by running **blockMesh** as before. Mesh grading towards the wall is not necessary when using the standard $k - \varepsilon$ model with wall functions since the flow in the near wall cell is modelled, rather than having to be resolved.

A range of wall function models is available in OpenFOAM that are applied as boundary conditions on individual patches. This enables different wall function models to be applied to different wall regions. The choice of wall function models are specified through the turbulent viscosity field, ν_t in the *0/nut* file:

```
17 dimensions      [0 2 -1 0 0 0 0];
18 internalField   uniform 0;
19 boundaryField
20 {
21     movingWall
22     {
23         type          nutkWallFunction;
24         value         uniform 0;
25     }
26     fixedWalls
27     {
28         type          nutkWallFunction;
29         value         uniform 0;
30     }
31     frontAndBack
32     {
33         type          empty;
34     }
35 }
36
37 }
38 // ****
39
```

This case uses standard wall functions, specified by the **nutkWallFunction** type on the **movingWall** and **fixedWalls** patches. Other wall function models include the rough wall functions, specified though the **nutRoughWallFunction** keyword.

The user should now open the field files for k and ε (*0/k* and *0/epsilon*) and examine their boundary conditions. For a wall boundary condition, ε is assigned a **epsilonWallFunction**

boundary condition and a `kqRwallFunction` boundary condition is assigned to k . The latter is a generic boundary condition that can be applied to any field that are of a turbulent kinetic energy type, *e.g.* k , q or Reynolds Stress R . The initial values for k and ε are set using an estimated fluctuating component of velocity \mathbf{U}' and a turbulent length scale, l . k and ε are defined in terms of these parameters as follows:

$$k = \frac{1}{2} \overline{\mathbf{U}' \cdot \mathbf{U}'} \quad (2.8)$$

$$\varepsilon = \frac{C_\mu^{0.75} k^{1.5}}{l} \quad (2.9)$$

where C_μ is a constant of the $k - \varepsilon$ model equal to 0.09. For a Cartesian coordinate system, k is given by:

$$k = \frac{1}{2} (U_x'^2 + U_y'^2 + U_z'^2) \quad (2.10)$$

where $U_x'^2$, $U_y'^2$ and $U_z'^2$ are the fluctuating components of velocity in the x , y and z directions respectively. Let us assume the initial turbulence is isotropic, *i.e.* $U_x'^2 = U_y'^2 = U_z'^2$, and equal to 5% of the lid velocity and that l , is equal to 5% of the box width, 0.1 m, then k and ε are given by:

$$U_x' = U_y' = U_z' = \frac{5}{100} 1 \text{ m s}^{-1} \quad (2.11)$$

$$\Rightarrow k = \frac{3}{2} \left(\frac{5}{100} \right)^2 \text{ m}^2 \text{s}^{-2} = 3.75 \times 10^{-3} \text{ m}^2 \text{s}^{-2} \quad (2.12)$$

$$\varepsilon = \frac{C_\mu^{0.75} k^{1.5}}{l} \approx 7.54 \times 10^{-3} \text{ m}^2 \text{s}^{-3} \quad (2.13)$$

These form the initial conditions for k and ε . The initial conditions for \mathbf{U} and p are $(0, 0, 0)$ and 0 respectively as before.

Turbulence modelling includes a range of methods, *e.g.* RAS or large-eddy simulation (LES), that are provided in OpenFOAM. The choice of turbulence modelling method is selectable at run-time through the `simulationType` keyword in `turbulenceProperties` dictionary. The user can view this file in the `constant` directory:

```

17  simulationType RAS;
18
19  RAS
20  {
21      RASModel      kEpsilon;
22
23      turbulence    on;
24
25      printCoeffs   on;
26  }
27 // ****
28

```

The options for `simulationType` are `laminar`, `RAS` and `LES`. With `RAS` selected in this case, the choice of RAS modelling is specified in a `RAS` subdictionary. The turbulence model is selected by the `RASModel` entry from a long list of available models that are listed in Table ???. The `kEpsilon` model should be selected which is the standard $k - \varepsilon$ model; the user should also ensure that `turbulence` calculation is switched `on`.

The coefficients for each turbulence model are stored within the respective code with a set of default values. Setting the optional switch called `printCoeffs` to `on` will make

the default values be printed to standard output, *i.e.* the terminal, when the model is called at run time. The coefficients are printed out as a sub-dictionary whose name is that of the model name with the word `Coeffs` appended, *e.g.* `kEpsilonCoeffs` in the case of the `kEpsilon` model. The coefficients of the model, *e.g.* `kEpsilon`, can be modified by optionally including (copying and pasting) that sub-dictionary within the `RAS` sub-dictionary and adjusting values accordingly.

The user should next set the laminar kinematic viscosity in the `transportProperties` dictionary. To achieve a Reynolds number of 10^4 , a kinematic viscosity of 10^{-5} m is required based on the Reynolds number definition given in Equation 2.1.

Finally the user should set the `startTime`, `stopTime`, `deltaT` and the `writeInterval` in the `controlDict`. Set `deltaT` to 0.005 s to satisfy the Courant number restriction and the `endTime` to 10 s.

2.1.8.2 Running the code

Execute `pisoFoam` by entering the case directory and typing “`pisoFoam`” in a terminal. In this case, where the viscosity is low, the boundary layer next to the moving lid is very thin and the cells next to the lid are comparatively large so the velocity at their centres are much less than the lid velocity. In fact, after ≈ 100 time steps it becomes apparent that the velocity in the cells adjacent to the lid reaches an upper limit of around 0.2 m s^{-1} hence the maximum Courant number does not rise much above 0.2. It is sensible to increase the solution time by increasing the time step to a level where the Courant number is much closer to 1. Therefore reset `deltaT` to 0.02 s and, on this occasion, set `startFrom` to `latestTime`. This instructs `pisoFoam` to read the start data from the latest time directory, *i.e.* `10.0`. The `endTime` should be set to 20 s since the run converges a lot slower than the laminar case. Restart the run as before and monitor the convergence of the solution. View the results at consecutive time steps as the solution progresses to see if the solution converges to a steady-state or perhaps reaches some periodically oscillating state. In the latter case, convergence may never occur but this does not mean the results are inaccurate.

2.1.9 Changing the case geometry

A user may wish to make changes to the geometry of a case and perform a new simulation. It may be useful to retain some or all of the original solution as the starting conditions for the new simulation. This is a little complex because the fields of the original solution are not consistent with the fields of the new case. However the `mapFields` utility can map fields that are inconsistent, either in terms of geometry or boundary types or both.

As an example, let us copy the `cavityClipped` case from the `tutorials` directory in the user’s `run` directory, and change into the `cavityClipped` directory:

```
run
cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity/cavityClipped .
cd cavityClipped
```

The case consists of the standard `cavity` geometry but with a square of length 0.04 m removed from the bottom right of the cavity, according to the `blockMeshDict` below:

```
17    convertToMeters 0.1;
18
```

```

19   vertices
20   (
21     (0 0 0)
22     (0.6 0 0)
23     (0 0.4 0)
24     (0.6 0.4 0)
25     (1 0.4 0)
26     (0 1 0)
27     (0.6 1 0)
28     (1 1 0)
29
30     (0 0 0.1)
31     (0.6 0 0.1)
32     (0 0.4 0.1)
33     (0.6 0.4 0.1)
34     (1 0.4 0.1)
35     (0 1 0.1)
36     (0.6 1 0.1)
37     (1 1 0.1)
38 );
39
40
41   blocks
42   (
43     hex (0 1 3 2 8 9 11 10) (12 8 1) simpleGrading (1 1 1)
44     hex (2 3 6 5 10 11 14 13) (12 12 1) simpleGrading (1 1 1)
45     hex (3 4 7 6 11 12 15 14) (8 12 1) simpleGrading (1 1 1)
46   );
47
48   edges
49   (
50   );
51
52   boundary
53   (
54     lid
55     {
56       type wall;
57       faces
58       (
59         (5 13 14 6)
60         (6 14 15 7)
61       );
62     }
63     fixedWalls
64     {
65       type wall;
66       faces
67       (
68         (0 8 10 2)
69         (2 10 13 5)
70         (7 15 12 4)
71         (4 12 11 3)
72         (3 11 9 1)
73         (1 9 8 0)
74       );
75     }
76     frontAndBack
77     {
78       type empty;
79       faces
80       (
81         (0 2 3 1)
82         (2 5 6 3)
83         (3 6 7 4)
84         (8 9 11 10)
85         (10 11 14 13)
86         (11 12 15 14)
87       );
88     }
89   );
90
91   mergePatchPairs
92   (
93   );
94
95 // ****

```

Generate the mesh with `blockMesh`. The patches are set accordingly as in previous cavity

cases. For the sake of clarity in describing the field mapping process, the upper wall patch is renamed `lid`, previously the `movingWall` patch of the original `cavity`.

In an inconsistent mapping, there is no guarantee that all the field data can be mapped from the source case. The remaining data must come from field files in the target case itself. Therefore field data must exist in the time directory of the target case before mapping takes place. In the `cavityClipped` case the mapping is set to occur at time 0.5 s, since the `startTime` is set to 0.5 s in the `controlDict`. Therefore the user needs to copy initial field data to that directory, *e.g.* from time 0:

```
cp -r 0 0.5
```

Before mapping the data, the user should view the geometry and fields at 0.5 s.

Now we wish to map the velocity and pressure fields from `cavity` onto the new fields of `cavityClipped`. Since the mapping is inconsistent, we need to edit the `mapFieldsDict` dictionary, located in the `system` directory. The dictionary contains 2 keyword entries: `patchMap` and `cuttingPatches`. The `patchMap` list contains a mapping of patches from the source fields to the target fields. It is used if the user wishes a patch in the target field to inherit values from a corresponding patch in the source field. In `cavityClipped`, we wish to inherit the boundary values on the `lid` patch from `movingWall` in `cavity` so we must set the `patchMap` as:

```
patchMap
(
    lid movingWall
);
```

The `cuttingPatches` list contains names of target patches whose values are to be mapped from the source internal field through which the target patch cuts. In this case, the `fixedWalls` patch is a `noSlip` condition so the internal values cannot be interpolated to the patch. Therefore the `cuttingPatches` list can simply be empty:

```
cuttingPatches
(
);
```

If the user does wish to interpolate internal values from the source case to the `fixedWalls` patch in the target case, a `fixedValue` boundary condition needs to be specified on the patch, whose `value` can then be overridden during the mapping process; the `fixedWalls` patch then needs to be included in the `cuttingPatches` list.

The user should run `mapFields`, from within the `cavityClipped` directory:

```
mapFields ../cavity
```

The user can view the mapped field as shown in Figure 2.14. The `fixedWalls` patch has not inherited values from the source case as we expected. The user can then run the case with `icoFoam`.

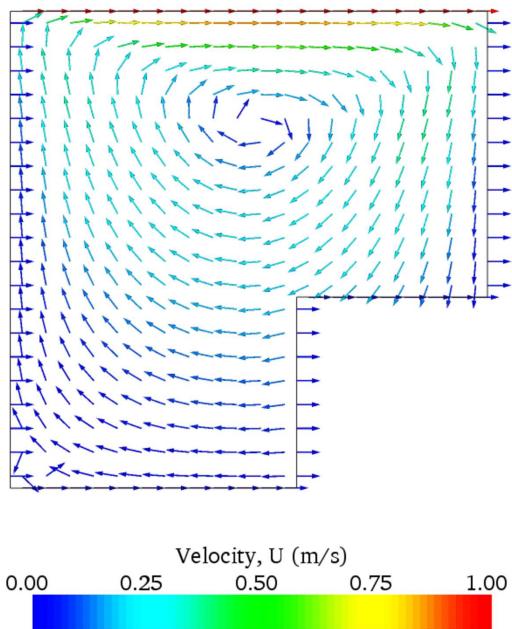


Figure 2.14: cavity solution velocity field mapped onto cavityClipped.

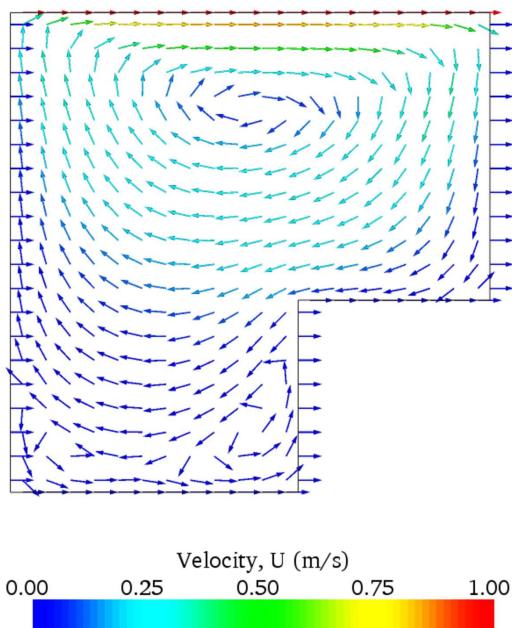


Figure 2.15: cavityClipped solution for velocity field.

2.1.10 Post-processing the modified geometry

Velocity glyphs can be generated for the case as normal, first at time 0.5 s and later at time 0.6 s, to compare the initial and final solutions. In addition, we provide an outline of the geometry which requires some care to generate for a 2D case. The user should select **Extract Block** from the **Filter** menu and, in the **Parameter** panel, highlight the patches of interest, namely the **lid** and **fixedWalls**. On clicking **Apply**, these items of geometry can be displayed by selecting **Wireframe** in the **Display** panel. Figure 2.15 displays the patches in black and shows vortices forming in the bottom corners of the modified geometry.

2.2 Stress analysis of a plate with a hole

This tutorial describes how to pre-process, run and post-process a case involving linear-elastic, steady-state stress analysis on a square plate with a circular hole at its centre. The plate dimensions are: side length 4 m and radius $R = 0.5$ m. It is loaded with a uniform traction of $\sigma = 10$ kPa over its left and right faces as shown in Figure 2.16. Two symmetry planes can be identified for this geometry and therefore the solution domain need only cover a quarter of the geometry, shown by the shaded area in Figure 2.16.

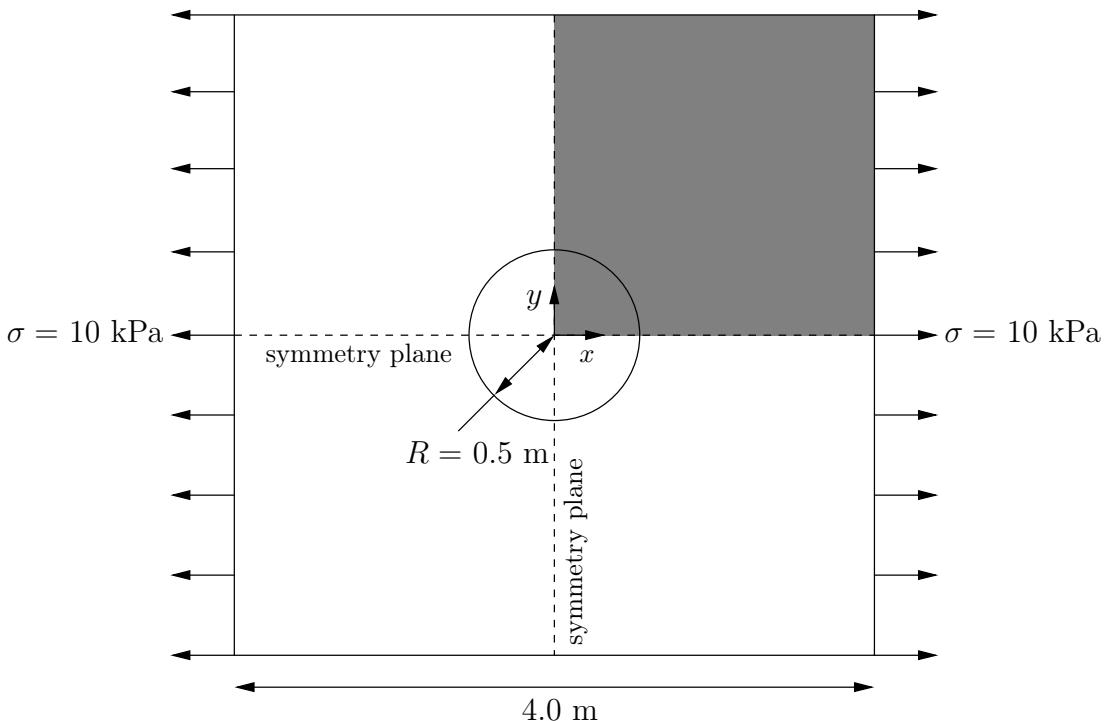


Figure 2.16: Geometry of the plate with a hole.

The problem can be approximated as 2-dimensional since the load is applied in the plane of the plate. In a Cartesian coordinate system there are two possible assumptions to take in regard to the behaviour of the structure in the third dimension: (1) the plane stress condition, in which the stress components acting out of the 2D plane are assumed to be negligible; (2) the plane strain condition, in which the strain components out of the 2D plane are assumed negligible. The plane stress condition is appropriate for solids whose third dimension is thin as in this case; the plane strain condition is applicable for solids where the third dimension is thick.

An analytical solution exists for loading of an infinitely large, thin plate with a circular hole. The solution for the stress normal to the vertical plane of symmetry is

$$(\sigma_{xx})_{x=0} = \begin{cases} \sigma \left(1 + \frac{R^2}{2y^2} + \frac{3R^4}{2y^4} \right) & \text{for } |y| \geq R \\ 0 & \text{for } |y| < R \end{cases} \quad (2.14)$$

Results from the simulation will be compared with this solution. At the end of the tutorial, the user can: investigate the sensitivity of the solution to mesh resolution and mesh grading; and, increase the size of the plate in comparison to the hole to try to estimate the error in comparing the analytical solution for an infinite plate to the solution of this problem of a finite plate.

2.2.1 Mesh generation

The domain consists of four blocks, some of which have arc-shaped edges. The block structure for the part of the mesh in the $x-y$ plane is shown in Figure 2.17. As already mentioned in section 2.1.1.1, all geometries are generated in 3 dimensions in OpenFOAM even if the case is to be as a 2 dimensional problem. Therefore a dimension of the block in the z direction has to be chosen; here, 0.5 m is selected. It does not affect the solution since the traction boundary condition is specified as a stress rather than a force, thereby making the solution independent of the cross-sectional area.

The user should change to the `run` directory and copy the `plateHole` case into it from the `$FOAM_TUTORIALS/stressAnalysis/solidDisplacementFoam` directory. The user should then go into the `plateHole` directory and open the `blockMeshDict` file in an editor, as listed below

```

17 convertToMeters 1;
18
19 vertices
20 (
21     (0.5 0 0)
22     (1 0 0)
23     (2 0 0)
24     (2 0.707107 0)
25     (0.707107 0.707107 0)
26     (0.353553 0.353553 0)
27     (2 2 0)
28     (0.707107 2 0)
29     (0 2 0)
30     (0 1 0)
31     (0 0.5 0)
32     (0.5 0 0.5)
33     (1 0 0.5)
34     (2 0 0.5)
35     (2 0.707107 0.5)
36     (0.707107 0.707107 0.5)
37     (0.353553 0.353553 0.5)
38     (2 2 0.5)
39     (0.707107 2 0.5)
40     (0 2 0.5)
41     (0 1 0.5)
42     (0 0.5 0.5)
43 );
44
45 blocks
46 (
47     hex (5 4 9 10 16 15 20 21) (10 10 1) simpleGrading (1 1 1)
48     hex (0 1 4 5 11 12 15 16) (10 10 1) simpleGrading (1 1 1)
49     hex (1 2 3 4 12 13 14 15) (20 10 1) simpleGrading (1 1 1)
50     hex (4 3 6 7 15 14 17 18) (20 20 1) simpleGrading (1 1 1)
51     hex (9 4 7 8 20 15 18 19) (10 20 1) simpleGrading (1 1 1)
52 );

```

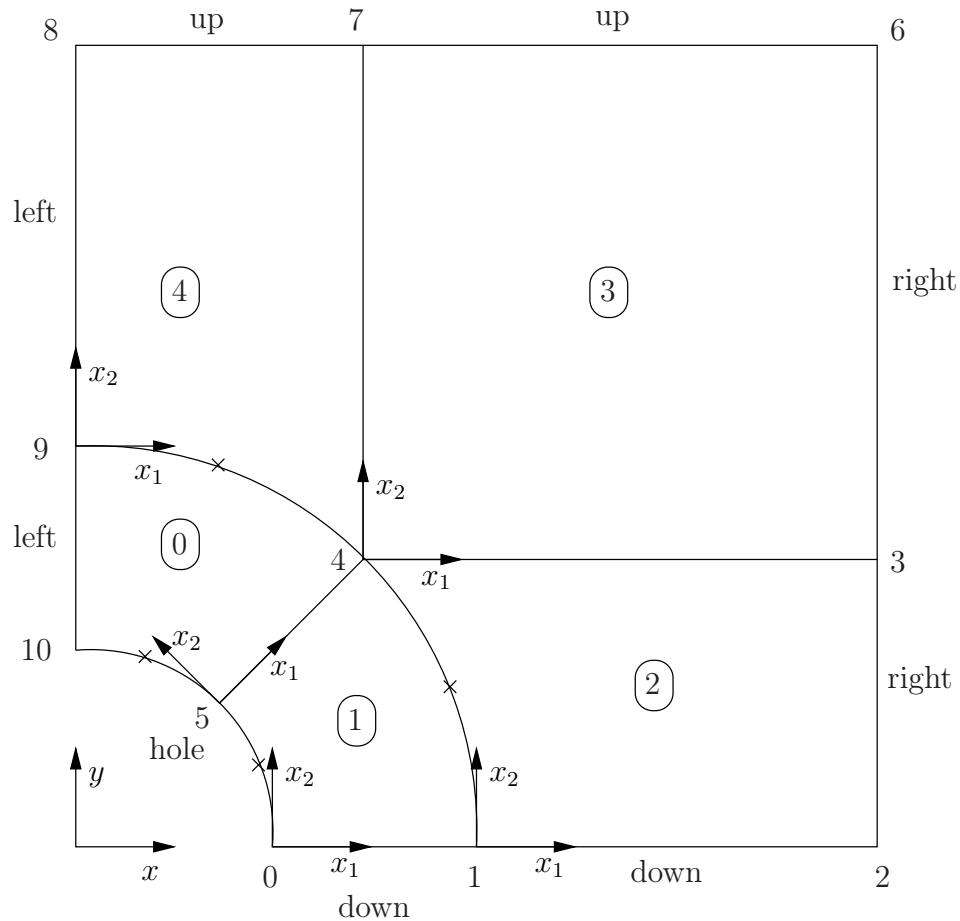


Figure 2.17: Block structure of the mesh for the plate with a hole.

```

53
54 edges
55 (
56     arc 0 5 (0.469846 0.17101 0)
57     arc 5 10 (0.17101 0.469846 0)
58     arc 1 4 (0.939693 0.34202 0)
59     arc 4 9 (0.34202 0.939693 0)
60     arc 11 16 (0.469846 0.17101 0.5)
61     arc 16 21 (0.17101 0.469846 0.5)
62     arc 12 15 (0.939693 0.34202 0.5)
63     arc 15 20 (0.34202 0.939693 0.5)
64 );
65
66 boundary
67 (
68     left
69     {
70         type symmetryPlane;
71         faces
72         (
73             (8 9 20 19)
74             (9 10 21 20)
75         );
76     }
77     right
78     {
79         type patch;
80         faces
81         (
82             (2 3 14 13)
83             (3 6 17 14)
84         );
85     }
86     down
87     {

```

```

88         type symmetryPlane;
89         faces
90         (
91             (0 1 12 11)
92             (1 2 13 12)
93         );
94     }
95     up
96     {
97         type patch;
98         faces
99         (
100            (7 8 19 18)
101            (6 7 18 17)
102        );
103    }
104    hole
105    {
106        type patch;
107        faces
108        (
109            (10 5 16 21)
110            (5 0 11 16)
111        );
112    }
113    frontAndBack
114    {
115        type empty;
116        faces
117        (
118            (10 9 4 5)
119            (5 4 1 0)
120            (1 4 3 2)
121            (4 7 6 3)
122            (4 9 8 7)
123            (21 16 15 20)
124            (16 11 12 15)
125            (12 13 14 15)
126            (15 14 17 18)
127            (15 18 19 20)
128        );
129    }
130  );
131
132 mergePatchPairs
133 (
134 );
135 // ****
136

```

Until now, we have only specified straight edges in the geometries of previous tutorials but here we need to specify curved edges. These are specified under the `edges` keyword entry which is a list of non-straight edges. The syntax of each list entry begins with the type of curve, including `arc`, `simpleSpline`, `polyLine` etc., described further in section 5.3.1. In this example, all the edges are circular and so can be specified by the `arc` keyword entry. The following entries are the labels of the start and end vertices of the arc and a point vector through which the circular arc passes.

The blocks in this `blockMeshDict` do not all have the same orientation. As can be seen in Figure 2.17 the x_2 direction of block 0 is equivalent to the $-x_1$ direction for block 4. This means care must be taken when defining the number and distribution of cells in each block so that the cells match up at the block faces.

6 patches are defined: one for each side of the plate, one for the hole and one for the front and back planes. The `left` and `down` patches are both a symmetry plane. Since this is a *geometric* constraint, it is included in the definition of the *mesh*, rather than being purely a specification on the boundary condition of the fields. Therefore they are defined as such using a special `symmetryPlane` type as shown in the `blockMeshDict`.

The `frontAndBack` patch represents the plane which is ignored in a 2D case. Again this is a geometric constraint so is defined within the mesh, using the `empty` type as shown in the

blockMeshDict. For further details of boundary types and geometric constraints, the user should refer to section 5.2.

The remaining patches are of the regular `patch` type. The mesh should be generated using `blockMesh` and can be viewed in `paraFoam` as described in section 2.1.2. It should appear as in Figure 2.18.

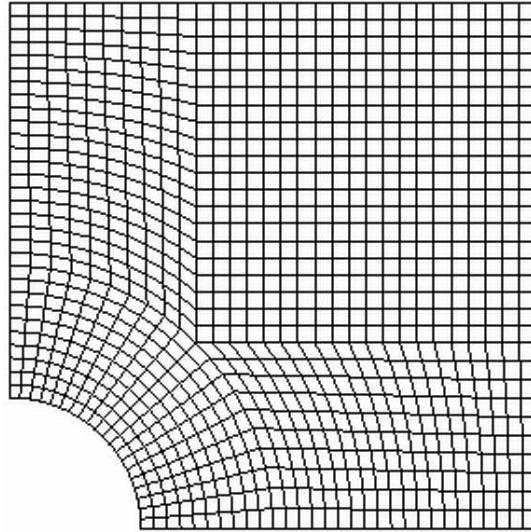


Figure 2.18: Mesh of the hole in a plate problem.

2.2.1.1 Boundary and initial conditions

Once the mesh generation is complete, the initial field with boundary conditions must be set. For a stress analysis case without thermal stresses, only displacement `D` needs to be set. The *O/D* is as follows:

```

17 dimensions      [0 1 0 0 0 0];
18 internalField   uniform (0 0 0);
20
21 boundaryField
22 {
23     left
24     {
25         type          symmetryPlane;
26     }
27     right
28     {
29         type          tractionDisplacement;
30         traction      uniform (10000 0 0);
31         pressure      uniform 0;
32         value         uniform (0 0 0);
33     }
34     down
35     {
36         type          symmetryPlane;
37     }
38     up
39     {
40         type          tractionDisplacement;
41         traction      uniform (0 0 0);
42         pressure      uniform 0;
43         value         uniform (0 0 0);
44     }
45     hole
46     {
47         type          tractionDisplacement;

```

```

48     traction      uniform (0 0 0);
49     pressure      uniform 0;
50     value         uniform (0 0 0);
51   }
52   frontAndBack
53   {
54     type          empty;
55   }
56 }
57 // ****

```

Firstly, it can be seen that the displacement initial conditions are set to (0, 0, 0) m. The `left` and `down` patches **must** be both of `symmetryPlane` type since they are specified as such in the mesh description in the `constant/polyMesh/boundary` file. Similarly the `frontAndBack` patch is declared `empty`.

The other patches are traction boundary conditions, set by a specialist `traction` boundary type. The traction boundary conditions are specified by a linear combination of: (1) a boundary traction vector under keyword `traction`; (2) a pressure that produces a traction normal to the boundary surface that is defined as negative when pointing out of the surface, under keyword `pressure`. The `up` and `hole` patches are zero traction so the boundary traction and pressure are set to zero. For the `right` patch the traction should be (1e4, 0, 0) Pa and the pressure should be 0 Pa.

2.2.1.2 Mechanical properties

The physical properties for the case are set in the `mechanicalProperties` dictionary in the `constant` directory. For this problem, we need to specify the mechanical properties of steel given in Table 2.1. In the mechanical properties dictionary, the user must also set `planeStress` to `yes`.

Property	Units	Keyword	Value
Density	kg m^{-3}	<code>rho</code>	7854
Young's modulus	Pa	<code>E</code>	2×10^{11}
Poisson's ratio	—	<code>nu</code>	0.3

Table 2.1: Mechanical properties for steel

2.2.1.3 Thermal properties

The temperature field variable `T` is present in the `solidDisplacementFoam` solver since the user may opt to solve a thermal equation that is coupled with the momentum equation through the thermal stresses that are generated. The user specifies at run time whether OpenFOAM should solve the thermal equation by the `thermalStress` switch in the `thermalProperties` dictionary. This dictionary also sets the thermal properties for the case, *e.g.* for steel as listed in Table 2.2.

In this case we do not want to solve for the thermal equation. Therefore we must set the `thermalStress` keyword entry to `no` in the `thermalProperties` dictionary.

2.2.1.4 Control

As before, the information relating to the control of the solution procedure are read in from the `controlDict` dictionary. For this case, the `startTime` is 0 s. The time step is not

Property	Units	Keyword	Value
Specific heat capacity	Jkg ⁻¹ K ⁻¹	C	434
Thermal conductivity	Wm ⁻¹ K ⁻¹	k	60.5
Thermal expansion coeff.	K ⁻¹	alpha	1.1 × 10 ⁻⁵

Table 2.2: Thermal properties for steel

important since this is a steady state case; in this situation it is best to set the time step `deltaT` to 1 so it simply acts as an iteration counter for the steady-state case. The `endTime`, set to 100, then acts as a limit on the number of iterations. The `writeInterval` can be set to 20.

The `controlDict` entries are as follows:

```

17 application solidDisplacementFoam;
18 startFrom startTime;
19 startTime 0;
20 stopAt endTime;
21 endTime 100;
22 deltaT 1;
23 writeControl timeStep;
24 writeInterval 20;
25 purgeWrite 0;
26 writeFormat ascii;
27 writePrecision 6;
28 writeCompression off;
29 timeFormat general;
30 timePrecision 6;
31 graphFormat raw;
32 runTimeModifiable true;
33 // ****

```

2.2.1.5 Discretisation schemes and linear-solver control

Let us turn our attention to the `fvSchemes` dictionary. Firstly, the problem we are analysing is steady-state so the user should select `SteadyState` for the time derivatives in `timeScheme`. This essentially switches off the time derivative terms. Not all solvers, especially in fluid dynamics, work for both steady-state and transient problems but `solidDisplacementFoam` does work, since the base algorithm is the same for both types of simulation.

The momentum equation in linear-elastic stress analysis includes several explicit terms containing the gradient of displacement. The calculations benefit from accurate and smooth evaluation of the gradient. Normally, in the finite volume method the discretisation is based on Gauss's theorem. The Gauss method is sufficiently accurate for most purposes but, in this case, the least squares method will be used. The user should therefore open the `fvSchemes` dictionary in the `system` directory and ensure the `leastSquares` method is selected for the `grad(U)` gradient discretisation scheme in the `gradSchemes` sub-dictionary:

```

17 d2dt2Schemes
18 {
19     default      steadyState;
20 }
21
22 ddtSchemes
23 {
24     default      Euler;
25 }
26
27 gradSchemes
28 {
29     default      leastSquares;
30     grad(D)    leastSquares;
31     grad(T)    leastSquares;
32 }
33
34 divSchemes
35 {
36     default      none;
37     div(sigmaD) Gauss linear;
38 }
39
40 laplacianSchemes
41 {
42     default      none;
43     laplacian(DD,D) Gauss linear corrected;
44     laplacian(DT,T) Gauss linear corrected;
45 }
46
47 interpolationSchemes
48 {
49     default      linear;
50 }
51
52 snGradSchemes
53 {
54     default      none;
55 }
56
57 // ****

```

The *fvSolution* dictionary in the *system* directory controls the linear equation solvers and algorithms used in the solution. The user should first look at the *solvers* sub-dictionary and notice that the choice of **solver** for D is GAMG. The solver **tolerance** should be set to 10^{-6} for this problem. The solver relative tolerance, denoted by **relTol**, sets the required reduction in the residuals within each iteration. It is uneconomical to set a tight (low) relative tolerance within each iteration since a lot of terms in each equation are explicit and are updated as part of the segregated iterative procedure. Therefore a reasonable value for the relative tolerance is 0.01, or possibly even higher, say 0.1, or in some cases even 0.9 (as in this case).

```

17 solvers
18 {
19     "(D|T)"
20     {
21         solver      GAMG;
22         tolerance   1e-06;
23         relTol     0.9;
24         smoother    GaussSeidel;
25         nCellsInCoarsestLevel 20;
26     }
27 }
28
29 stressAnalysis
30 {
31     compactNormalStress yes;
32     nCorrectors   1;
33     D            1e-06;
34 }
35
36 // ****

```

The `fvSolution` dictionary contains a sub-dictionary, `stressAnalysis` that contains some control parameters specific to the application solver. Firstly there is `nCorrectors` which specifies the number of outer loops around the complete system of equations, including traction boundary conditions *within each time step*. Since this problem is steady-state, we are performing a set of iterations towards a converged solution with the 'time step' acting as an iteration counter. We can therefore set `nCorrectors` to 1.

The `D` keyword specifies a convergence tolerance for the outer iteration loop, *i.e.* sets a level of initial residual below which solving will cease. It should be set to the desired solver tolerance specified earlier, 10^{-6} for this problem.

2.2.2 Running the code

The user should run the code here in the background from the command line as specified below, so he/she can look at convergence information in the log file afterwards.

```
solidDisplacementFoam > log &
```

The user should check the convergence information by viewing the generated *log* file which shows the number of iterations and the initial and final residuals of the displacement in each direction being solved. The final residual should always be less than 0.9 times the initial residual as this iteration tolerance set. Once both initial residuals have dropped below the convergence tolerance of 10^{-6} the run has converged and can be stopped by killing the batch job.

2.2.3 Post-processing

Post processing can be performed as in section 2.1.4. The `solidDisplacementFoam` solver outputs the stress field σ as a symmetric tensor field `sigma`. This is consistent with the way variables are usually represented in OpenFOAM solvers by the mathematical symbol by which they are represented; in the case of Greek symbols, the variable is named phonetically.

For post-processing individual scalar field components, σ_{xx} , σ_{xy} *etc.*, can be generated by running the `postProcess` utility as before in section 2.1.5.7, this time on `sigma`:

```
postProcess -func "components(sigma)"
```

Components named `sigmaxx`, `sigmaxy` *etc.* are written to time directories of the case. The σ_{xx} stresses can be viewed in `paraFoam` as shown in Figure 2.19.

We would like to compare the analytical solution of Equation 2.14 to our solution. We therefore must output a set of data of σ_{xx} along the left edge symmetry plane of our domain. The user may generate the required graph data using the `postProcess` utility with the `singleGraph` function. Unlike earlier examples of `postProcess` where no configuration is required, this example includes a `singleGraph` file pre-configured in the `system` directory. The sample line is set between (0.0, 0.5, 0.25) and (0.0, 2.0, 0.25), and the fields are specified in the `fields` list:

```
9  singleGraph
10 {
11     start    (0 0.5 0.25);
12     end      (0 2 0.25);
13     fields   (sigmaxx);
```

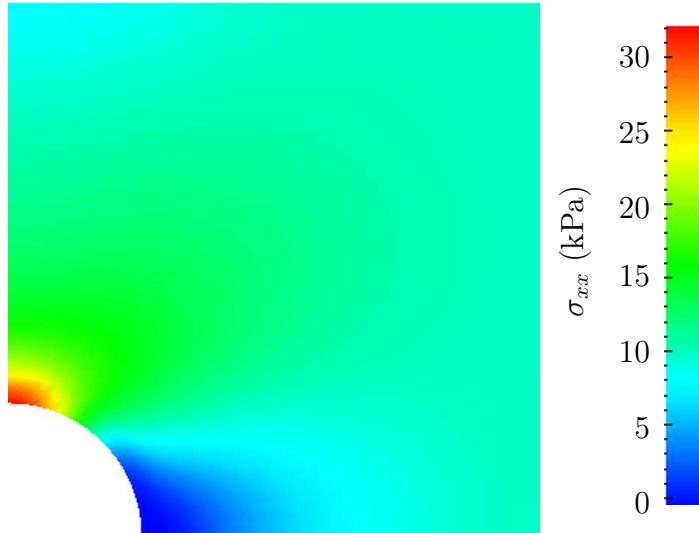


Figure 2.19: σ_{xx} stress field in the plate with hole.

```

14 #includeEtc "caseDicts/postProcessing/graphs/sampleDict.cfg"
15
16 setConfig
17 {
18     axis      y;
19 }
20
21 // Must be last entry
22 #includeEtc "caseDicts/postProcessing/graphs/graph.cfg"
23
24 }
25
26 // ****

```

The user should execute `postProcessing` with the `singleGraph` function:

```
postProcess -func "singleGraph"
```

Data is written in raw 2 column format into files within time subdirectories of a *postProcessing/singleGraph* directory, e.g. the data at $t = 100$ s is found within the file *singleGraph/100/line_sigmaxx.xy*. If the user has GnuPlot installed they launch it (by typing `gnuplot`) and then plot both the numerical data and analytical solution as follows:

```
plot [0.5:2] [0:] "postProcessing/singleGraph/100/line_sigmaxx.xy",
1e4*(1+(0.125/(x**2))+(0.09375/(x**4)))
```

An example plot is shown in Figure 2.20.

2.2.4 Exercises

The user may wish to experiment with `solidDisplacementFoam` by trying the following exercises:

2.2.4.1 Increasing mesh resolution

Increase the mesh resolution in each of the x and y directions. Use `mapFields` to map the final coarse mesh results from section 2.2.3 to the initial conditions for the fine mesh.

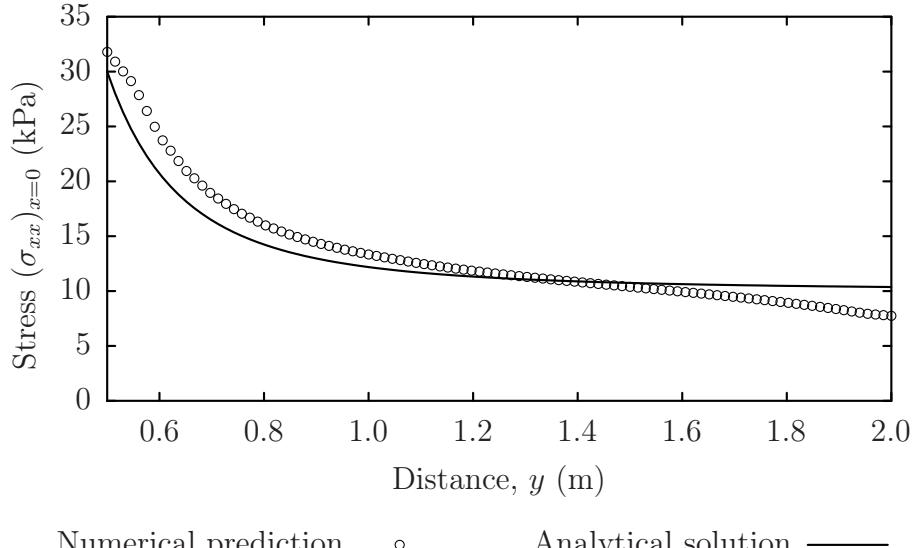


Figure 2.20: Normal stress along the vertical symmetry $(\sigma_{xx})_{x=0}$

2.2.4.2 Introducing mesh grading

Grade the mesh so that the cells near the hole are finer than those away from the hole. Design the mesh so that the ratio of sizes between adjacent cells is no more than 1.1 and so that the ratio of cell sizes between blocks is similar to the ratios within blocks. Mesh grading is described in section 2.1.6. Again use `mapFields` to map the final coarse mesh results from section 2.2.3 to the initial conditions for the graded mesh. Compare the results with those from the analytical solution and previous calculations. Can this solution be improved upon using the same number of cells with a different solution?

2.2.4.3 Changing the plate size

The analytical solution is for an infinitely large plate with a finite sized hole in it. Therefore this solution is not completely accurate for a finite sized plate. To estimate the error, increase the plate size while maintaining the hole size at the same value.

2.3 Breaking of a dam

In this tutorial we shall solve a problem of simplified dam break in 2 dimensions using the `interFoam`. The feature of the problem is a transient flow of two fluids separated by a sharp interface, or free surface. The two-phase algorithm in `interFoam` is based on the volume of fluid (VOF) method in which a species transport equation is used to determine the relative volume fraction of the two phases, or phase fraction α , in each computational cell. Physical properties are calculated as weighted averages based on this fraction. The nature of the VOF method means that an interface between the species is not explicitly computed, but rather emerges as a property of the phase fraction field. Since the phase fraction can have any value between 0 and 1, the interface is never sharply defined, but occupies a volume around the region where a sharp interface should exist.

The test setup consists of a column of water at rest located behind a membrane on the left side of a tank. At time $t = 0$ s, the membrane is removed and the column of water

collapses. During the collapse, the water impacts an obstacle at the bottom of the tank and creates a complicated flow structure, including several captured pockets of air. The geometry and the initial setup is shown in Figure 2.21.

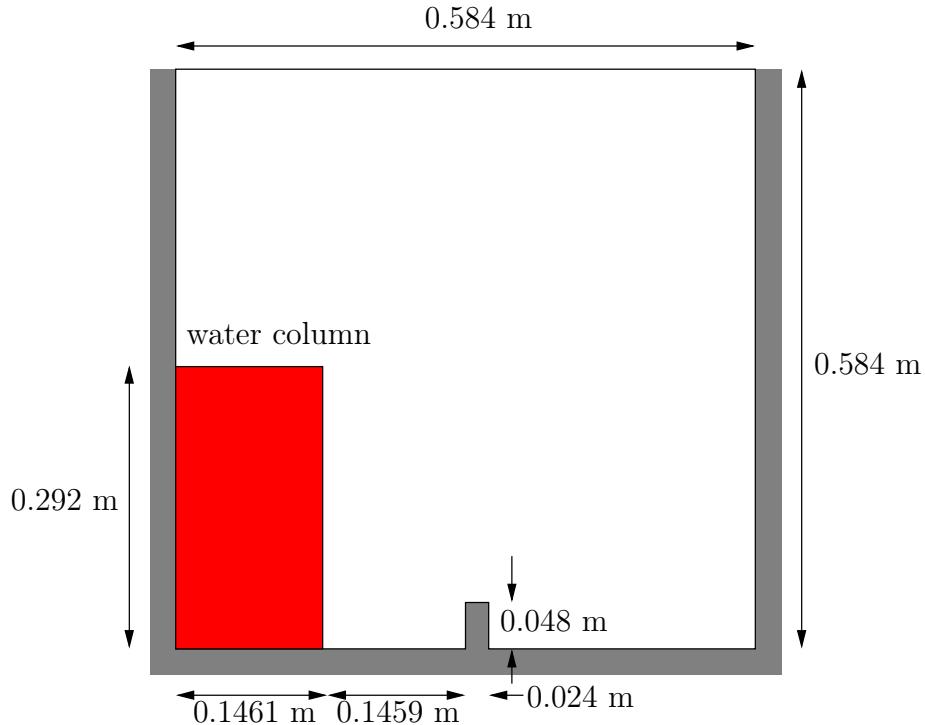


Figure 2.21: Geometry of the dam break.

2.3.1 Mesh generation

The user should go to their `run` directory and copy the `damBreak` case from the `$FOAM_TUTORIALS/multiphase/interFoam/laminar/damBreak` directory, *i.e.*

```
run
cp -r $FOAM_TUTORIALS/multiphase/interFoam/laminar/damBreak/damBreak .
```

Go into the `damBreak` case directory and generate the mesh running `blockMesh` as described previously. The `damBreak` mesh consist of 5 blocks; the `blockMeshDict` entries are given below.

```
17 convertToMeters 0.146;
18
19 vertices
20 (
21     (0 0 0)
22     (2 0 0)
23     (2.16438 0 0)
24     (4 0 0)
25     (0 0.32876 0)
26     (2 0.32876 0)
27     (2.16438 0.32876 0)
28     (4 0.32876 0)
29     (0 4 0)
30     (2 4 0)
31     (2.16438 4 0)
32     (4 4 0)
33     (0 0 0.1)
```

```

34      (2 0 0.1)
35      (2.16438 0 0.1)
36      (4 0 0.1)
37      (0 0.32876 0.1)
38      (2 0.32876 0.1)
39      (2.16438 0.32876 0.1)
40      (4 0.32876 0.1)
41      (0 4 0.1)
42      (2 4 0.1)
43      (2.16438 4 0.1)
44      (4 4 0.1)
45  );
46
47  blocks
48  (
49      hex (0 1 5 4 12 13 17 16) (23 8 1) simpleGrading (1 1 1)
50      hex (2 3 7 6 14 15 19 18) (19 8 1) simpleGrading (1 1 1)
51      hex (4 5 9 8 16 17 21 20) (23 42 1) simpleGrading (1 1 1)
52      hex (5 6 10 9 17 18 22 21) (4 42 1) simpleGrading (1 1 1)
53      hex (6 7 11 10 18 19 23 22) (19 42 1) simpleGrading (1 1 1)
54  );
55
56  edges
57  (
58  );
59
60  boundary
61  (
62      leftWall
63      {
64          type wall;
65          faces
66          (
67              (0 12 16 4)
68              (4 16 20 8)
69          );
70      }
71      rightWall
72      {
73          type wall;
74          faces
75          (
76              (7 19 15 3)
77              (11 23 19 7)
78          );
79      }
80      lowerWall
81      {
82          type wall;
83          faces
84          (
85              (0 1 13 12)
86              (1 5 17 13)
87              (5 6 18 17)
88              (2 14 18 6)
89              (2 3 15 14)
90          );
91      }
92      atmosphere
93      {
94          type patch;
95          faces
96          (
97              (8 20 21 9)
98              (9 21 22 10)
99              (10 22 23 11)
100         );
101     }
102   );
103
104  mergePatchPairs
105  (
106  );
107 // ****
108 // ****

```

2.3.2 Boundary conditions

The user can examine the boundary geometry generated by `blockMesh` by viewing the `boundary` file in the `constant/polyMesh` directory. The file contains a list of 5 boundary patches: `leftWall`, `rightWall`, `lowerWall`, `atmosphere` and `defaultFaces`. The user should notice the type of the patches. The `atmosphere` is a standard patch, *i.e.* has no special attributes, merely an entity on which boundary conditions can be specified. The `defaultFaces` patch is empty since the patch normal is in the direction we will not solve in this 2D case. The `leftWall`, `rightWall` and `lowerWall` patches are each a wall.

Like the generic patch, the wall type contains no geometric or topological information about the mesh and only differs from the plain patch in that it identifies the patch as a wall, should an application need to know, *e.g.* to apply special wall surface modelling. For example, the `interFoam` solver includes modelling of surface tension and can include wall adhesion at the contact point between the interface and wall surface. Wall adhesion models can be applied through a special boundary condition on the `alpha` (α) field, *e.g.* the `constantAlphaContactAngle` boundary condition, which requires the user to specify a static contact angle, `theta0`.

In this tutorial we would like to ignore surface tension effects between the wall and interface. We can do this by setting the static contact angle, $\theta_0 = 90^\circ$. However, rather than using the `constantAlphaContactAngle` boundary condition, the simpler `zeroGradient` can be applied to `alpha` on the walls.

The `top` boundary is free to the atmosphere so needs to permit both outflow and inflow according to the internal flow. We therefore use a combination of boundary conditions for pressure and velocity that does this while maintaining stability. They are:

- `totalPressure` which is a `fixedValue` condition calculated from specified total pressure p_0 and local velocity U ;
- `pressureInletOutletVelocity`, which applies `zeroGradient` on all components, except where there is inflow, in which case a `fixedValue` condition is applied to the *tangential* component;
- `inletOutlet`, which is a `zeroGradient` condition when flow outwards, `fixedValue` when flow is inwards.

At all wall boundaries, the `fixedFluxPressure` boundary condition is applied to the pressure field, which adjusts the pressure gradient so that the boundary flux matches the velocity boundary condition for solvers that include body forces such as gravity and surface tension.

The `defaultFaces` patch representing the front and back planes of the 2D problem, is, as usual, an empty type.

2.3.3 Setting initial field

Unlike the previous cases, we shall now specify a non-uniform initial condition for the phase fraction α_{water} where

$$\alpha_{\text{water}} = \begin{cases} 1 & \text{for the water phase} \\ 0 & \text{for the air phase} \end{cases} \quad (2.15)$$

This will be done by running the `setFields` utility. It requires a `setFieldsDict` dictionary, located in the `system` directory, whose entries for this case are shown below.

```

17   defaultFieldValues
18   (
19     volScalarFieldValue alpha.water 0
20   );
21
22 regions
23 (
24   boxToCell
25   {
26     box (0 0 -1) (0.1461 0.292 1);
27     fieldValues
28     (
29       volScalarFieldValue alpha.water 1
30     );
31   }
32 );
33 // ****
34

```

The `defaultFieldValues` sets the default value of the fields, *i.e.* the value the field takes unless specified otherwise in the `regions` sub-dictionary. That sub-dictionary contains a list of subdictionaries containing `fieldValues` that override the defaults in a specified region. The region is expressed in terms of a `topoSetSource` that creates a set of points, cells or faces based on some topological constraint. Here, `boxToCell` creates a bounding box within a vector minimum and maximum to define the set of cells of the water region. The phase fraction α_{water} is defined as 1 in this region.

The `setFields` utility reads fields from file and, after re-calculating those fields, will write them back to file. Because the files are then overridden, it is recommended that a backup is made before `setFields` is executed. In the `damBreak` tutorial, the `alpha.water` field is initially stored as a backup named `alpha.water.orig`. Before running `setFields`, the user first needs to copy `alpha.water.orig` to `alpha.water`, *e.g.* by typing:

```
cp 0/alpha.water.orig 0/alpha.water
```

The user should then execute `setFields` like any other utility by:

```
setFields
```

Using `paraFoam`, check that the initial `alpha.water` field corresponds to the desired distribution as in Figure 2.22.

2.3.4 Fluid properties

Let us examine the `transportProperties` file in the `constant` directory. The dictionary first contains the names of each fluid phase in the `phases` list, here `water` and `air`. The material properties for each fluid are then separated into two dictionaries `water` and `air`. The transport model for each phase is selected by the `transportModel` keyword. The user should select `Newtonian` in which case the kinematic viscosity is single valued and specified under the keyword `nu`. The viscosity parameters for other models, *e.g.*`CrossPowerLaw`, would otherwise be specified as described in section 7.3. The density is specified under the keyword `rho`.

The surface tension between the two phases is specified by the keyword `sigma`. The values used in this tutorial are listed in Table 2.3.

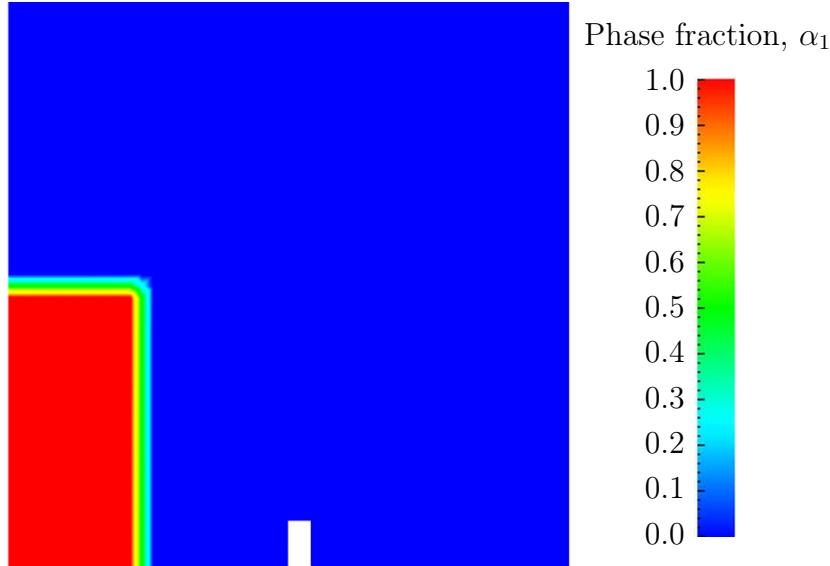


Figure 2.22: Initial conditions for phase fraction α_{water} .

water properties			
Kinematic viscosity	$\text{m}^2 \text{s}^{-1}$	nu	1.0×10^{-6}
Density	kg m^{-3}	rho	1.0×10^3
air properties			
Kinematic viscosity	$\text{m}^2 \text{s}^{-1}$	nu	1.48×10^{-5}
Density	kg m^{-3}	rho	1.0
Properties of both phases			
Surface tension	N m^{-1}	sigma	0.07

Table 2.3: Fluid properties for the `damBreak` tutorial

Gravitational acceleration is uniform across the domain and is specified in a file named `g` in the `constant` directory. Unlike a normal field file, *e.g.* `U` and `p`, `g` is a `uniformDimensionedVectorField` and so simply contains a set of `dimensions` and a `value` that represents $(0, 9.81, 0)$ m s^{-2} for this tutorial:

```
17 dimensions      [0 1 -2 0 0 0];
18 value          (0 -9.81 0);
19 // ****
20 // **** //
```

2.3.5 Turbulence modelling

As in the cavity example, the choice of turbulence modelling method is selectable at run-time through the `simulationType` keyword in `turbulenceProperties` dictionary. In this example, we wish to run without turbulence modelling so we set `laminar`:

```
17 simulationType laminar;
18 // ****
19 // **** //
```

2.3.6 Time step control

Time step control is an important issue in transient simulation and the surface-tracking algorithm in interface capturing solvers. The Courant number Co needs to be limited depending on the choice of algorithm: with the “explicit” MULES algorithm, an upper limit of $Co \approx 0.25$ for stability is typical in the region of the interface; but with “semi-implicit” MULES, specified by the `MULESCorr` keyword in the `fvSolution` file, there is really no upper limit in Co for stability, but instead the level is determined by requirements of temporal accuracy.

In general it is difficult to specify a fixed time-step to satisfy the Co criterion, so `interFoam` offers automatic adjustment of the time step as standard in the `controlDict`. The user should specify `adjustTimeStep` to be `on` and the the maximum Co for the phase fields, `maxAlphaCo`, and other fields, `maxCo`, to be 1.0. The upper limit on time step `maxDeltaT` can be set to a value that will not be exceeded in this simulation, *e.g.* 1.0.

By using automatic time step control, the steps themselves are never rounded to a convenient value. Consequently if we request that OpenFOAM saves results at a fixed number of time step intervals, the times at which results are saved are somewhat arbitrary. However even with automatic time step adjustment, OpenFOAM allows the user to specify that results are written at fixed times; in this case OpenFOAM forces the automatic time stepping procedure to adjust time steps so that it ‘hits’ on the exact times specified for write output. The user selects this with the `adjustableRunTime` option for `writeControl` in the `controlDict` dictionary. The `controlDict` dictionary entries should be:

```

17 application      interFoam;
18 startFrom       startTime;
19 startTime        0;
20 stopAt          endTime;
21 endTime          1;
22 deltaT          0.001;
23 writeControl    adjustableRunTime;
24 writeInterval   0.05;
25 purgeWrite      0;
26 writeFormat     ascii;
27 writePrecision  6;
28 writeCompression uncompressed;
29 timeFormat      general;
30 timePrecision   6;
31 runTimeModifiable yes;
32 adjustTimeStep  yes;
33 maxCo           1;
34 maxAlphaCo      1;
35 maxDeltaT       1;
36 // **** //
```

2.3.7 Discretisation schemes

The `interFoam` solver uses the multidimensional universal limiter for explicit solution (MULES) method, created by Henry Weller, to maintain boundedness of the phase fraction independent of underlying numerical scheme, mesh structure, *etc.* The choice of schemes for convection are therefore not restricted to those that are strongly stable or bounded, *e.g.* upwind differencing.

The convection schemes settings are made in the `divSchemes` sub-dictionary of the `fvSchemes` dictionary. In this example, the convection term in the momentum equation ($\nabla \cdot (\rho \mathbf{U} \mathbf{U})$), denoted by the `div(rho*phi,U)` keyword, uses `Gauss linearUpwind grad(U)` to produce good accuracy. The limited linear schemes require a coefficient ϕ as described in section 4.4.6. Here, we have opted for best stability with $\phi = 1.0$. The $\nabla \cdot (\mathbf{U} \alpha_1)$ term, represented by the `div(phi,alpha)` keyword uses the `vanLeer` scheme. The $\nabla \cdot (\mathbf{U}_{rb} \alpha_1)$ term, represented by the `div(phirb,alpha)` keyword, can use second order `linear` (central) differencing as boundedness is assured by the MULES algorithm.

The other discretised terms use commonly employed schemes so that the `fvSchemes` dictionary entries should therefore be:

```

17 ddtSchemes
18 {
19     default      Euler;
20 }
21
22 gradSchemes
23 {
24     default      Gauss linear;
25 }
26
27 divSchemes
28 {
29     div(rhoPhi,U) Gauss linearUpwind grad(U);
30     div(phi,alpha) Gauss vanLeer;
31     div(phirb,alpha) Gauss linear;
32     div((rho*nuEff)*dev2(T(grad(U)))) Gauss linear;
33 }
34
35 laplacianSchemes
36 {
37     default      Gauss linear corrected;
38 }
39
40 interpolationSchemes
41 {
42     default      linear;
43 }
44
45 snGradSchemes
46 {
47     default      corrected;
48 }
49 // ****

```

2.3.8 Linear-solver control

In the `fvSolution` file, the `alpha.water` sub-dictionary in `solvers` contains elements that are specific to `interFoam`. Of particular interest are the `nAlphaSubCycles` and `cAlpha` keywords. `nAlphaSubCycles` represents the number of sub-cycles within the α equation; sub-cycles are additional solutions to an equation within a given time step. It is used to enable the solution to be stable without reducing the time step and vastly increasing the solution time. Here we specify 2 sub-cycles, which means that the α equation is solved in $2\times$ half length time steps within each actual time step.

The `cAlpha` keyword is a factor that controls the compression of the interface where: 0 corresponds to no compression; 1 corresponds to conservative compression; and, anything larger than 1, relates to enhanced compression of the interface. We generally adopt a value of 1.0 which is employed in this example.

2.3.9 Running the code

Running of the code has been described in detail in previous tutorials. Try the following, that uses `tee`, a command that enables output to be written to both standard output and files:

```
cd $FOAM_RUN/damBreak
interFoam | tee log
```

The code will now be run interactively, with a copy of output stored in the `log` file.

2.3.10 Post-processing

Post-processing of the results can now be done in the usual way. The user can monitor the development of the phase fraction `alpha.water` in time, *e.g.* see Figure 2.23.

2.3.11 Running in parallel

The results from the previous example are generated using a fairly coarse mesh. We now wish to increase the mesh resolution and re-run the case. The new case will typically take a few hours to run with a single processor so, should the user have access to multiple processors, we can demonstrate the parallel processing capability of OpenFOAM.

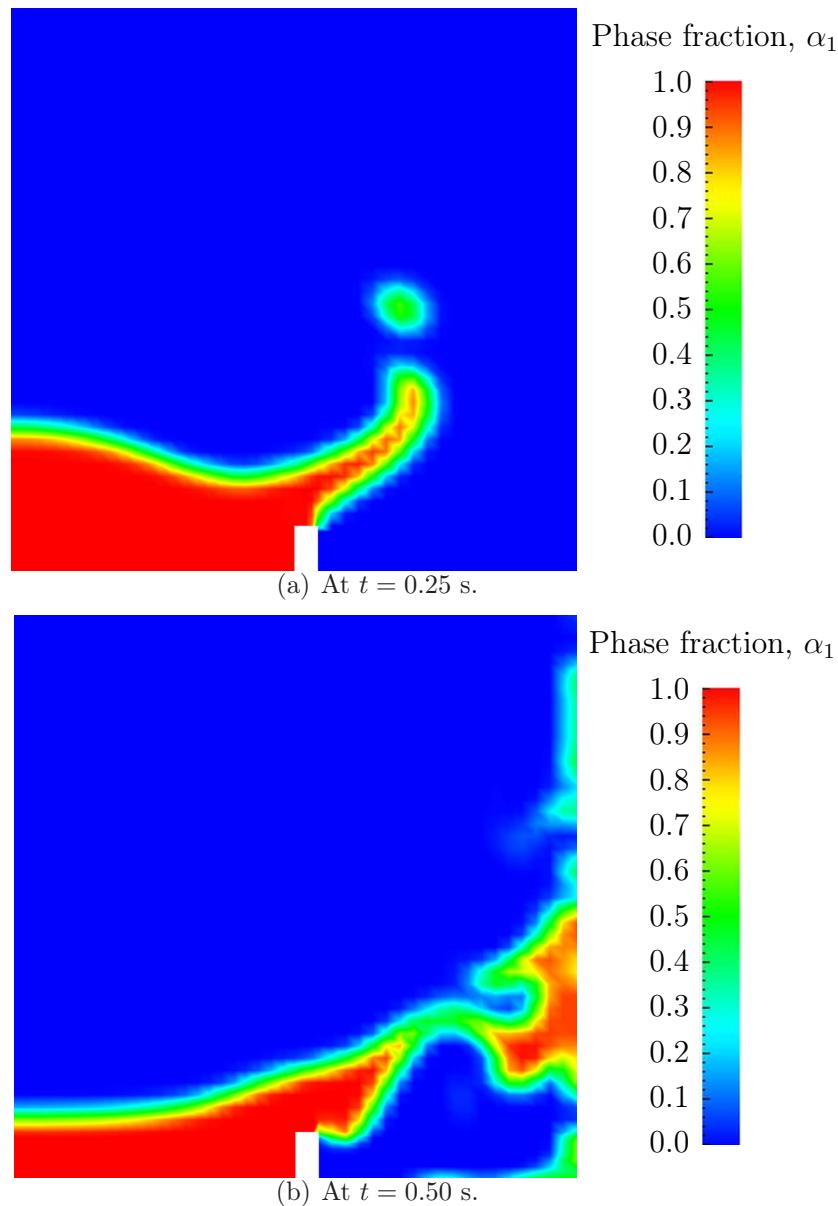
The user should first clone the `damBreak` case, *e.g.* by

```
run
foamCloneCase damBreak damBreakFine
```

Enter the new case directory and change the `blocks` description in the `blockMeshDict` dictionary to

```
blocks
(
    hex (0 1 5 4 12 13 17 16) (46 10 1) simpleGrading (1 1 1)
    hex (2 3 7 6 14 15 19 18) (40 10 1) simpleGrading (1 1 1)
    hex (4 5 9 8 16 17 21 20) (46 76 1) simpleGrading (1 2 1)
    hex (5 6 10 9 17 18 22 21) (4 76 1) simpleGrading (1 2 1)
    hex (6 7 11 10 18 19 23 22) (40 76 1) simpleGrading (1 2 1)
);
```

Here, the entry is presented as printed from the `blockMeshDict` file; in short the user must change the mesh densities, *e.g.* the `46 10 1` entry, and some of the mesh grading entries to `1 2 1`. Once the dictionary is correct, generate the mesh.

Figure 2.23: Snapshots of phase α .

As the mesh has now changed from the `damBreak` example, the user must re-initialise the phase field `alpha.water` in the `0` time directory since it contains a number of elements that is inconsistent with the new mesh. Note that there is no need to change the `U` and `p_rgh` fields since they are specified as `uniform` which is independent of the number of elements in the field. We wish to initialise the field with a sharp interface, *i.e.* its elements would have $\alpha = 1$ or $\alpha = 0$. Updating the field with `mapFields` may produce interpolated values $0 < \alpha < 1$ at the interface, so it is better to rerun the `setFields` utility. The user should copy the backup copy of the initial uniform α field, `0/alpha.water.orig`, to `0/alpha.water` before running `setFields`:

```
cp -r 0/alpha.water.orig 0/alpha.water
setFields
```

The method of parallel computing used by OpenFOAM is known as domain decomposition, in which the geometry and associated fields are broken into pieces and allocated to separate processors for solution. The first step required to run a parallel case is therefore to decompose the domain using the `decomposePar` utility. There is a dictionary associated with `decomposePar` named `decomposeParDict` which is located in the `system` directory of the tutorial case; also, like with many utilities, a default dictionary can be found in the directory of the source code of the specific utility, *i.e.* in `$FOAM_UTILITIES/parallelProcessing/decomposePar` for this case.

The first entry is `numberOfSubdomains` which specifies the number of subdomains into which the case will be decomposed, usually corresponding to the number of processors available for the case.

In this tutorial, the `method` of decomposition should be `simple` and the corresponding `simpleCoeffs` should be edited according to the following criteria. The domain is split into pieces, or subdomains, in the x , y and z directions, the number of subdomains in each direction being given by the vector `n`. As this geometry is 2 dimensional, the 3rd direction, z , cannot be split, hence n_z must equal 1. The n_x and n_y components of `n` split the domain in the x and y directions and must be specified so that the number of subdomains specified by n_x and n_y equals the specified `numberOfSubdomains`, *i.e.* $n_x n_y = \text{numberOfSubdomains}$. It is beneficial to keep the number of cell faces adjoining the subdomains to a minimum so, for a square geometry, it is best to keep the split between the x and y directions should be fairly even. The `delta` keyword should be set to 0.001.

For example, let us assume we wish to run on 4 processors. We would set `numberOfSubdomains` to 4 and `n` = $(2, 2, 1)$. When running `decomposePar`, we can see from the screen messages that the decomposition is distributed fairly even between the processors.

The user should consult section 3.4 for details of how to run a case in parallel; in this tutorial we merely present an example of running in parallel. We use the `openMPI` implementation of the standard message-passing interface (MPI). As a test here, the user can run in parallel on a single node, the local host only, by typing:

```
mpirun -np 4 interFoam -parallel > log &
```

The user may run on more nodes over a network by creating a file that lists the host names of the machines on which the case is to be run as described in section 3.4.3. The case should run in the background and the user can follow its progress by monitoring the `log` file as usual.

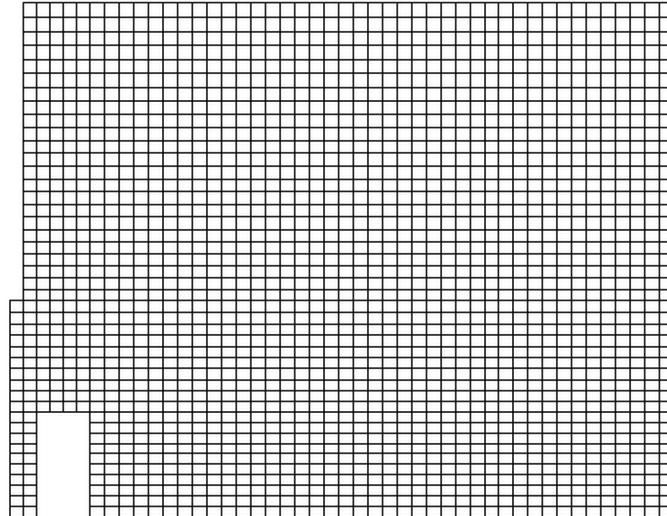


Figure 2.24: Mesh of processor 2 in parallel processed case.

2.3.12 Post-processing a case run in parallel

Once the case has completed running, the decomposed fields and mesh must be reassembled for post-processing using the `reconstructPar` utility. Simply execute it from the command line. The results from the fine mesh are shown in Figure 2.25. The user can see that the resolution of interface has improved significantly compared to the coarse mesh.

The user may also post-process a segment of the decomposed domain individually by simply treating the individual processor directory as a case in its own right. For example if the user starts `paraFoam` by

```
paraFoam -case processor1
```

then `processor1` will appear as a case module in ParaView. Figure 2.24 shows the mesh from processor 1 following the decomposition of the domain using the `simple` method.

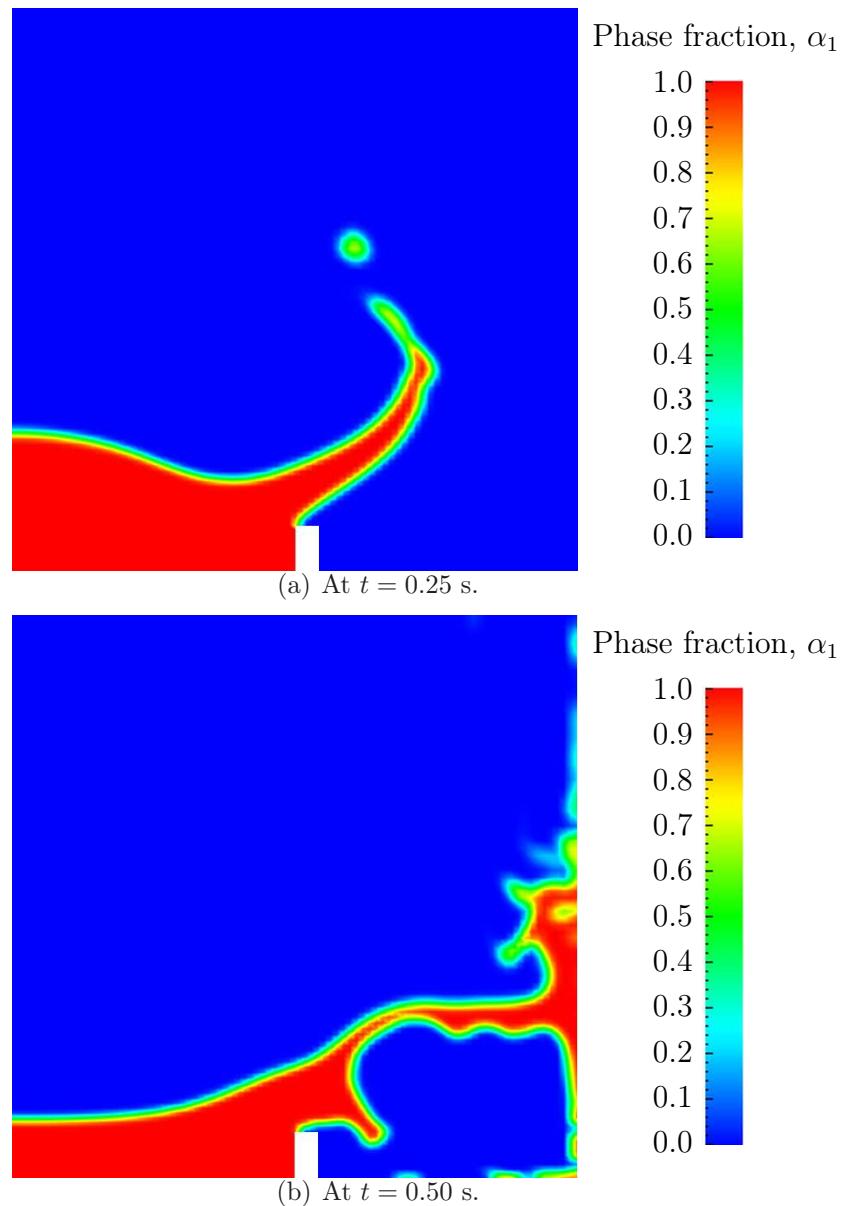


Figure 2.25: Snapshots of phase α with refined mesh.

Chapter 3

Applications and libraries

We should reiterate from the outset that OpenFOAM is a C++ library used primarily to create executables, known as *applications*. OpenFOAM is distributed with a large set of precompiled applications but users also have the freedom to create their own or modify existing ones. Applications are split into two main categories:

solvers that are each designed to solve a specific problem in computational continuum mechanics;

utilities that perform simple pre-and post-processing tasks, mainly involving data manipulation and algebraic calculations.

OpenFOAM is divided into a set of precompiled libraries that are dynamically linked during compilation of the solvers and utilities. Libraries such as those for physical models are supplied as source code so that users may conveniently add their own models to the libraries. This chapter gives an overview of solvers, utilities and libraries, their creation, modification, compilation and execution.

3.1 The programming language of OpenFOAM

In order to understand the way in which the OpenFOAM library works, some background knowledge of C++, the base language of OpenFOAM, is required; the necessary information will be presented in this chapter. Before doing so, it is worthwhile addressing the concept of language in general terms to explain some of the ideas behind object-oriented programming and our choice of C++ as the main programming language of OpenFOAM.

3.1.1 Language in general

The success of verbal language and mathematics is based on efficiency, especially in expressing abstract concepts. For example, in fluid flow, we use the term “velocity field”, which has meaning without any reference to the nature of the flow or any specific velocity data. The term encapsulates the idea of movement with direction and magnitude and relates to other physical properties. In mathematics, we can represent velocity field by a single symbol, *e.g.* \mathbf{U} , and express certain concepts using symbols, *e.g.* “the field of velocity magnitude” by $|\mathbf{U}|$. The advantage of mathematics over verbal language is its greater efficiency, making it possible to express complex concepts with extreme clarity.

The problems that we wish to solve in continuum mechanics are not presented in terms of intrinsic entities, or types, known to a computer, *e.g.* bits, bytes, integers. They are usually presented first in verbal language, then as partial differential equations in 3 dimensions of space and time. The equations contain the following concepts: scalars, vectors, tensors, and fields thereof; tensor algebra; tensor calculus; dimensional units. The solution to these equations involves discretisation procedures, matrices, solvers, and solution algorithms.

3.1.2 Object-orientation and C++

Programming languages that are object-oriented, such as C++, provide the mechanism — *classes* — to declare types and associated operations that are part of the verbal and mathematical languages used in science and engineering. Our velocity field introduced earlier can be represented in programming code by the symbol `U` and “the field of velocity magnitude” can be `mag(U)`. The velocity is a vector field for which there should exist, in an object-oriented code, a `vectorField` class. The velocity field `U` would then be an instance, or *object*, of the `vectorField` class; hence the term object-oriented.

The clarity of having objects in programming that represent physical objects and abstract entities should not be underestimated. The class structure concentrates code development to contained regions of the code, *i.e.* the classes themselves, thereby making the code easier to manage. New classes can be derived or inherit properties from other classes, *e.g.* the `vectorField` can be derived from a `vector` class and a `Field` class. C++ provides the mechanism of *template classes* such that the template class `Field<Type>` can represent a field of any `<Type>`, *e.g.* `scalar`, `vector`, `tensor`. The general features of the template class are passed on to any class created from the template. Templating and inheritance reduce duplication of code and create class hierarchies that impose an overall structure on the code.

3.1.3 Equation representation

A central theme of the OpenFOAM design is that the solver applications, written using the OpenFOAM classes, have a syntax that closely resembles the partial differential equations being solved. For example the equation

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot \phi \mathbf{U} - \nabla \cdot \mu \nabla \mathbf{U} = -\nabla p$$

is represented by the code

```
solve
(
    fvm::ddt(rho, U)
    + fvm::div(phi, U)
    - fvm::laplacian(mu, U)
    ==
    - fvc::grad(p)
);
```

This and other requirements demand that the principal programming language of OpenFOAM has object-oriented features such as inheritance, template classes, virtual functions and operator overloading. These features are not available in many languages that purport

to be object-orientated but actually have very limited object-orientated capability, such as FORTRAN-90. C++, however, possesses all these features while having the additional advantage that it is widely used with a standard specification so that reliable compilers are available that produce efficient executables. It is therefore the primary language of OpenFOAM.

3.1.4 Solver codes

Solver codes are largely procedural since they are a close representation of solution algorithms and equations, which are themselves procedural in nature. Users do not need a deep knowledge of object-orientation and C++ programming to write a solver but should know the principles behind object-orientation and classes, and to have a basic knowledge of some C++ code syntax. An understanding of the underlying equations, models and solution method and algorithms is far more important.

There is often little need for a user to immerse themselves in the code of any of the OpenFOAM classes. The essence of object-orientation is that the user should not have to go through the code of each class they use; merely the knowledge of the class' existence and its functionality are sufficient to use the class. A description of each class, its functions *etc.* is supplied with the OpenFOAM distribution in HTML documentation generated with Doxygen at <http://openfoam.org/docs/cpp>

3.2 Compiling applications and libraries

Compilation is an integral part of application development that requires careful management since every piece of code requires its own set instructions to access dependent components of the OpenFOAM library. In UNIX/Linux systems these instructions are often organised and delivered to the compiler using the standard UNIXmake utility. OpenFOAM uses its own wmake compilation script that is based on make but is considerably more versatile and easier to use (wmake can be used on any code, not only the OpenFOAM library). To understand the compilation process, we first need to explain certain aspects of C++ and its file structure, shown schematically in Figure 3.1. A class is defined through a set of instructions such as object construction, data storage and class member functions. The file that defines these functions — the class *definition* — takes a .C extension, *e.g.* a class nc would be written in the file *nc.C*. This file can be compiled independently of other code into a binary executable library file known as a shared object library with the .so file extension, *i.e.* *nc.so*. When compiling a piece of code, say *newApp.C*, that uses the nc class, *nc.C* need not be recompiled, rather *newApp.C* calls the *nc.so* library at runtime. This is known as *dynamic linking*.

3.2.1 Header .H files

As a means of checking errors, the piece of code being compiled must know that the classes it uses and the operations they perform actually exist. Therefore each class requires a class *declaration*, contained in a header file with a .H file extension, *e.g.* *nc.H*, that includes the names of the class and its functions. This file is included at the beginning of any piece of code using the class, using the #include directive described below, including the class declaration code itself. Any piece of .C code can resource any number of classes and must

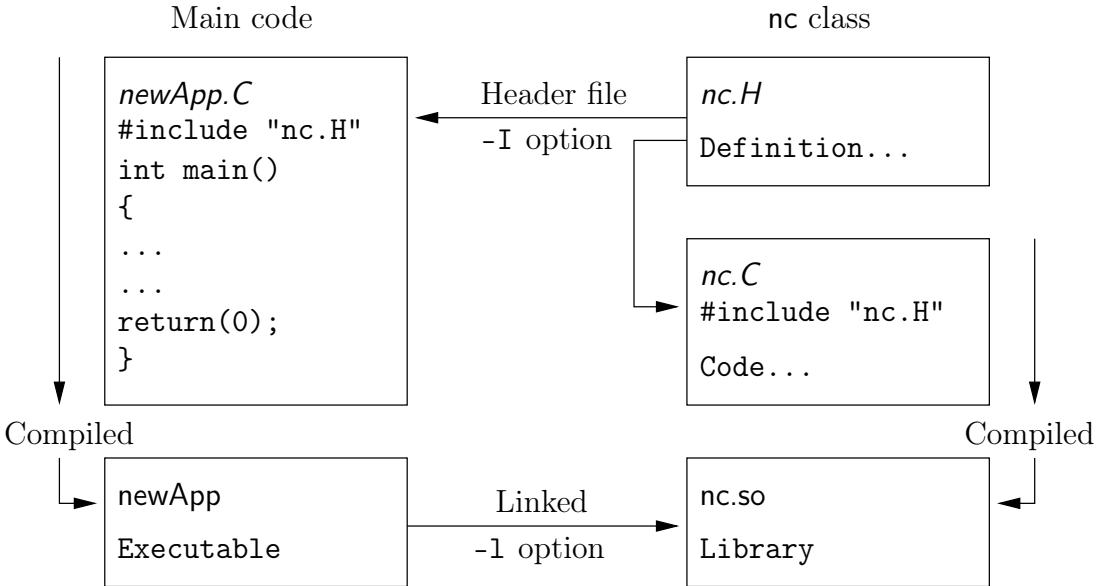


Figure 3.1: Header files, source files, compilation and linking

begin by including all the *.H* files required to declare these classes. Those classes in turn can resource other classes and so also begin by including the relevant *.H* files. By searching recursively down the class hierarchy we can produce a complete list of header files for all the classes on which the top level *.C* code ultimately depends; these *.H* files are known as the *dependencies*. With a dependency list, a compiler can check whether the source files have been updated since their last compilation and selectively compile only those that need to be.

Header files are included in the code using the `# include` directive, *e.g.*

```
# include "otherHeader.H";
```

This causes the compiler to suspend reading from the current file, to read the included file. This mechanism allows any self-contained piece of code to be put into a header file and included at the relevant location in the main code in order to improve code readability. For example, in most OpenFOAM applications the code for creating fields and reading field input data is included in a file *createFields.H* which is called at the beginning of the code. In this way, header files are not solely used as class declarations.

It is `wmake` that performs the task of maintaining file dependency lists amongst other functions listed below.

- Automatic generation and maintenance of file dependency lists, *i.e.* lists of files which are included in the source files and hence on which they depend.
- Multi-platform compilation and linkage, handled through appropriate directory structure.
- Multi-language compilation and linkage, *e.g.* C, C++, Java.
- Multi-option compilation and linkage, *e.g.* debug, optimised, parallel and profiling.
- Support for source code generation programs, *e.g.* lex, yacc, IDL, MOC.

- Simple syntax for source file lists.
- Automatic creation of source file lists for new codes.
- Simple handling of multiple shared or static libraries.
- Extensible to new machine types.
- Extremely portable, works on any machine with: `make`; `sh`, `ksh` or `csh`; `lex`, `cc`.

3.2.2 Compiling with wmake

OpenFOAM applications are organised using a standard convention that the source code of each application is placed in a directory whose name is that of the application. The top level source file then takes the application name with the `.C` extension. For example, the source code for an application called `newApp` would reside in a directory `newApp` and the top level file would be `newApp.C` as shown in Figure 3.2. `wmake` then requires the directory

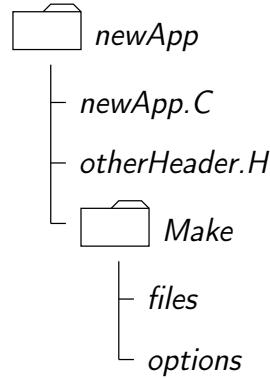


Figure 3.2: Directory structure for an application

must contain a `Make` subdirectory containing 2 files, `options` and `files`, that are described in the following sections.

3.2.2.1 Including headers

The compiler searches for the included header files in the following order, specified with the `-I` option in `wmake`:

1. the `$WM_PROJECT_DIR/src/OpenFOAM/lnInclude` directory;
2. a local `lnInclude` directory, *i.e.* `newApp/lnInclude`;
3. the local directory, *i.e.* `newApp`;
4. platform dependent paths set in files in the `$WM_PROJECT_DIR/wmake/rules/-$WM_ARCH/` directory, *e.g.* `/usr/X11/include` and `$(MPICH_ARCH_PATH)/include`;
5. other directories specified explicitly in the `Make/options` file with the `-I` option.

The `Make/options` file contains the full directory paths to locate header files using the syntax:

```
EXE_INC = \
-I<directoryPath1> \
-I<directoryPath2> \
... \
-I<directoryPathN>
```

Notice first that the directory names are preceded by the `-I` flag and that the syntax uses the `\` to continue the `EXE_INC` across several lines, with no `\` after the final entry.

3.2.2.2 Linking to libraries

The compiler links to shared object library files in the following directory **paths**, specified with the `-L` option in `wmake`:

1. the `$FOAM_LIBBIN` directory;
2. platform dependent paths set in files in the `$WM_DIR/rules/$WM_ARCH/` directory, *e.g.* `/usr/X11/lib` and `$(MPICH_ARCH_PATH)/lib`;
3. other directories specified in the *Make/options* file.

The actual library **files** to be linked must be specified using the `-l` option and removing the `lib` prefix and `.so` extension from the library file name, *e.g.* `libnew.so` is included with the flag `-lnew`. By default, `wmake` loads the following libraries:

1. the `libOpenFOAM.so` library from the `$FOAM_LIBBIN` directory;
2. platform dependent libraries specified in set in files in the `$WM_DIR/rules/$WM_ARCH/` directory, *e.g.* `libm.so` from `/usr/X11/lib` and `liblam.so` from `$(LAM_ARCH_PATH)/lib`;
3. other libraries specified in the *Make/options* file.

The *Make/options* file contains the full directory paths and library names using the syntax:

```
EXE_LIBS = \
-L<libraryPath> \
-l<library1> \
-l<library2> \
... \
-l<libraryN>
```

To summarise: the directory paths are preceded by the `-L` flag, the library names are preceded by the `-l` flag.

3.2.2.3 Source files to be compiled

The compiler requires a list of *.C* source files that must be compiled. The list must contain the main *.C* file but also any other source files that are created for the specific application but are not included in a class library. For example, users may create a new class or some new functionality to an existing class for a particular application. The full list of *.C* source files must be included in the *Make/files* file. For many applications the list only includes the name of the main *.C* file, *e.g.* *newApp.C* in the case of our earlier example.

The *Make/files* file also includes a full path and name of the compiled executable, specified by the *EXE =* syntax. Standard convention stipulates the name is that of the application, *i.e.* *newApp* in our example. The OpenFOAM release offers two useful choices for path: standard release applications are stored in *\$FOAM_APPBIN*; applications developed by the user are stored in *\$FOAM_USER_APPBIN*.

If the user is developing their own applications, we recommend they create an *applications* subdirectory in their *\$WM_PROJECT_USER_DIR* directory containing the source code for personal OpenFOAM applications. As with standard applications, the source code for each OpenFOAM application should be stored within its own directory. The only difference between a user application and one from the standard release is that the *Make/files* file should specify that the user's executables are written into their *\$FOAM_USER_APPBIN* directory. The *Make/files* file for our example would appear as follows:

```
newApp.C
EXE = $(FOAM_USER_APPBIN)/newApp
```

3.2.2.4 Running wmake

The *wmake* script is generally executed by typing:

```
wmake <optionalDirectory>
```

The *<optionalDirectory>* is the directory path of the application that is being compiled. Typically, *wmake* is executed from within the directory of the application being compiled, in which case *<optionalDirectory>* can be omitted.

3.2.2.5 wmake environment variables

For information, the environment variable settings used by *wmake* are listed in Table 3.1.

3.2.3 Removing dependency lists: wclean

On execution, *wmake* builds a dependency list file with a *.dep* file extension, *e.g.* *newApp.C.dep* in our example, in a *\$WM_OPTIONS* sub-directory of the *Make* directory, *e.g.* *Make/linux-GccDPInt64Opt*. If the user wishes to remove these files, *e.g.* after making code changes, the user can run the *wclean* script by typing:

```
wclean <optionalDirectory>
```

Main paths	
\$WM_PROJECT_INST_DIR	Full path to installation directory, <i>e.g. \$HOME/OpenFOAM</i>
\$WM_PROJECT	Name of the project being compiled: OpenFOAM
\$WM_PROJECT_VERSION	Version of the project being compiled: 5.0
\$WM_PROJECT_DIR	Full path to locate binary executables of OpenFOAM release, <i>e.g. \$HOME/OpenFOAM/OpenFOAM-5.0</i>
\$WM_PROJECT_USER_DIR	Full path to locate binary executables of the user <i>e.g. \$HOME/OpenFOAM/\${USER}-5.0</i>
\$WM_THIRD_PARTY_DIR	Full path to the <i>ThirdParty</i> software directory <i>e.g. \$HOME/OpenFOAM/ThirdParty-5.0</i>
Other paths/settings	
\$WM_ARCH	Machine architecture: linux, linux64, linuxIa64, linuxARM7, linuxPPC64, linuxPPC64le
\$WM_ARCH_OPTION	32 or 64 bit architecture
\$WM_COMPILER	Compiler being used: Gcc - gcc, ICC - Intel, Clang - LLVM Clang
\$WM_COMPILE_OPTION	Compilation option: Debug - debugging, Opt optimisation.
\$WM_COMPILER_TYPE	Choice of compiler: system, ThirdParty - compiled in <i>ThirdParty</i> directory
\$WM_DIR	Full path of the <i>wmake</i> directory
\$WM_LABEL_SIZE	32 or 64 bit size for labels (integers)
\$WM_LABEL_OPTION	Int32 or Int64 compilation of labels
\$WM_LINK_LANGUAGE	Compiler used to link libraries and executables c++.
\$WM_MPLIB	Parallel communications library: SYSTEMOPENMPI - system version of openMPI, OPENMPI, SYSTEMMPI, MPICH, MPICH-GM, HPMPI, MPI, QSMPI, SGIMPI.
\$WM_OPTIONS	= \$WM_ARCH... \$WM_COMPILER... \$WM_PRECISION_OPTION... \$WM_LABEL_OPTION... \$WM_COMPILE_OPTION <i>e.g. linuxGccDPInt64Opt</i>
\$WM_PRECISION_OPTION	Precision of the compiled binaries, SP, single precision or DP, double precision

Table 3.1: Environment variable settings for *wmake*.

Again, the <optionalDirectory> is a path to the directory of the application that is being compiled. Typically, `wclean` is executed from within the directory of the application, in which case the path can be omitted.

3.2.4 Compiling libraries

When compiling a library, there are 2 critical differences in the configuration of the file in the `Make` directory:

- in the *files* file, `EXE =` is replaced by `LIB =` and the target directory for the compiled entity changes from `$FOAM_APPBIN` to `$FOAM_LIBBIN` (and an equivalent `$FOAM_USER_LIBBIN` directory);
- in the *options* file, `EXE_LIBS =` is replaced by `LIB_LIBS =` to indicate libraries linked to library being compiled.

When `wmake` is executed it additionally creates a directory named `InInclude` that contains soft links to all the files in the library. The `InInclude` directory is deleted by the `wclean` script when cleaning library source code.

3.2.5 Compilation example: the pisoFoam application

The source code for application `pisoFoam` is in the `$FOAM_APP/solvers/incompressible/pisoFoam` directory and the top level source file is named `pisoFoam.C`. The `pisoFoam.C` source code is:

```

1  /*-----*\
2  =====
3  \  /  F ield           | OpenFOAM: The Open Source CFD Toolbox
4  \  /  O peration        |
5  \  /  A nd              | Copyright (C) 2011-2016 OpenFOAM Foundation
6  \  /  M anipulation     |
7  -----
8  License
9   This file is part of OpenFOAM.
10
11  OpenFOAM is free software: you can redistribute it and/or modify it
12  under the terms of the GNU General Public License as published by
13  the Free Software Foundation, either version 3 of the License, or
14  (at your option) any later version.
15
16  OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
17  ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
18  FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
19  for more details.
20
21  You should have received a copy of the GNU General Public License
22  along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>.
23
24 Application
25   pisoFoam
26
27 Description
28   Transient solver for incompressible, turbulent flow, using the PISO
29   algorithm.
30
31 Sub-models include:
32   - turbulence modelling, i.e. laminar, RAS or LES
33   - run-time selectable MRF and finite volume options, e.g. explicit porosity
34
35 */
36
37 #include "fvCFD.H"
38 #include "singlePhaseTransportModel.H"
39 #include "turbulentTransportModel.H"
```

```

40 #include "pisoControl.H"
41 #include "fvOptions.H"
42 // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
43 int main(int argc, char *argv[])
44 {
45     #include "postProcess.H"
46
47     #include "setRootCase.H"
48     #include "createTime.H"
49     #include "createMesh.H"
50     #include "createControl.H"
51     #include "createFields.H"
52     #include "createFvOptions.H"
53     #include "initContinuityErrs.H"
54
55     turbulence->validate();
56
57     // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
58     Info<< "\nStarting time loop\n" << endl;
59
60     while (runTime.loop())
61     {
62         Info<< "Time = " << runTime.timeName() << nl << endl;
63
64         #include "CourantNo.H"
65
66         // Pressure-velocity PISO corrector
67         {
68             #include "UEqn.H"
69
70             // --- PISO loop
71             while (piso.correct())
72             {
73                 #include "pEqn.H"
74             }
75
76             laminarTransport.correct();
77             turbulence->correct();
78
79             runTime.write();
80
81             Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s"
82             << " ClockTime = " << runTime.elapsedClockTime() << " s"
83             << nl << endl;
84         }
85
86         Info<< "End\n" << endl;
87
88         return 0;
89     }
90
91
92 } // ****

```

The code begins with a brief description of the application contained within comments over 1 line (//) and multiple lines /*...*/. Following that, the code contains several `# include` statements, e.g. `# include "fvCFD.H"`, which causes the compiler to suspend reading from the current file, *pisoFoam.C* to read the *fvCFD.H*.

pisoFoam resources the turbulence and transport model libraries and therefore requires the necessary header files, specified by the `EXE_INC = -I...` option, and links to the libraries with the `EXE_LIBS = -l...` option. The *Make/options* therefore contains the following:

```

1 EXE_INC = \
2   -I$(LIB_SRC)/TurbulenceModels/turbulenceModels/lnInclude \
3   -I$(LIB_SRC)/TurbulenceModels/incompressible/lnInclude \
4   -I$(LIB_SRC)/transportModels \
5   -I$(LIB_SRC)/transportModels/incompressible/singlePhaseTransportModel \
6   -I$(LIB_SRC)/finiteVolume/lnInclude \

```

```

7   -I$(LIB_SRC)/meshTools/lnInclude \
8   -I$(LIB_SRC)/sampling/lnInclude \
9
10  EXE_LIBS = \
11    -lturbulenceModels \
12    -lincompressibleTurbulenceModels \
13    -lincompressibleTransportModels \
14    -lfiniteVolume \
15    -lmeshTools \
16    -lfvOptions \
17    -lsampling

```

pisoFoam contains only the *pisoFoam.C* source and the executable is written to the *\$FOAM_APPBIN* directory as all standard applications are. The *Make/files* therefore contains:

```

1  pisoFoam.C
2
3  EXE = $(FOAM_APPBIN)/pisoFoam

```

Following the recommendations of section 3.2.2.3, the user can compile a separate version of pisoFoam into their local *\$FOAM_USER_DIR* directory by the following:

- copying the pisoFoam source code to a local directory, e.g. *\$FOAM_RUN*;

```

cd $FOAM_RUN
cp -r $FOAM_SOLVERS/incompressible/pisoFoam .
cd pisoFoam

```

- editing the *Make/files* file as follows;

```

1  pisoFoam.C
2
3  EXE = $(FOAM_USER_APPBIN)/pisoFoam

```

- executing wmake.

```
wmake
```

The code should compile and produce a message similar to the following

```

Making dependency list for source file pisoFoam.C
g++ -std=c++0x -m32...
...
-o ... platforms/linuxGccDPInt64Opt/bin/pisoFoam

```

The user can now try recompiling and will receive a message similar to the following to say that the executable is up to date and compiling is not necessary:

```
make: `.../bin/pisoFoam` is up to date.
```

The user can compile the application from scratch by removing the dependency list with

```
wclean
```

and running wmake.

3.2.6 Debug messaging and optimisation switches

OpenFOAM provides a system of messaging that is written during runtime, most of which are to help debugging problems encountered during running of a OpenFOAM case. The switches are listed in the `$WM_PROJECT_DIR/etc/controlDict` file; should the user wish to change the settings they should make a copy to their `$HOME` directory, *i.e.* `$HOME/.OpenFOAM/5.0/controlDict` file. The list of possible switches is extensive and can be viewed by running the `foamDebugSwitches` application. Most of the switches correspond to a class or range of functionality and can be switched on by their inclusion in the `controlDict` file, and by being set to 1. For example, OpenFOAM can perform the checking of dimensional units in all calculations by setting the `dimensionSet` switch to 1. There are some switches that control messaging at a higher level than most, listed in Table 3.2.

In addition, there are some switches that control certain operational and optimisation issues. These switches are also listed in Table 3.2. Of particular importance is `fileModificationSkew`. OpenFOAM scans the write time of data files to check for modification. When running over a NFS with some disparity in the clock settings on different machines, field data files appear to be modified ahead of time. This can cause a problem if OpenFOAM views the files as newly modified and attempting to re-read this data. The `fileModificationSkew` keyword is the time in seconds that OpenFOAM will subtract from the file write time when assessing whether the file has been newly modified.

High level debugging switches - sub-dictionary `DebugSwitches`

<code>level</code>	Overall level of debugging messaging for OpenFOAM- - 3 levels 0, 1, 2
<code>lduMatrix</code>	Messaging for solver convergence during a run - 3 levels 0, 1, 2

Optimisation switches - sub-dictionary `OptimisationSwitches`

<code>fileModificationSkew</code>	A time in seconds that should be set higher than the maximum delay in NFS updates and clock difference for running OpenFOAM over a NFS.
<code>fileModificationChecking</code>	Method of checking whether files have been modified during a simulation, either reading the <code>timeStamp</code> or using <code>inotify</code> ; versions that read only master-node data exist, <code>timeStampMaster</code> , <code>inotifyMaster</code> .
<code>commsType</code>	Parallel communications type: <code>nonBlocking</code> , <code>scheduled</code> , <code>blocking</code> .
<code>floatTransfer</code>	If 1, will compact numbers to <code>float</code> precision before transfer; default is 0
<code>nProcsSimpleSum</code>	Optimises global sum for parallel processing; sets number of processors above which hierarchical sum is performed rather than a linear sum (default 16)

Table 3.2: Runtime message switches.

3.2.7 Linking new user-defined libraries to existing applications

The situation may arise that a user creates a new library, say `new`, and wishes the features within that library to be available across a range of applications. For example, the user may create a new boundary condition, compiled into `new`, that would need to be recognised by a range of solver applications, pre- and post-processing utilities, mesh tools, *etc.* Under normal circumstances, the user would need to recompile every application with the `new` linked to it.

Instead there is a simple mechanism to link one or more shared object libraries dynamically at run-time in OpenFOAM. Simply add the optional keyword entry `libs` to the `controlDict` file for a case and enter the full names of the libraries within a list (as quoted string entries). For example, if a user wished to link the libraries `new1` and `new2` at run-time, they would simply need to add the following to the case `controlDict` file:

```
libs
(
    "libnew1.so"
    "libnew2.so"
);
```

3.3 Running applications

Each application is designed to be executed from a terminal command line, typically reading and writing a set of data files associated with a particular case. The data files for a case are stored in a directory named after the case as described in section 4.1; the directory name with full path is here given the generic name `<caseDir>`.

For any application, the form of the command line entry for any can be found by simply entering the application name at the command line with the `-help` option, *e.g.* typing

```
blockMesh -help
```

returns the usage

```
Usage: blockMesh [OPTIONS]
options:
    -blockTopology      write block edges and centres as .obj files
    -case <dir>         specify alternate case directory, default is the
                        cwd
    -dict <file>        specify alternative dictionary for the blockMesh
                        description
    -noFunctionObjects do not execute functionObjects
    -region <name>      specify alternative mesh region
    -srcDoc              display source code in browser
    -doc                 display application documentation in browser
    -help                print the usage
```

If the application is executed from within a case directory, it will operate on that case. Alternatively, the `-case <caseDir>` option allows the case to be specified directly so that the application can be executed from anywhere in the filing system.

Like any UNIX/Linux executable, applications can be run as a background process, *i.e.* one which does not have to be completed before the user can give the shell additional commands. If the user wished to run the `blockMesh` example as a background process and output the case progress to a *log* file, they could enter:

```
blockMesh > log &
```

3.4 Running applications in parallel

This section describes how to run OpenFOAM in parallel on distributed processors. The method of parallel computing used by OpenFOAM is known as domain decomposition, in which the geometry and associated fields are broken into pieces and allocated to separate processors for solution. The process of parallel computation involves: decomposition of mesh and fields; running the application in parallel; and, post-processing the decomposed case as described in the following sections. The parallel running uses the public domain `openMPI` implementation of the standard message passing interface (MPI) by default, although other libraries can be used.

3.4.1 Decomposition of mesh and initial field data

The mesh and fields are decomposed using the `decomposePar` utility. The underlying aim is to break up the domain with minimal effort but in such a way to guarantee an economic solution. The geometry and fields are broken up according to a set of parameters specified in a dictionary named `decomposeParDict` that must be located in the `system` directory of the case of interest. An example `decomposeParDict` dictionary is available from the `interFoam/damBreak` tutorial if the user requires one; the dictionary entries within it are reproduced below:

```
17   numberOfSubdomains 4;
18
19   method           simple;
20
21   simpleCoeffs
22   {
23     n              (2 2 1);
24     delta          0.001;
25   }
26
27   hierarchicalCoeffs
28   {
29     n              (1 1 1);
30     delta          0.001;
31     order          xyz;
32   }
33
34   manualCoeffs
35   {
36     dataFile        "";
37   }
38
39   distributed      no;
40
41   roots            ( );
42
43 // **** //
```

The user has a choice of four methods of decomposition, specified by the `method` keyword as described below.

simple Simple geometric decomposition in which the domain is split into pieces by direction, *e.g.* 2 pieces in the x direction, 1 in y *etc.*

hierarchical Hierarchical geometric decomposition which is the same as `simple` except the user specifies the order in which the directional split is done, *e.g.* first in the y -direction, then the x -direction *etc.*

scotch Scotch decomposition which requires no geometric input from the user and attempts to minimise the number of processor boundaries. The user can specify a weighting for the decomposition between processors, through an optional `processorWeights` keyword which can be useful on machines with differing performance between processors. There is also an optional keyword entry `strategy` that controls the decomposition strategy through a complex string supplied to Scotch. For more information, see the source code file: `$FOAM_SRC/parallel/decompose/scotchDecomp/scotchDecomp.C`

manual Manual decomposition, where the user directly specifies the allocation of each cell to a particular processor.

For each `method` there are a set of coefficients specified in a sub-dictionary of `decompositionDict`, named `<method>Coeffs` as shown in the dictionary listing. The full set of keyword entries in the `decomposeParDict` dictionary are explained in Table 3.3.

The `decomposePar` utility is executed in the normal manner by typing

```
decomposePar
```

3.4.2 File input/output in parallel

Using standard file input/output completion, a set of subdirectories will have been created, one for each processor, in the case directory. The directories are named `processorN` where $N = 0, 1, \dots$ represents a processor number and contains a time directory, containing the decomposed field descriptions, and a `constant/polyMesh` directory containing the decomposed mesh description.

While this file structure is well-organised, for large parallel cases, it generates a large number of files. In very large simulations, users can experience problems including hitting limits on the number of open files imposed by the operating system.

As an alternative, the **collated** file format was introduced in OpenFOAM in which the data for each decomposed field (and mesh) is collated into a single file that is written (and read) on the master processor. The files are stored in a single directory named `processors`.

The file writing can be threaded allowing the simulation to continue running while the data is being written to file — see below for details. NFS (Network File System) is not needed when using the collated format and, additionally, there is a `masterUncollated` option to write data with the original `uncollated` format without NFS.

The controls for the file handling are in the `OptimisationSwitches` of the global `etc/controlDict` file:

Compulsory entries

<code>numberOfSubdomains</code>	Total number of subdomains	N
<code>method</code>	Method of decomposition	<code>simple/</code> <code>hierarchical/</code> <code>scotch/ manual</code> /

simpleCoeffs entries

<code>n</code>	Number of subdomains in x, y, z	$(n_x \ n_y \ n_z)$
<code>delta</code>	Cell skew factor	Typically, 10^{-3}

hierarchicalCoeffs entries

<code>n</code>	Number of subdomains in x, y, z	$(n_x \ n_y \ n_z)$
<code>delta</code>	Cell skew factor	Typically, 10^{-3}
<code>order</code>	Order of decomposition	<code>xyz/xzy/yxz...</code>

scotchCoeffs entries

<code>processorWeights</code> (optional)	List of weighting factors for allocation of cells to processors; <code><wt1></code> is the weighting factor for processor 1, <i>etc.</i> ; weights are normalised so can take any range of values.	<code>(<wt1>...<wtN>)</code>
<code>strategy</code>	Decomposition strategy: optional and complex	

manualCoeffs entries

<code>dataFile</code>	Name of file containing data of alloca- tion of cells to processors	<code>"<fileName>"</code>
-----------------------	--	---------------------------------

Distributed data entries (optional) — see section 3.4.4

<code>distributed</code>	Is the data distributed across several disks?	<code>yes/no</code>
<code>roots</code>	Root paths to case directories; <code><rt1></code> (<code><rt1>...<rtN></code>) is the root path for node 1, <i>etc.</i>	

Table 3.3: Keywords in *decompositionDict* dictionary.

```

OptimisationSwitches
{
    ...
    //-
    // Parallel IO file handler
    // uncollated (default), collated or masterUncollated
    fileHandler uncollated;

    //-
    // collated: thread buffer size for queued file writes.
    // If set to 0 or not sufficient for the file size threading is not used.
    // Default: 2e9
    maxThreadFileBufferSize 2e9;

    //-
    // masterUncollated: non-blocking buffer size.
    // If the file exceeds this buffer size scheduled transfer is used.
    // Default: 2e9
}

```

```

    maxMasterFileBufferSize 2e9;
}

```

3.4.2.1 Selecting the file handler

The `fileHandler` can be set for a specific simulation by:

- over-riding the global `OptimisationSwitches {fileHandler ...;}` in the case `controlDict` file;
- using the `-fileHandler` command line argument to the solver;
- setting the `$FOAM_FILEHANDLER` environment variable.

3.4.2.2 Updating existing files

A `foamFormatConvert` utility allows users to convert files between the collated and uncollated formats, e.g.

```
mpirun -np 2 foamFormatConvert -parallel -fileHandler uncollated
```

An example case demonstrating the file handling methods is provided in:
`$FOAM_TUTORIALS/IO/fileHandling`

3.4.2.3 Threading support

Collated file handling runs faster with threading, especially on large cases. But it requires threading support to be enabled in the underlying MPI. Without it, the simulation will “hang” or crash. For `openMPI`, threading support is not set by default prior to version 2, but is generally switched on from version 2 onwards. The user can check whether `openMPI` is compiled with threading support by the following command:

```
ompi_info -c | grep -oE "MPI_THREAD_MULTIPLE[^,]*"
```

When using the collated file handling, memory is allocated for the data in the thread. `maxThreadFileSize` sets the maximum size of memory that is allocated in bytes. If the data exceeds this size, the write does not use threading.

Note: if threading is **not enabled** in the MPI, it must be disabled for collated file handling by setting in the global `etc/controlDict` file:

```
maxThreadFileSize    0;
```

When using the `masterUncollated` file handling, non-blocking MPI communication requires a sufficiently large memory buffer on the master node. `maxMasterFileSize` sets the maximum size of the buffer. If the data exceeds this size, the system uses scheduled communication.

3.4.3 Running a decomposed case

A decomposed OpenFOAM case is run in parallel using the `openMPI` implementation of MPI.

`openMPI` can be run on a local multiprocessor machine very simply but when running on machines across a network, a file must be created that contains the host names of the machines. The file can be given any name and located at any path. In the following description we shall refer to such a file by the generic name, including full path, `<machines>`.

The `<machines>` file contains the names of the machines listed one machine per line. The names must correspond to a fully resolved hostname in the `/etc/hosts` file of the machine on which the `openMPI` is run. The list must contain the name of the machine running the `openMPI`. Where a machine node contains more than one processor, the node name may be followed by the entry `cpu=n` where `n` is the number of processors `openMPI` should run on that node.

For example, let us imagine a user wishes to run `openMPI` from machine `aaa` on the following machines: `aaa`; `bbb`, which has 2 processors; and `ccc`. The `<machines>` would contain:

```
aaa
bbb cpu=2
ccc
```

An application is run in parallel using `mpirun`.

```
mpirun --hostfile <machines> -np <nProcs>
<foamExec> <otherArgs> -parallel > log &
```

where: `<nProcs>` is the number of processors; `<foamExec>` is the executable, e.g.`icoFoam`; and, the output is redirected to a file named `log`. For example, if `icoFoam` is run on 4 nodes, specified in a file named `machines`, on the `cavity` tutorial in the `$FOAM_RUN/tutorials/-incompressible/icoFoam` directory, then the following command should be executed:

```
mpirun --hostfile machines -np 4 icoFoam -parallel > log &
```

3.4.4 Distributing data across several disks

Data files may need to be distributed if, for example, if only local disks are used in order to improve performance. In this case, the user may find that the root path to the case directory may differ between machines. The paths must then be specified in the `decomposeParDict` dictionary using `distributed` and `roots` keywords. The `distributed` entry should read

```
distributed yes;
```

and the `roots` entry is a list of root paths, `<root0>`, `<root1>`, ..., for each node

```
roots
<nRoots>
()
```

```

"<root0>"  

"<root1>"  

...  

);

```

where `<nRoots>` is the number of roots.

Each of the `processorN` directories should be placed in the case directory at each of the root paths specified in the `decomposeParDict` dictionary. The `system` directory and `files` within the `constant` directory must also be present in each case directory. Note: the files in the `constant` directory are needed, but the `polyMesh` directory is not.

3.4.5 Post-processing parallel processed cases

When post-processing cases that have been run in parallel the user has two options:

- reconstruction of the mesh and field data to recreate the complete domain and fields, which can be post-processed as normal;
- post-processing each segment of decomposed domain individually.

3.4.5.1 Reconstructing mesh and data

After a case has been run in parallel, it can be reconstructed for post-processing. The case is reconstructed by merging the sets of time directories from each `processorN` directory into a single set of time directories. The `reconstructPar` utility performs such a reconstruction by executing the command:

```
reconstructPar
```

When the data is distributed across several disks, it must be first copied to the local case directory for reconstruction.

3.4.5.2 Post-processing decomposed cases

The user may post-process decomposed cases using the `paraFoam` post-processor, described in section 6.1. The whole simulation can be post-processed by reconstructing the case or alternatively it is possible to post-process a segment of the decomposed domain individually by simply treating the individual processor directory as a case in its own right.

3.5 Standard solvers

The solvers with the OpenFOAM distribution are in the `$FOAM_SOLVERS` directory, reached quickly by typing `sol` at the command line. This directory is further subdivided into several directories by category of continuum mechanics, *e.g.* incompressible flow, combustion and solid body stress analysis. Each solver is given a name that is reasonably descriptive, *e.g.* `icoFoam` solves incompressible, laminar flow. The current list of solvers distributed with OpenFOAM is given in the following Sections.

3.5.1 ‘Basic’ CFD codes

laplacianFoam Solves a simple Laplace equation, e.g. for thermal diffusion in a solid.

potentialFoam Potential flow solver which solves for the velocity potential, to calculate the flux-field, from which the velocity field is obtained by reconstructing the flux.

scalarTransportFoam Solves the steady or transient transport equation for a passive scalar.

3.5.2 Incompressible flow

adjointShapeOptimizationFoam Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids with optimisation of duct shape by applying "blockage" in regions causing pressure loss as estimated using an adjoint formulation.

boundaryFoam Steady-state solver for incompressible, 1D turbulent flow, typically to generate boundary layer conditions at an inlet, for use in a simulation.

icoFoam Transient solver for incompressible, laminar flow of Newtonian fluids.

nonNewtonianIcoFoam Transient solver for incompressible, laminar flow of non-Newtonian fluids.

pimpleFoam Large time-step transient solver for incompressible, turbulent flow, using the PIMPLE (merged PISO-SIMPLE) algorithm.

pimpleDyMFoam Transient solver for incompressible, turbulent flow of Newtonian fluids on a moving mesh.

SRFPimpleFoam Large time-step transient solver for incompressible, turbulent flow in a single rotating frame.

pisoFoam Transient solver for incompressible, turbulent flow, using the PISO algorithm.

shallowWaterFoam Transient solver for inviscid shallow-water equations with rotation.

simpleFoam Steady-state solver for incompressible, turbulent flow, using the SIMPLE algorithm.

porousSimpleFoam Steady-state solver for incompressible, turbulent flow with implicit or explicit porosity treatment and support for multiple reference frames (MRF).

SRFSimpleFoam Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids in a single rotating frame.

3.5.3 Compressible flow

rhoCentralFoam Density-based compressible flow solver based on central-upwind schemes of Kurganov and Tadmor.

rhoCentralDyMFoam Density-based compressible flow solver based on central-upwind schemes of Kurganov and Tadmor with support for mesh-motion and topology changes.

rhoPimpleFoam Transient solver for turbulent flow of compressible fluids for HVAC and similar applications.

rhoPimpleDyMFoam Transient solver for turbulent flow of compressible fluids for HVAC and similar applications, with optional mesh motion and mesh topology changes.

rhoSimpleFoam Steady-state solver for turbulent flow of compressible fluids.

rhoPorousSimpleFoam Steady-state solver for turbulent flow of compressible fluids, with implicit or explicit porosity treatment and optional sources.

sonicFoam Transient solver for trans-sonic/supersonic, turbulent flow of a compressible gas.

sonicDyMFoam Transient solver for trans-sonic/supersonic, turbulent flow of a compressible gas, with optional mesh motion and mesh topology changes.

sonicLiquidFoam Transient solver for trans-sonic/supersonic, laminar flow of a compressible liquid.

3.5.4 Multiphase flow

cavitatingFoam Transient cavitation code based on the homogeneous equilibrium model from which the compressibility of the liquid/vapour "mixture" is obtained.

cavitatingDyMFoam Transient cavitation code based on the homogeneous equilibrium model from which the compressibility of the liquid/vapour "mixture" is obtained, with optional mesh motion and mesh topology changes.

compressibleInterFoam Solver for 2 compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach.

compressibleInterDyMFoam Solver for 2 compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

compressibleMultiphaselnterFoam Solver for n compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach.

driftFluxFoam Solver for 2 incompressible fluids using the mixture approach with the drift-flux approximation for relative motion of the phases.

interFoam Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach.

interDyMFoam Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

interMixingFoam Solver for 3 incompressible fluids, two of which are miscible, using a VOF method to capture the interface.

interPhaseChangeFoam Solver for 2 incompressible, isothermal immiscible fluids with phase-change (e.g. cavitation). Uses a VOF (volume of fluid) phase-fraction based interface capturing approach.

interPhaseChangeDyMFoam Solver for 2 incompressible, isothermal immiscible fluids with phase-change (e.g. cavitation). Uses a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

multiphaseEulerFoam Solver for a system of many compressible fluid phases including heat-transfer.

multiphaselInterFoam Solver for n incompressible fluids which captures the interfaces and includes surface-tension and contact-angle effects for each phase.

multiphaselInterDyMFoam Solver for n incompressible fluids which captures the interfaces and includes surface-tension and contact-angle effects for each phase, with optional mesh motion and mesh topology changes.

potentialFreeSurfaceFoam Incompressible Navier-Stokes solver with inclusion of a wave height field to enable single-phase free-surface approximations

potentialFreeSurfaceDyMFoam Incompressible Navier-Stokes solver with inclusion of a wave height field to enable single-phase free-surface approximations, with optional mesh motion and mesh topology changes.

reactingMultiphaseEulerFoam Solver for a system of any number of compressible fluid phases with a common pressure, but otherwise separate properties. The type of phase model is run time selectable and can optionally represent multiple species and in-phase reactions. The phase system is also run time selectable and can optionally represent different types of momentum, heat and mass transfer.

reactingTwoPhaseEulerFoam Solver for a system of 2 compressible fluid phases with a common pressure, but otherwise separate properties. The type of phase model is run time selectable and can optionally represent multiple species and in-phase reactions. The phase system is also run time selectable and can optionally represent different types of momentum, heat and mass transfer.

twoLiquidMixingFoam Solver for mixing 2 incompressible fluids.

twoPhaseEulerFoam Solver for a system of 2 compressible fluid phases with one phase dispersed, e.g. gas bubbles in a liquid including heat-transfer.

3.5.5 Direct numerical simulation (DNS)

dnsFoam Direct numerical simulation solver for boxes of isotropic turbulence.

3.5.6 Combustion

chemFoam Solver for chemistry problems, designed for use on single cell cases to provide comparison against other chemistry solvers, that uses a single cell mesh, and fields created from the initial conditions.

coldEngineFoam Solver for cold-flow in internal combustion engines.

engineFoam Solver for internal combustion engines.

fireFoam Transient solver for fires and turbulent diffusion flames with reacting particle clouds, surface film and pyrolysis modelling.

PDRFoam Solver for compressible premixed/partially-premixed combustion with turbulence modelling.

reactingFoam Solver for combustion with chemical reactions.

rhoReactingBuoyantFoam Solver for combustion with chemical reactions using a density based thermodynamics package with enhanced buoyancy treatment.

rhoReactingFoam Solver for combustion with chemical reactions using density based thermodynamics package.

XiFoam Solver for compressible premixed/partially-premixed combustion with turbulence modelling.

3.5.7 Heat transfer and buoyancy-driven flows

buoyantBoussinesqPimpleFoam Transient solver for buoyant, turbulent flow of incompressible fluids.

buoyantBoussinesqSimpleFoam Steady-state solver for buoyant, turbulent flow of incompressible fluids.

buoyantPimpleFoam Transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer.

buoyantSimpleFoam Steady-state solver for buoyant, turbulent flow of compressible fluids, including radiation, for ventilation and heat-transfer.

chtMultiRegionFoam Transient solver for buoyant, turbulent fluid flow and solid heat conduction with conjugate heat transfer between solid and fluid regions.

chtMultiRegionSimpleFoam Steady-state solver for buoyant, turbulent fluid flow and solid heat conduction with conjugate heat transfer between solid and fluid regions.

thermoFoam Solver for energy transport and thermodynamics on a frozen flow field.

3.5.8 Particle-tracking flows

coalChemistryFoam Transient solver for compressible, turbulent flow, with coal and limestone particle clouds, an energy source, and combustion.

DPMFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase.

DPMDyMFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase, with optional mesh motion and mesh topology changes.

MPPICFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase. Multi-Phase Particle In Cell (MPPIC) modeling is used to represent collisions without resolving particle-particle interactions.

MPPICDyMFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase. Multi-Phase Particle In Cell (MPPIC) modeling is used to represent collisions without resolving particle-particle interactions, with optional mesh motion and mesh topology changes.

icoUncoupledKinematicParcelFoam Transient solver for the passive transport of a single kinematic particle cloud.

icoUncoupledKinematicParcelDyMFoam Transient solver for the passive transport of a single kinematic particle cloud, with optional mesh motion and mesh topology changes.

reactingParcelFilmFoam Transient solver for compressible, turbulent flow with a reacting, multiphase particle cloud, and surface film modelling.

reactingParcelFoam Transient solver for compressible, turbulent flow with a reacting, multiphase particle cloud, and optional sources/constraints.

simpleReactingParcelFoam Steady state solver for compressible, turbulent flow with reacting, multiphase particle clouds and optional sources/constraints.

sprayFoam Transient solver for compressible, turbulent flow with a spray particle cloud.

sprayDyMFoam Transient solver for compressible, turbulent flow with a spray particle cloud, with optional mesh motion and mesh topology changes.

sprayEngineFoam Transient solver for compressible, turbulent engine flow with a spray particle cloud.

uncoupledKinematicParcelFoam Transient solver for the passive transport of a particle cloud.

uncoupledKinematicParcelDyMFoam Transient solver for the passive transport of a particle cloud, with optional mesh motion and mesh topology changes.

3.5.9 Discrete methods

dsmcFoam Direct simulation Monte Carlo (DSMC) solver for, transient, multi-species flows.

mdEquilibrationFoam Solver to equilibrate and/or precondition molecular dynamics systems.

mdFoam Molecular dynamics solver for fluid dynamics.

3.5.10 Electromagnetics

electrostaticFoam Solver for electrostatics.

magneticFoam Solver for the magnetic field generated by permanent magnets.

mhdFoam Solver for magnetohydrodynamics (MHD): incompressible, laminar flow of a conducting fluid under the influence of a magnetic field.

3.5.11 Stress analysis of solids

solidDisplacementFoam Transient segregated finite-volume solver of linear-elastic, small-strain deformation of a solid body, with optional thermal diffusion and thermal stresses.

solidEquilibriumDisplacementFoam Steady-state segregated finite-volume solver of linear-elastic, small-strain deformation of a solid body, with optional thermal diffusion and thermal stresses.

3.5.12 Finance

financialFoam Solves the Black-Scholes equation to price commodities.

3.6 Standard utilities

The utilities with the OpenFOAM distribution are in the **\$FOAM_UTILITIES** directory. The names are reasonably descriptive, *e.g.* **ideasToFoam** converts mesh data from the format written by I-DEAS to the OpenFOAM format. The descriptions of current utilities distributed with OpenFOAM are given in the following Sections.

3.6.1 Pre-processing

applyBoundaryLayer Apply a simplified boundary-layer model to the velocity and turbulence fields based on the 1/7th power-law.

boxTurb Makes a box of turbulence which conforms to a given energy spectrum and is divergence free.

changeDictionary Utility to change dictionary entries, *e.g.* can be used to change the patch type in the field and **polyMesh/boundary** files.

createExternalCoupledPatchGeometry Application to generate the patch geometry (points and faces) for use with the **externalCoupled** boundary condition.

dsmcInitialise Initialise a case for **dsmcFoam** by reading the initialisation dictionary system/-
dsmcInitialise.

engineSwirl Generates a swirling flow for engine calulations.

foamUpgradeCyclics Tool to upgrade mesh and fields for split cyclics.

mapFields Maps volume fields from one mesh to another, reading and interpolating all fields present in the time directory of both cases. Parallel and non-parallel cases are handled without the need to reconstruct them first.

mapFieldsPar Maps volume fields from one mesh to another, reading and interpolating all fields present in the time directory of both cases.

mdInitialise Initialises fields for a molecular dynamics (MD) simulation.

setFields Set values on a selected set of cells/patchfaces through a dictionary.

viewFactorsGen View factors are calculated based on a face agglomeration array (**finalAgglom** generated by **faceAgglomerate** utility).

wallFunctionTable Generates a table suitable for use by tabulated wall functions.

3.6.2 Mesh generation

blockMesh A multi-block mesh generator.

extrudeMesh Extrude mesh from existing patch (by default outwards facing normals; optional flips faces) or from patch read from file.

extrudeToRegionMesh Extrude **faceZones** (internal or boundary faces) or **faceSets** (boundary faces only) into a separate mesh (as a different region).

foamyHexMesh Conformal Voronoi automatic mesh generator

foamyHexMeshBackgroundMesh Writes out background mesh as constructed by **foamyHexMesh** and constructs **distanceSurface**.

foamyHexMeshSurfaceSimplify Simplifies surfaces by resampling.

foamyQuadMesh Conformal-Voronoi 2D extruding automatic mesher with grid or read initial points and point position relaxation with optional "squarification".

snappyHexMesh Automatic split hex mesher. Refines and snaps to surface.

3.6.3 Mesh conversion

ansysToFoam Converts an ANSYS input mesh file, exported from I-DEAS, to OpenFOAM format.

cfx4ToFoam Converts a CFX 4 mesh to OpenFOAM format.

datToFoam Reads in a **datToFoam** mesh file and outputs a points file. Used in conjunction with **blockMesh**.

fluent3DMeshToFoam Converts a Fluent mesh to OpenFOAM format.

fluentMeshToFoam Converts a Fluent mesh to OpenFOAM format including multiple region and region boundary handling.

foamMeshToFluent Writes out the OpenFOAM mesh in Fluent mesh format.

foamToStarMesh Reads an OpenFOAM mesh and writes a pro-STAR (v4) bnd/cel/vrt format.

foamToSurface Reads an OpenFOAM mesh and writes the boundaries in a surface format.

gambitToFoam Converts a GAMBIT mesh to OpenFOAM format.

gmshToFoam Reads .msh file as written by Gmsh.

ideasUnvToFoam I-Deas unv format mesh conversion.

kivaToFoam Converts a KIVA3v grid to OpenFOAM format.

mshToFoam Converts .msh file generated by the Adventure system.

netgenNeutralToFoam Converts neutral file format as written by Netgen v4.4.

ccm26ToFoam Reads CCM files as written by Prostar/ccm using ccm 2.6 (not 2.4)

plot3dToFoam Plot3d mesh (ascii/formatted format) converter.

sammToFoam Converts a Star-CD (v3) SAMM mesh to OpenFOAM format.

star3ToFoam Converts a Star-CD (v3) pro-STAR mesh into OpenFOAM format.

star4ToFoam Converts a Star-CD (v4) pro-STAR mesh into OpenFOAM format.

tetgenToFoam Converts .ele and .node and .face files, written by tetgen.

vtkUnstructuredToFoam Converts ascii .vtk (legacy format) file generated by vtk/paraview.

writeMeshObj For mesh debugging: writes mesh as three separate OBJ files which can be viewed with e.g. javaview.

3.6.4 Mesh manipulation

attachMesh Attach topologically detached mesh using prescribed mesh modifiers.

autoPatch Divides external faces into patches based on (user supplied) feature angle.

checkMesh Checks validity of a mesh.

createBaffles Makes internal faces into boundary faces. Does not duplicate points, unlike **mergeOrSplitBaffles**.

createPatch Utility to create patches out of selected boundary faces. Faces come either from existing patches or from a **faceSet**.

deformedGeom Deforms a **polyMesh** using a displacement field **U** and a scaling factor supplied as an argument.

flattenMesh Flattens the front and back planes of a 2D cartesian mesh.

insideCells Picks up cells with cell centre 'inside' of surface. Requires surface to be closed and singly connected.

mergeMeshes Merges two meshes.

mergeOrSplitBaffles Detects faces that share points (baffles). Either merge them or duplicate the points.

mirrorMesh Mirrors a mesh around a given plane.

moveDynamicMesh Mesh motion and topological mesh changes utility.

moveEngineMesh Solver for moving meshes for engine calculations.

moveMesh Solver for moving meshes.

objToVTK Read obj line (not surface!) file and convert into vtk.

orientFaceZone Corrects orientation of **faceZone**.

polyDualMesh Calculates the dual of a **polyMesh**. Adheres to all the feature and patch edges.

refineMesh Utility to refine cells in multiple directions.

renumberMesh Renumbers the cell list in order to reduce the bandwidth, reading and renumbering all fields from all the time directories.

rotateMesh Rotates the mesh and fields from the direction n1 to direction n2.

setSet Manipulate a cell/face/point/ set or zone interactively.

setsToZones Add **pointZones/faceZones/cellZones** to the mesh from similar named **pointSets/-faceSets/cellSets**.

singleCellMesh Reads all fields and maps them to a mesh with all internal faces removed (**singleCellFvMesh**) which gets written to region "singleCell".

splitMesh Splits mesh by making internal faces external. Uses **attachDetach**.

splitMeshRegions Splits mesh into multiple regions.

stitchMesh 'Stitches' a mesh.

subsetMesh Selects a section of mesh based on a **cellSet**.

topoSet Operates on **cellSets/faceSets/pointSets** through a dictionary.

transformPoints Transforms the mesh points in the **polyMesh** directory according to the translate, rotate and scale options.

zipUpMesh Reads in a mesh with hanging vertices and zips up the cells to guarantee that all polyhedral cells of valid shape are closed.

3.6.5 Other mesh tools

`autoRefineMesh` Utility to refine cells near to a surface.

`collapseEdges` Collapses short edges and combines edges that are in line.

`combinePatchFaces` Checks for multiple patch faces on same cell and combines them. Multiple patch faces can result from e.g. removal of refined neighbouring cells, leaving 4 exposed faces with same owner.

`modifyMesh` Manipulates mesh elements.

`PDRMesh` Mesh and field preparation utility for PDR type simulations.

`refineHexMesh` Refines a hex mesh by 2x2x2 cell splitting.

`refinementLevel` Tries to figure out what the refinement level is on refined cartesian meshes.
Run BEFORE snapping.

`refineWallLayer` Utility to refine cells next to patches.

`removeFaces` Utility to remove faces (combines cells on both sides).

`selectCells` Select cells in relation to surface.

`splitCells` Utility to split cells with flat faces.

3.6.6 Post-processing

`engineCompRatio` Calculate the geometric compression ratio. Note that if you have valves and/or extra volumes it will not work, since it calculates the volume at BDC and TCD.

`pdfPlot` Generates a graph of a probability distribution function.

`postChannel` Post-processes data from channel flow calculations.

`temporalInterpolate` Interpolate fields between time-steps e.g. for animation.

`noise` Utility to perform noise analysis of pressure data using the `noiseFFT` library.

`postProcess` Execute the set of `functionObjects` specified in the selected dictionary (which defaults to `system/controlDict`) or on the command-line for the selected set of times on the selected set of fields.

`particleTracks` Generates a VTK file of particle tracks for cases that were computed using a tracked-parcel-type cloud.

`steadyParticleTracks` Generates a VTK file of particle tracks for cases that were computed using a steady-state cloud NOTE: case must be re-constructed (if running in parallel) before use

3.6.7 Post-processing data converters

foamDataToFluent Translates OpenFOAM data to Fluent format.

foamToEnsight Translates OpenFOAM data to EnSight format.

foamToEnsightParts Translates OpenFOAM data to Ensight format. An Ensight part is created for each **cellZone** and patch.

foamToGMV Translates foam output to GMV readable files.

foamToTecplot360 Tecplot binary file format writer.

foamToTetDualMesh Converts **polyMesh** results to **tetDualMesh**.

foamToVTK Legacy VTK file format writer.

smapToFoam Translates a STAR-CD SMAP data file into OpenFOAM field format.

3.6.8 Surface mesh (e.g. OBJ/STL) tools

surfaceAdd Add two surfaces. Does geometric merge on points. Does not check for overlapping/intersecting triangles.

surfaceAutoPatch Patches surface according to feature angle. Like **autoPatch**.

surfaceCheck Checks geometric and topological quality of a surface.

surfaceClean - removes baffles - collapses small edges, removing triangles. - converts sliver triangles into split edges by projecting point onto base of triangle.

surfaceCoarsen Surface coarsening using 'bunnylod':

surfaceConvert Converts from one surface mesh format to another.

surfaceFeatureConvert Convert between **edgeMesh** formats.

surfaceFeatureExtract Extracts and writes surface features to file. All but the basic feature extraction is WIP.

surfaceFind Finds nearest face and vertex.

surfaceHookUp Find close open edges and stitches the surface along them

surfacelnertia Calculates the inertia tensor, principal axes and moments of a command line specified **triSurface**. Inertia can either be of the solid body or of a thin shell.

surfaceLambdaMuSmooth Smooths a surface using lambda/mu smoothing.

surfaceMeshConvert Converts between surface formats with optional scaling or transformations (rotate/translate) on a **coordinateSystem**.

surfaceMeshConvertTesting Converts from one surface mesh format to another, but primarily used for testing functionality.

surfaceMeshExport Export from `surfMesh` to various third-party surface formats with optional scaling or transformations (rotate/translate) on a `coordinateSystem`.

surfaceMeshImport Import from various third-party surface formats into `surfMesh` with optional scaling or transformations (rotate/translate) on a `coordinateSystem`.

surfaceMeshInfo Miscellaneous information about surface meshes.

surfaceMeshTriangulate Extracts surface from a `polyMesh`. Depending on output surface format triangulates faces.

surfaceOrient Set normal consistent with respect to a user provided 'outside' point. If the -inside option is used the point is considered inside.

surfacePointMerge Merges points on surface if they are within absolute distance. Since absolute distance use with care!

surfaceRedistributePar (Re)distribution of `triSurface`. Either takes an undecomposed surface or an already decomposed surface and redistributes it so that each processor has all triangles that overlap its mesh.

surfaceRefineRedGreen Refine by splitting all three edges of triangle ('red' refinement). Neighbouring triangles (which are not marked for refinement get split in half ('green' refinement). (R. Verfuerth, "A review of a posteriori error estimation and adaptive mesh refinement techniques", Wiley-Teubner, 1996)

surfaceSplitByPatch Writes regions of `triSurface` to separate files.

surfaceSplitNonManifolds Takes multiply connected surface and tries to split surface at multiply connected edges by duplicating points. Introduces concept of - `borderEdge`. Edge with 4 faces connected to it. - `borderPoint`. Point connected to exactly 2 `borderEdges`. - `borderLine`. Connected list of `borderEdges`.

surfaceSubset A surface analysis tool which sub-sets the `triSurface` to choose only a part of interest. Based on `subsetMesh`.

surfaceToPatch Reads surface and applies surface regioning to a mesh. Uses `boundaryMesh` to do the hard work.

surfaceTransformPoints Transform (scale/rotate) a surface. Like `transformPoints` but for surfaces.

3.6.9 Parallel processing

decomposePar Automatically decomposes a mesh and fields of a case for parallel execution of OpenFOAM.

reconstructPar Reconstructs fields of a case that is decomposed for parallel execution of OpenFOAM.

reconstructParMesh Reconstructs a mesh using geometric information only.

redistributePar Redistributes existing decomposed mesh and fields according to the current settings in the `decomposeParDict` file.

3.6.10 Thermophysical-related utilities

adiabaticFlameT Calculates the adiabatic flame temperature for a given fuel over a range of unburnt temperatures and equivalence ratios.

chemkinToFoam Converts CHEMKINIII thermodynamics and reaction data files into OpenFOAM format.

equilibriumCO Calculates the equilibrium level of carbon monoxide.

equilibriumFlameT Calculates the equilibrium flame temperature for a given fuel and pressure for a range of unburnt gas temperatures and equivalence ratios; the effects of dissociation on O₂, H₂O and CO₂ are included.

mixtureAdiabaticFlameT Calculates the adiabatic flame temperature for a given mixture at a given temperature.

3.6.11 Miscellaneous utilities

foamDictionary Interrogates and manipulates dictionaries.

foamFormatConvert Converts all IOobjects associated with a case into the format specified in the **controlDict**.

foamHelp Top level wrapper utility around foam help utilities

foamListTimes List times using **timeSelector**.

patchSummary Writes fields and boundary condition info for each patch at each requested time instance.

Chapter 4

OpenFOAM cases

This chapter deals with the file structure and organisation of OpenFOAM cases. Normally, a user would assign a name to a case, *e.g.* the tutorial case of flow in a cavity is simply named `cavity`. This name becomes the name of a directory in which all the case files and subdirectories are stored. The case directories themselves can be located anywhere but we recommend they are within a `run` subdirectory of the user's project directory, *i.e.* `$HOME/OpenFOAM/${USER}-5.0` as described at the beginning of chapter 2. One advantage of this is that the `$FOAM_RUN` environment variable is set to `$HOME/OpenFOAM/${USER}-5.0/run` by default; the user can quickly move to that directory by executing a preset alias, `run`, at the command line.

The tutorial cases that accompany the OpenFOAM distribution provide useful examples of the case directory structures. The tutorials are located in the `$FOAM_TUTORIALS` directory, reached quickly by executing the `tut` alias at the command line. Users can view tutorial examples at their leisure while reading this chapter.

4.1 File structure of OpenFOAM cases

The basic directory structure for a OpenFOAM case, that contains the minimum set of files required to run an application, is shown in Figure 4.1 and described as follows:

A *constant* directory that contains a full description of the case mesh in a subdirectory `polyMesh` and files specifying physical properties for the application concerned, *e.g.* `transportProperties`.

A *system* directory for setting parameters associated with the solution procedure itself.

It contains *at least* the following 3 files: `controlDict` where run control parameters are set including start/end time, time step and parameters for data output; `fvSchemes` where discretisation schemes used in the solution may be selected at run-time; and, `fvSolution` where the equation solvers, tolerances and other algorithm controls are set for the run.

The ‘time’ directories containing individual files of data for particular fields, *e.g.* velocity and pressure. The data can be: either, initial values and boundary conditions that the user must specify to define the problem; or, results written to file by OpenFOAM. Note that the OpenFOAM fields must always be initialised, even when the solution does not strictly require it, as in steady-state problems. The name of each time directory

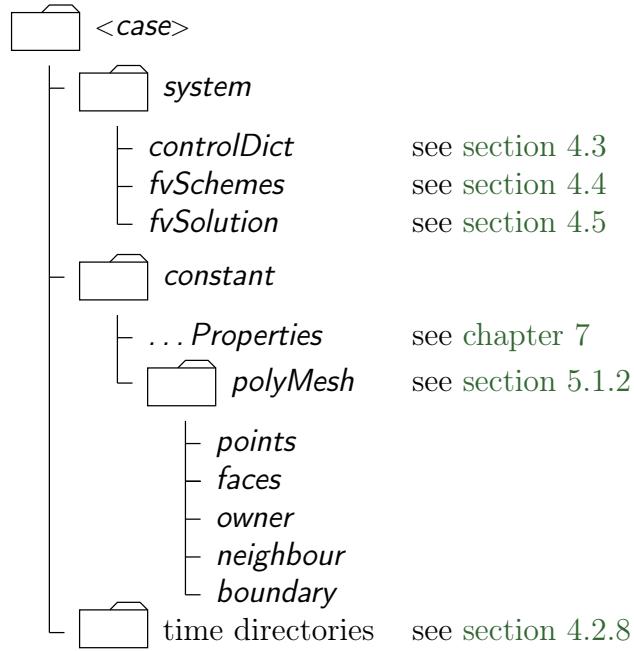


Figure 4.1: Case directory structure

is based on the simulated time at which the data is written and is described fully in section 4.3. It is sufficient to say now that since we usually start our simulations at time $t = 0$, the initial conditions are usually stored in a directory named 0 or $0.000000e+00$, depending on the name format specified. For example, in the `cavity` tutorial, the velocity field \mathbf{U} and pressure field p are initialised from files $0/U$ and $0/p$ respectively.

4.2 Basic input/output file format

OpenFOAM needs to read a range of data structures such as strings, scalars, vectors, tensors, lists and fields. The input/output (I/O) format of files is designed to be extremely flexible to enable the user to modify the I/O in OpenFOAM applications as easily as possible. The I/O follows a simple set of rules that make the files extremely easy to understand, in contrast to many software packages whose file format may not only be difficult to understand intuitively but also not be published. The OpenFOAM file format is described in the following sections.

4.2.1 General syntax rules

The format follows some general principles of C++ source code.

- Files have free form, with no particular meaning assigned to any column and no need to indicate continuation across lines.
- Lines have no particular meaning except to a // comment delimiter which makes OpenFOAM ignore any text that follows it until the end of line.
- A comment over multiple lines is done by enclosing the text between /* and */ delimiters.

4.2.2 Dictionaries

OpenFOAM uses *dictionaries* as the most common means of specifying data. A dictionary is an entity that contains data entries that can be retrieved by the I/O by means of *keywords*. The keyword entries follow the general format

```
<keyword> <dataEntry1> ... <dataEntryN>;
```

Most entries are single data entries of the form:

```
<keyword> <dataEntry>;
```

Most OpenFOAM data files are themselves dictionaries containing a set of keyword entries. Dictionaries provide the means for organising entries into logical categories and can be specified hierarchically so that any dictionary can itself contain one or more dictionary entries. The format for a dictionary is to specify the dictionary name followed by keyword entries enclosed in curly braces {} as follows.

```
<dictionaryName>
{
    ... keyword entries ...
}
```

4.2.3 The data file header

All data files that are read and written by OpenFOAM begin with a dictionary named `FoamFile` containing a standard set of keyword entries, listed in Table 4.1. The table

Keyword	Description	Entry
<code>version</code>	I/O format version	2.0
<code>format</code>	Data format	<code>ascii</code> / <code>binary</code>
<code>location</code>	Path to the file, in "..."	(optional)
<code>class</code>	OpenFOAM class constructed from the data file concerned	typically <code>dictionary</code> or a field, e.g. <code>volVectorField</code>
<code>object</code>	Filename	e.g. <code>controlDict</code>

Table 4.1: Header keywords entries for data files.

provides brief descriptions of each entry, which is probably sufficient for most entries with the notable exception of `class`. The `class` entry is the name of the C++ class in the OpenFOAM library that will be constructed from the data in the file. Without knowledge of the underlying code which calls the file to be read, and knowledge of the OpenFOAM classes, the user will probably be unable to surmise the `class` entry correctly. However, most data files with simple keyword entries are read into an internal `dictionary` class and therefore the `class` entry is `dictionary` in those cases.

The following example shows the use of keywords to provide data for a case using the types of entry described so far. The extract, from an *fvSolution* dictionary file, contains 2 dictionaries, *solvers* and *PISO*. The *solvers* dictionary contains multiple data entries for

solver and tolerances for each of the pressure and velocity equations, represented by the p and U keywords respectively; the *PISO* dictionary contains algorithm controls.

```

17  solvers
18  {
19      p
20      {
21          solver          PCG;
22          preconditioner DIC;
23          tolerance      1e-06;
24          relTol         0.05;
25      }
26
27      pFinal
28      {
29          $p;
30          relTol         0;
31      }
32
33      U
34      {
35          solver          smoothSolver;
36          smoother        symGaussSeidel;
37          tolerance      1e-05;
38          relTol         0;
39      }
40
41  PISO
42  {
43      nCorrectors     2;
44      nNonOrthogonalCorrectors 0;
45      pRefCell       0;
46      pRefValue      0;
47  }
48
49 // ****
50

```

4.2.4 Lists

OpenFOAM applications contain lists, *e.g.* a list of vertex coordinates for a mesh description. Lists are commonly found in I/O and have a format of their own in which the entries are contained within round braces (). There is also a choice of format preceding the round braces:

simple the keyword is followed immediately by round braces

```

<listName>
(
    ...
);

```

numbered the keyword is followed by the number of elements <n> in the list

```

<listName>
<n>
(
    ...
);

```

token identifier the keyword is followed by a class name identifier **Label<Type>** where <Type> states what the list contains, *e.g.* for a list of **scalar** elements is

```
<listName>
List<scalar>
<n>      // optional
(
    ...
);

```

Note that `<scalar>` in `List<scalar>` is not a generic name but the actual text that should be entered.

The simple format is a convenient way of writing a list. The other formats allow the code to read the data faster since the size of the list can be allocated to memory in advance of reading the data. The simple format is therefore preferred for short lists, where read time is minimal, and the other formats are preferred for long lists.

4.2.5 Scalars, vectors and tensors

A scalar is a single number represented as such in a data file. A vector is a `VectorSpace` of rank 1 and dimension 3, and since the number of elements is always fixed to 3, the simple List format is used. Therefore a vector (1.0, 1.1, 1.2) is written:

```
(1.0 1.1 1.2)
```

In OpenFOAM, a tensor is a `VectorSpace` of rank 2 and dimension 3 and therefore the data entries are always fixed to 9 real numbers. Therefore the identity tensor can be written:

```
(  
    1 0 0  
    0 1 0  
    0 0 1  
)
```

This example demonstrates the way in which OpenFOAM ignores the line return so that the entry can be written over multiple lines. It is treated no differently to listing the numbers on a single line:

```
( 1 0 0 0 1 0 0 0 1 )
```

4.2.6 Dimensional units

In continuum mechanics, properties are represented in some chosen units, *e.g.* mass in kilograms (kg), volume in cubic metres (m^3), pressure in Pascals ($\text{kg m}^{-1} \text{s}^{-2}$). Algebraic operations must be performed on these properties using consistent units of measurement; in particular, addition, subtraction and equality are only physically meaningful for properties of the same dimensional units. As a safeguard against implementing a meaningless operation, OpenFOAM attaches dimensions to field data and physical properties and performs dimension checking on any tensor operation.

The I/O format for a `dimensionSet` is 7 scalars delimited by square brackets, *e.g.*

No.	Property	SI unit	USCS unit
1	Mass	kilogram (kg)	pound-mass (lbf)
2	Length	metre (m)	foot (ft)
3	Time	second (s)	second (s)
4	Temperature	Kelvin (K)	degree Rankine ($^{\circ}$ R)
5	Quantity	mole (mol)	mole (mol)
6	Current	ampere (A)	ampere (A)
7	Luminous intensity	candela (cd)	candela (cd)

Table 4.2: Base units for SI and USCS

```
[0 2 -1 0 0 0]
```

where each of the values corresponds to the power of each of the base units of measurement listed in Table 4.2. The table gives the base units for the Système International (SI) and the United States Customary System (USCS) but OpenFOAM can be used with any system of units. All that is required is that the *input data is correct for the chosen set of units*. It is particularly important to recognise that OpenFOAM requires some dimensioned physical constants, *e.g.* the Universal Gas Constant R , for certain calculations, *e.g.* thermophysical modelling. These dimensioned constants are specified in a *DimensionedConstant* sub-dictionary of main *controlDict* file of the OpenFOAM installation (*\$WM_PROJECT_DIR/etc/controlDict*). By default these constants are set in SI units. Those wishing to use the USCS or any other system of units should modify these constants to their chosen set of units accordingly.

4.2.7 Dimensioned types

Physical properties are typically specified with their associated dimensions. These entries formally have the format that the following example of a *dimensionedScalar* demonstrates:

```
nu           nu [0 2 -1 0 0 0] 1;
```

The first **nu** is the keyword; the second **nu** is the word name stored in class **word**, usually chosen to be the same as the keyword; the next entry is the **dimensionSet** and the final entry is the **scalar** value.

The majority of dimensioned keyword lookups set a default for the word name which can therefore be omitted from the entry, so the more common syntax is:

```
nu           [0 2 -1 0 0 0] 1;
```

4.2.8 Fields

Much of the I/O data in OpenFOAM are tensor fields, *e.g.* velocity, pressure data, that are read from and written into the time directories. OpenFOAM writes field data using keyword entries as described in Table 4.3.

The data begins with an entry for its **dimensions**. Following that, is the **internalField**, described in one of the following ways.

Keyword	Description	Example
dimensions	Dimensions of field	[1 1 -2 0 0 0 0]
internalField	Value of internal field	uniform (1 0 0)
boundaryField	Boundary field	see file listing in section 4.2.8

Table 4.3: Main keywords used in field dictionaries.

Uniform field a single value is assigned to all elements within the field, taking the form:

```
internalField uniform <entry>;
```

Nonuniform field each field element is assigned a unique value from a list, taking the following form where the token identifier form of list is recommended:

```
internalField nonuniform <List>;
```

The **boundaryField** is a dictionary containing a set of entries whose names correspond to each of the names of the boundary patches listed in the *boundary* file in the *polyMesh* directory. Each patch entry is itself a dictionary containing a list of keyword entries. The mandatory entry, **type**, describes the patch field condition specified for the field. The remaining entries correspond to the type of patch field condition selected and can typically include field data specifying initial conditions on patch faces. A selection of patch field conditions available in OpenFOAM are listed in section 5.2.1, section 5.2.2 and section 5.2.3, with a description and the data that must be specified with it. Example field dictionary entries for velocity U are shown below:

```

17 dimensions      [0 1 -1 0 0 0];
18
19 internalField   uniform (0 0 0);
20
21 boundaryField
22 {
23     movingWall
24     {
25         type          fixedValue;
26         value         uniform (1 0 0);
27     }
28
29     fixedWalls
30     {
31         type          noSlip;
32     }
33
34     frontAndBack
35     {
36         type          empty;
37     }
38 }
39 // ****
40

```

4.2.9 Macro expansion

OpenFOAM dictionary files include a macro syntax to allow convenient configuration of case files. The syntax uses the the dollar (\$) symbol in front of a keyword to expand the data associated with the keyword. For example the value set for keyword a below, 10, is expanded in the following line, so that the value of b is also 10.

```
a 10;
b $a;
```

Variables can be accessed within different levels of sub-dictionaries, or scope. Scoping is performed using a ‘.’ (dot) syntax, illustrated by the following example, where **b** is set to the value of **a**, specified in a sub-dictionary called **subdict**.

```
subdict
{
    a 10;
}
b $subdict.a;
```

There are further syntax rules for macro expansions:

- to traverse up one level of sub-dictionary, use the ‘..’ (double-dot) prefix, see below;
- to traverse up two levels use ‘...’ (triple-dot) prefix, *etc.*;
- to traverse to the top level dictionary use the ‘:’ (colon) prefix (most useful), see below;
- for multiple levels of macro substitution, each specified with the ‘\$’ dollar syntax, ‘{}’ brackets are required to protect the expansion, see below.

```
a 10;
b a;
c ${${b}}; // returns 10, since $b returns "a", and $a returns 10

subdict
{
    b $..a; // double-dot takes scope up 1 level, then "a" is available

    subsubdict
    {
        c $:a; // colon takes scope to top level, then "a" is available
    }
}
```

4.2.10 Including files

There is additional file syntax that provides further flexibility for setting up of OpenFOAM case files, namely directives. Directives are commands that can be contained within case files that begin with the hash (#) symbol. The first set of directive commands are those for reading a data file from within another data file. For example, let us say a user wishes to set an initial value of pressure once to be used as the internal field and initial value at a boundary. We could create a file, *e.g.* named *initialConditions*, which contains the following entries:

```
pressure 1e+05;
```

In order to use this pressure for both the internal and initial boundary fields, the user would simply include the *initialConditions* file using the `#include` directive, then use macro expansions for the `pressure` keyword, as follows.

```
#include "initialConditions"
internalField uniform $pressure;
boundaryField
{
    patch1
    {
        type fixedValue;
        value $internalField;
    }
}
```

The file include directives are as follows:

```
#include "<path>/<fileName>" reads the file of name <fileName> from an absolute or
relative directory path <path>;

#includeIfPresent "<path>/<fileName>" reads the file if it exists;

#includeEtc "<path>/<fileName>" reads the file of name <fileName> from the directory
path <path>, relative to the $FOAM_ETCH directory;

#includeFunc <fileName> reads the file of name <fileName>, searched from the case system
directory, followed by the $FOAM_ETCH directory;

#remove <keywordEntry> removes any included keyword entry; can take a word or regular
expression;
```

4.2.11 Regular expressions

When running an application, data is initialised by looking up keywords from dictionaries. The user can either provide an entry with a keyword that directly matches the one being looked up, or can provide a **POSIX regular expression** that matches the keyword, specified inside double-quotations ("..."). Regular expressions have an extensive syntax for various matches of text patterns but they are typically only used in the following ways in OpenFOAM input files.

- "`inlet.*`" matches any word beginning `inlet...`, including `inlet` itself, because ‘.’ denotes “any character” and ‘*’ denotes “repeated any number of times, including 0 times”.
- "`(inlet|output)`" matches `inlet` and `outlet` because () specified an expression grouping and | is an OR operator.

4.2.12 Keyword ordering

The order in which keywords are listed does not matter, except when *the same keyword is specified multiple times*. Where the same keyword is duplicated, the last instance is used. The most common example of a duplicate keyword occurs when a keyword is included from the file or expanded from a macro, and then overridden. The example below demonstrates this, where `pFinal` adopts all the keyword entries, including `relTol 0.05` in the `p` sub-dictionary by the macro expansion `$p`, then overrides the `relTol` entry.

```

p
{
    solver          PCG;
    preconditioner DIC;
    tolerance       1e-6;
    relTol         0.05;
}
pFinal
{
    $p;
    relTol        0;
}

```

Where a data lookup matches both a keyword and a regular expression, the keyword match takes precedence irrespective of the order of the entries.

4.2.13 Inline calculations and code

There are two further directives that enable calculations from within input files: `#calc`, for simple calculations; `#codeStream`, for more complex calculations.

The pipeCyclic tutorial in `$FOAM_TUTORIALS/incompressible/simpleFoam` demonstrates the `#calc` directive through its `blockMesh` configuration in `blockMeshDict`:

```

//- Half angle of wedge in degrees
halfAngle 45.0;

//- Radius of pipe [m]
radius 0.5;

radHalfAngle #calc "degToRad($halfAngle)";
y            #calc "$radius*sin($radHalfAngle)";
minY         #calc "-1.0*$y";
z            #calc "$radius*cos($radHalfAngle)";
minZ         #calc "-1.0*$z";

```

The file contains several calculations that calculate vertex ordinates, *e.g.* `y`, `z`, *etc.*, from geometry dimensions, *e.g.* `radius`. Calculations include standard C++ functions including unit conversions, *e.g.* `degToRad`, and trigonometric functions, *e.g.* `sin`.

The `#codeStream` directive takes C++ code which is compiled and executed to deliver the dictionary entry. The code and compilation instructions are specified through the following keywords.

- `code`: specifies the code, called with arguments `OStream& os` and `const dictionary& dict` which the user can use in the code, *e.g.* to lookup keyword entries from within the current case dictionary (file).
- `codeInclude` (optional): specifies additional C++ `#include` statements to include OpenFOAM files.
- `codeOptions` (optional): specifies any extra compilation flags to be added to `EXE_INC` in *Make/options*.
- `codeLibs` (optional): specifies any extra compilation flags to be added to `LIB_LIBS` in *Make/options*.

Code, like any string, can be written across multiple lines by enclosing it within hash-bracket delimiters, *i.e.* `#{...#}`. Anything in between these two delimiters becomes a string with all newlines, quotes, *etc.* preserved.

An example of `#codeStream` is given below, where the code in the calculates moment of inertia of a box shaped geometry.

```
momentOfInertia #codeStream
{
    codeInclude
    #{  
        #include "diagTensor.H"  
    };  
  
    code
    #{
        scalar sqrLx = sqr($Lx);
        scalar sqrLy = sqr($Ly);
        scalar sqrLz = sqr($Lz);
        os <<
            $mass
        *diagTensor(sqrLy + sqrLz, sqrLx + sqrLz, sqrLx + sqrLy)/12.0;
    };  
};
```

4.3 Time and data input/output control

The OpenFOAM solvers begin all runs by setting up a database. The database controls I/O and, since output of data is usually requested at intervals of time during the run, time is an inextricable part of the database. The `controlDict` dictionary sets input parameters *essential* for the creation of the database. The keyword entries in `controlDict` are listed in the following sections. Only the time control and `writeInterval` entries are mandatory, with the database using default values for any of the optional entries that are omitted. Example entries from a `controlDict` dictionary are given below:

```
17 application      icoFoam;
18 startFrom       startTime;
20 startTime        0;
21 stopAt          endTime;
```

```

24
25   endTime       0.5;
26
27   deltaT        0.005;
28
29   writeControl   timeStep;
30
31   writeInterval  20;
32
33   purgeWrite     0;
34
35   writeFormat    ascii;
36
37   writePrecision 6;
38
39   writeCompression off;
40
41   timeFormat     general;
42
43   timePrecision  6;
44
45   runTimeModifiable true;
46
47 // ****

```

4.3.1 Time control

startFrom Controls the start time of the simulation.

- **firstTime**: Earliest time step from the set of time directories.
- **startTime**: Time specified by the **startTime** keyword entry.
- **latestTime**: Most recent time step from the set of time directories.

startTime Start time for the simulation with **startFrom startTime;**

stopAt Controls the end time of the simulation.

- **endTime**: Time specified by the **endTime** keyword entry.
- **writeNow**: Stops simulation on completion of current time step and writes data.
- **noWriteNow**: Stops simulation on completion of current time step and does not write out data.
- **nextWrite**: Stops simulation on completion of next scheduled write time, specified by **writeControl**.

endTime End time for the simulation when **stopAt endTime;** is specified.

deltaT Time step of the simulation.

4.3.2 Data writing

writeControl Controls the timing of write output to file.

- **timeStep**: Writes data every **writeInterval** time steps.
- **runTime**: Writes data every **writeInterval** seconds of simulated time.
- **adjustableRunTime**: Writes data every **writeInterval** seconds of simulated time, adjusting the time steps to coincide with the **writeInterval** if necessary — used in cases with automatic time step adjustment.

- `cpuTime`: Writes data every `writeInterval` seconds of CPU time.
- `clockTime`: Writes data out every `writeInterval` seconds of real time.

`writeInterval` Scalar used in conjunction with `writeControl` described above.

`purgeWrite` Integer representing a limit on the number of time directories that are stored by overwriting time directories on a cyclic basis. For example, if the simulations starts at $t = 5\text{s}$ and $\Delta t = 1\text{s}$, then with `purgeWrite 2;`, data is first written into 2 directories, 6 and 7, then when 8 is written, 6 is deleted, and so on so that only 2 new results directories exists at any time. *To disable the purging, specify `purgeWrite 0;` (default).*

`writeFormat` Specifies the format of the data files.

- `ascii` (default): ASCII format, written to `writePrecision` significant figures.
- `binary`: binary format.

`writePrecision` Integer used in conjunction with `writeFormat` described above, 6 by default.

`writeCompression` Switch to specify whether files are compressed with `gzip` when written: `on/off (yes/no, true/false)`

`timeFormat` Choice of format of the naming of the time directories.

- `fixed`: $\pm m.aaaaaaaaaa$ where the number of a s is set by `timePrecision`.
- `scientific`: $\pm m.aaaaaaaaaa \times 10^{xx}$ where the number of a s is set by `timePrecision`.
- `general` (default): Specifies `scientific` format if the exponent is less than -4 or greater than or equal to that specified by `timePrecision`.

`timePrecision` Integer used in conjunction with `timeFormat` described above, 6 by default.

`graphFormat` Format for graph data written by an application.

- `raw` (default): Raw ASCII format in columns.
- `gnuplot`: Data in `gnuplot` format.
- `xmgr`: Data in `Grace/xmgr` format.
- `jplot`: Data in `jPlot` format.

4.3.3 Other settings

`adjustTimeStep` Switch used by some solvers to adjust the time step during the simulation, usually according to `maxCo`.

`maxCo` Maximum Courant number, e.g. 0.5

`runTimeModifiable` Switch for whether dictionaries, e.g. `controlDict`, are re-read during a simulation at the beginning of each time step, allowing the user to modify parameters during a simulation.

```

libs List of additional libraries (on $LD_LIBRARY_PATH) to be loaded at run-time,
e.g.("libNew1.so" "libNew2.so")

functions Dictionary of functions, e.g. probes to be loaded at run-time; see examples in
$FOAM_TUTORIALS

```

4.4 Numerical schemes

The *fvSchemes* dictionary in the *system* directory sets the numerical schemes for terms, such as derivatives in equations, that are calculated during a simulation. This section describes how to specify the schemes in the *fvSchemes* dictionary.

The terms that must typically be assigned a numerical scheme in *fvSchemes* range from derivatives, *e.g.* gradient ∇ , to interpolations of values from one set of points to another. The aim in OpenFOAM is to offer an unrestricted choice to the user, starting with the choice of discretisation practice which is generally standard Gaussian finite volume integration. Gaussian integration is based on summing values on cell faces, which must be interpolated from cell centres. The user has a wide range of options for interpolation scheme, with certain schemes being specifically designed for particular derivative terms, especially the advection divergence $\nabla \cdot$ terms.

The set of terms, for which numerical schemes must be specified, are subdivided within the *fvSchemes* dictionary into the categories below.

- **timeScheme**: first and second time derivatives, *e.g.* $\partial/\partial t, \partial^2/\partial t^2$
- **gradSchemes**: gradient ∇
- **divSchemes**: divergence $\nabla \cdot$
- **laplacianSchemes**: Laplacian ∇^2
- **interpolationSchemes**: cell to face interpolations of values.
- **snGradSchemes**: component of gradient normal to a cell face.
- **wallDist**: distance to wall calculation, where required.

Each keyword in represents the name of a sub-dictionary which contains terms of a particular type, *e.g.* **gradSchemes** contains all the gradient derivative terms such as **grad(p)** (which represents ∇p). Further examples can be seen in the extract from an *fvSchemes* dictionary below:

```

17 ddtSchemes
18 {
19     default      Euler;
20 }
21
22 gradSchemes
23 {
24     default      Gauss linear;
25 }
26
27 divSchemes
28 {
29     default      none;
30     div(phi,U)   bounded Gauss linearUpwind grad(U);
31     div(phi,k)   bounded Gauss upwind;
32     div(phi,epsilon) bounded Gauss upwind;

```

```

33     div(phi,R)      bounded Gauss upwind;
34     div(R)          Gauss linear;
35     div(phi,nuTilda) bounded Gauss upwind;
36     div((nuEff*dev2(T(grad(U))))) Gauss linear;
37 }
38
39 laplacianSchemes
40 {
41     default          Gauss linear corrected;
42 }
43
44 interpolationSchemes
45 {
46     default          linear;
47 }
48
49 snGradSchemes
50 {
51     default          corrected;
52 }
53 // ****

```

The example shows that the `fvSchemes` dictionary contains 6 ... *Schemes* subdictionaries containing keyword entries for each term specified within including: a `default` entry; other entries whose names correspond to a word identifier for the particular term specified, *e.g.* `grad(p)` for ∇p . If a `default` scheme is specified in a particular ... *Schemes* sub-dictionary, it is assigned to all of the terms to which the sub-dictionary refers, *e.g.* specifying a `default` in `gradSchemes` sets the scheme for all gradient terms in the application, *e.g.* ∇p , ∇U . When a `default` is specified, it is not necessary to specify each specific term itself in that sub-dictionary, *i.e.* the entries for `grad(p)`, `grad(U)` in this example. However, if any of these terms are included, the specified scheme overrides the `default` scheme for that term.

Alternatively the user can specify that no `default` scheme by the `none` entry, as in the `divSchemes` in the example above. In this instance the user is obliged to specify all terms in that sub-dictionary individually. Setting `default` to `none` may appear superfluous since `default` can be overridden. However, specifying `none` forces the user to specify all terms individually which can be useful to remind the user which terms are actually present in the application.

OpenFOAM includes a vast number of discretisation schemes, from which only a few are typically recommended for real-world, engineering applications. The user can get help with scheme selection by interrogating the tutorial cases for example scheme settings. They should look at the schemes used in relevant cases, *e.g.* for running a large-eddy simulation (LES), look at schemes used in tutorials running LES. Additionally, `foamSearch` provides a useful tool to get a quick list of schemes used in all the tutorials. For example, to print all the `default` entries for `ddtSchemes` for cases in the `$FOAM_TUTORIALS` directory, the user can type:

```
foamSearch $FOAM_TUTORIALS fvSchemes ddtSchemes.default
```

which prints:

```

default      backward;
default      CrankNicolson 0.9;
default      Euler;
default      localEuler;
```

```
default      none;
default      steadyState;
```

The schemes listed using `foamSearch` are described in the following sections.

4.4.1 Time schemes

The first time derivative ($\partial/\partial t$) terms are specified in the `ddtSchemes` sub-dictionary. The discretisation schemes for each term can be selected from those listed below.

- `steadyState`: sets time derivatives to zero.
- `Euler`: transient, first order implicit, bounded.
- `backward`: transient, second order implicit, potentially unbounded.
- `CrankNicolson`: transient, second order implicit, bounded; requires an off-centering coefficient ψ where:

$$\psi = \begin{cases} 1 & \text{corresponds to pure CrankNicolson,} \\ 0 & \text{corresponds to Euler;} \end{cases}$$

generally $\psi = 0.9$ is used to bound/stabilise the scheme for practical engineering problems.

- `localEuler`: pseudo transient for accelerating a solution to steady-state using local-time stepping; first order implicit.

Solvers are generally configured to simulate either transient or steady-state. Changing the time scheme from one which is steady-state to transient, or visa versa, does not affect the fundamental nature of the solver and so fails to achieve its purpose, yielding a nonsensical solution.

Any second time derivative ($\partial^2/\partial t^2$) terms are specified in the `d2dt2Schemes` sub-dictionary. Only the `Euler` scheme is available for `d2dt2Schemes`.

4.4.2 Gradient schemes

The `gradSchemes` sub-dictionary contains gradient terms. The `default` discretisation scheme that is primarily used for gradient terms is:

```
default      Gauss linear;
```

The `Gauss` entry specifies the standard finite volume discretisation of Gaussian integration which requires the interpolation of values from cell centres to face centres. The interpolation scheme is then given by the `linear` entry, meaning linear interpolation or central differencing.

In some tutorials cases, particular involving poorer quality meshes, the discretisation of specific gradient terms is overridden to improve boundedness and stability. The terms that are overridden in those cases are the velocity gradient

```
grad(U)           cellLimited Gauss linear 1;
```

and, less frequently, the gradient of turbulence fields, *e.g.*

```
grad(k)           cellLimited Gauss linear 1;
grad(epsilon)    cellLimited Gauss linear 1;
```

They use the **cellLimited** scheme which limits the gradient such that when cell values are extrapolated to faces using the calculated gradient, the face values do not fall outside the bounds of values in surrounding cells. A limiting coefficient is specified after the underlying scheme for which 1 guarantees boundedness and 0 applies no limiting; 1 is invariably used.

Other schemes that are rarely used are as follows.

- **leastSquares**: a second-order, least squares distance calculation using all neighbour cells.
- **Gauss cubic**: third-order scheme that appears in the **dnsFoam** simulation on a regular mesh.

4.4.3 Divergence schemes

The **divSchemes** sub-dictionary contains divergence terms, *i.e.* terms of the form $\nabla \cdot \dots$, excluding Laplacian terms (of the form $\nabla \cdot (\Gamma \nabla \dots)$). This includes both advection terms, *e.g.* $\nabla \cdot (\mathbf{U} k)$, where velocity **U** provides the advective flux, and other terms, that are often diffusive in nature, *e.g.* $\nabla \cdot \nu(\nabla \mathbf{U})^T$.

The fact that terms that are fundamentally different reside in one sub-dictionary means that the **default** scheme is generally set to **none** in **divSchemes**. The non-advective terms then generally use the **Gauss** integration with **linear** interpolation, *e.g.*

```
div(U)           Gauss linear;
```

The treatment of advective terms is one of the major challenges in CFD numerics and so the options are more extensive. The keyword identifier for the advective terms are usually of the form **div(phi, ...)**, where **phi** generally denotes the (volumetric) flux of velocity on the cell faces for constant-density flows and the mass flux for compressible flows, *e.g.* **div(phi,U)** for the advection of velocity, **div(phi,e)** for the advection of internal energy, **div(phi,k)** for turbulent kinetic energy, *etc.* For advection of velocity, the user can run the **foamSearch** script to extract the **div(phi,U)** keyword from all tutorials.

```
foamSearch $FOAM_TUTORIALS fvSchemes "divSchemes.div(phi,U)"
```

The schemes are all based on **Gauss** integration, using the flux **phi** and the advected field being interpolated to the cell faces by one of a selection of schemes, *e.g.* **linear**, **linearUpwind**, *etc.* There is a **bounded** variant of the discretisation, discussed later.

Ignoring ‘V’-schemes (with keywords ending “V”), and rarely-used schemes such as **Gauss cubic** and **vanLeerV**, the interpolation schemes used in the tutorials are as follows.

- **linear**: second order, unbounded.

- `linearUpwind`: second order, upwind-biased, unbounded (but much less so than `linear`), that requires discretisation of the velocity gradient to be specified.
- `LUST`: blended 75% `linear` / 25%`linearUpwind` scheme, that requires discretisation of the velocity gradient to be specified.
- `limitedLinear`: `linear` scheme that limits towards `upwind` in regions of rapidly changing gradient; requires a coefficient, where 1 is strongest limiting, tending towards `linear` as the coefficient tends to 0.
- `upwind`: first-order bounded, generally too inaccurate to be recommended.

Example syntax for these schemes is as follows.

```
div(phi,U)      Gauss linear;
div(phi,U)      Gauss linearUpwind grad(U);
div(phi,U)      Gauss LUST grad(U);
div(phi,U)      Gauss LUST unlimitedGrad(U);
div(phi,U)      Gauss limitedLinear 1;
div(phi,U)      Gauss upwind;
```

‘V’-schemes are specialised versions of schemes designed for vector fields. They differ from conventional schemes by calculating a single limiter which is applied to all components of the vectors, rather than calculating separate limiters for each component of the vector. The ‘V’-schemes’ single limiter is calculated based on the direction of most rapidly changing gradient, resulting in the strongest limiter being calculated which is most stable but arguably less accurate. Example syntax is as follows.

```
div(phi,U)      Gauss limitedLinearV 1;
div(phi,U)      Gauss linearUpwindV grad(U);
```

The bounded variants of schemes relate to the treatment of the material time derivative which can be expressed in terms of a spatial time derivative and convection, *e.g.* for field e in incompressible flow

$$\frac{De}{Dt} = \frac{\partial e}{\partial t} + \mathbf{U} \cdot \nabla e = \frac{\partial e}{\partial t} + \nabla \cdot (\mathbf{U}e) - (\nabla \cdot \mathbf{U})e \quad (4.1)$$

For numerical solution of incompressible flows, $\nabla \cdot \mathbf{U} = 0$ at convergence, at which point the third term on the right hand side is zero. Before convergence is reached, however, $\nabla \cdot \mathbf{U} \neq 0$ and in some circumstances, particularly steady-state simulations, it is better to include the third term within a numerical solution because it helps maintain boundedness of the solution variable and promotes better convergence. The bounded variant of the `Gauss` scheme provides this, automatically including the discretisation of the third-term with the advection term. Example syntax is as follows, as seen in `fvSchemes` files for steady-state cases, *e.g.* for the `simpleFoam` tutorials

```
div(phi,U)      bounded Gauss limitedLinearV 1;
div(phi,U)      bounded Gauss linearUpwindV grad(U);
```

The schemes used for advection of scalar fields are similar to those for advection of velocity, although in general there is greater emphasis placed on boundedness than accuracy when selecting the schemes. For example, a search for schemes for advection of internal energy (*e*) reveals the following.

```
foamSearch $FOAM_TUTORIALS fvSchemes "divSchemes.div(phi,e)"

div(phi,e)      bounded Gauss upwind;
div(phi,e)      Gauss limitedLinear 1;
div(phi,e)      Gauss LUST grad(e);
div(phi,e)      Gauss upwind;
div(phi,e)      Gauss vanLeer;
```

In comparison with advection of velocity, there are no cases set up to use `linear` or `linearUpwind`. Instead the `limitedLinear` and `upwind` schemes are commonly used, with the additional appearance of `vanLeer`, another limited scheme, with less strong limiting than `limitedLinear`.

There are specialised versions of the limited schemes for scalar fields that are commonly bounded between 0 and 1, *e.g.* the laminar flame speed regress variable *b*. A search for the discretisation used for advection in the laminar flame transport equation yields:

```
div(phiSt,b)    Gauss limitedLinear01 1;
```

The underlying scheme is `limitedLinear`, specialised for stronger bounding between 0 and 1 by adding `01` to the name of the scheme.

The `multivariateSelection` mechanism also exists for grouping multiple equation terms together, and applying the same limiters on all terms, using the strongest limiter calculated for all terms. A good example of this is in a set of mass transport equations for fluid species, where it is good practice to apply the same discretisation to all equations for consistency. The example below comes from the `smallPoolFire3D` tutorial in `$FOAM_TUTORIALS/combustion/fireFoam/les`, in which the equation for enthalpy *h* is included with the specie mass transport equations in the calculation of a single limiter.

```
div(phi,Yi_h)    Gauss multivariateSelection
{
    O2 limitedLinear01 1;
    CH4 limitedLinear01 1;
    N2 limitedLinear01 1;
    H2O limitedLinear01 1;
    CO2 limitedLinear01 1;
    h limitedLinear 1 ;
}
```

4.4.4 Surface normal gradient schemes

It is worth explaining the `snGradSchemes` sub-dictionary that contains surface normal gradient terms, before discussion of `laplacianSchemes`, because they are required to evaluate a

Laplacian term using Gaussian integration. A surface normal gradient is evaluated at a cell face; it is the component, normal to the face, of the gradient of values at the centres of the 2 cells that the face connects.

A search for the `default` scheme for `snGradSchemes` reveals the following entries.

```
default      corrected;
default      limited corrected 0.33;
default      limited corrected 0.5;
default      orthogonal;
default      uncorrected;
```

The basis of the gradient calculation at a face is to subtract the value at the cell centre on one side of the face from the value in the centre on the other side and divide by the distance. The calculation is second-order accurate for the gradient *normal to the face* if the vector connecting the cell centres is orthogonal to the face, *i.e.* they are at right-angles. This is the `orthogonal` scheme.

Orthogonality requires a regular mesh, typically aligned with the Catersian co-ordinate system, which does not normally occur in meshes for real world, engineering geometries. Therefore, to maintain second-order accuracy, an explicit non-orthogonal correction can be added to the orthogonal component, known as the `corrected` scheme. The correction increases in size as the non-orthonormality, the angle α between the cell-cell vector and face normal vector, increases.

As α tends towards 90° , *e.g.* beyond 70° , the explicit correction can be so large to cause a solution to go unstable. The solution can be stabilised by applying the `limited` scheme to the correction which requires a coefficient ψ , $0 \leq \psi \leq 1$ where

$$\psi = \begin{cases} 0 & \text{corresponds to } \text{uncorrected}, \\ 0.333 & \text{non-orthogonal correction } \leq 0.5 \times \text{orthogonal part}, \\ 0.5 & \text{non-orthogonal correction } \leq \text{orthogonal part}, \\ 1 & \text{corresponds to } \text{corrected}. \end{cases} \quad (4.2)$$

Typically, ψ is chosen to be 0.33 or 0.5, where 0.33 offers greater stability and 0.5 greater accuracy.

The corrected scheme applies under-relaxation in which the implicit orthogonal calculation is increased by $\cos^{-1}\alpha$, with an equivalent boost within the non-orthogonal correction. The `uncorrected` scheme is equivalent to the `corrected` scheme, without the non-orthogonal correction, so includes is like `orthogonal` but with the $\cos^{-1}\alpha$ under-relaxation.

Generally the `uncorrected` and `orthogonal` schemes are only recommended for meshes with very low non-orthogonality (*e.g.* maximum 5°). The `corrected` scheme is generally recommended, but for maximum non-orthogonality above 70° , `limited` may be required. At non-orthogonality above 80° , convergence is generally hard to achieve.

4.4.5 Laplacian schemes

The `laplacianSchemes` sub-dictionary contains Laplacian terms. A typical Laplacian term is $\nabla \cdot (\nu \nabla \mathbf{U})$, the diffusion term in the momentum equations, which corresponds to the keyword `laplacian(nu,U)` in `laplacianSchemes`. The `Gauss` scheme is the only choice of discretisation and requires a selection of both an interpolation scheme for the diffusion

coefficient, *i.e.* ν in our example, and a surface normal gradient scheme, *i.e.* $\nabla \mathbf{U}$. To summarise, the entries required are:

```
Gauss <interpolationScheme> <snGradScheme>
```

The user can search for the `default` scheme for *laplacianSchemes* in all the cases in the `$FOAM_TUTORIALS` directory.

```
foamSearch $FOAM_TUTORIALS fvSchemes laplacianSchemes.default
```

It reveals the following entries.

```
default      Gauss linear corrected;
default      Gauss linear limited corrected 0.33;
default      Gauss linear limited corrected 0.5;
default      Gauss linear orthogonal;
default      Gauss linear uncorrected;
```

In all cases, the `linear` interpolation scheme is used for interpolation of the diffusivity. The cases uses the same array of `snGradSchemes` based on level on non-orthogonality, as described in section 4.4.4.

4.4.6 Interpolation schemes

The *interpolationSchemes* sub-dictionary contains terms that are interpolations of values typically from cell centres to face centres, primarily used in the interpolation of velocity to face centres for the calculation of flux phi. There are numerous interpolation schemes in OpenFOAM, but a search for the `default` scheme in all the tutorial cases reveals that `linear` interpolation is used in almost every case, except for 2-3 unusual cases, *e.g.* DNS on a regular mesh, stress analysis, where `cubic` interpolation is used.

4.5 Solution and algorithm control

The equation solvers, tolerances and algorithms are controlled from the *fvSolution* dictionary in the `system` directory. Below is an example set of entries from the *fvSolution* dictionary required for the `icoFoam` solver.

```
17  solvers
18  {
19    p
20    {
21      solver          PCG;
22      preconditioner DIC;
23      tolerance       1e-06;
24      relTol          0.05;
25    }
26
27    pFinal
28    {
29      $p;
30      relTol          0;
31    }
32
33    U
34  {
```

```

35     solver      smoothSolver;
36     smoother    symGaussSeidel;
37     tolerance   1e-05;
38     relTol     0;
39 }
40
41 PISO
42 {
43     nCorrectors 2;
44     nNonOrthogonalCorrectors 0;
45     pRefCell    0;
46     pRefValue   0;
47 }
48 // ****
49
50

```

fvSolution contains a set of subdictionaries, described in the remainder of this section that includes: *solvers*; *relaxationFactors*; and, *PISO*, *SIMPLE* or *PIMPLE*.

4.5.1 Linear solver control

The first sub-dictionary in our example is **solvers**. It specifies each linear-solver that is used for each discretised equation; here, the term *linear*-solver refers to the method of number-crunching to solve a matrix equation, as opposed to an *application* solver, such as **simpleFoam** which describes the entire set of equations and algorithms to solve a particular problem. The term ‘linear-solver’ is abbreviated to ‘solver’ in much of what follows; hopefully the context of the term avoids any ambiguity.

The syntax for each entry within **solvers** starts with a keyword that is of the variable being solved in the particular equation. For example, **icoFoam** solves equations for velocity **U** and pressure **p**, hence the entries for **U** and **p**. The keyword relates to a sub-dictionary containing the type of solver and the parameters that the solver uses. The solver is selected through the **solver** keyword from the options listed below. The parameters, including **tolerance**, **relTol**, **preconditioner**, etc. are described in following sections.

- **PCG/PBiCGStab**: Stabilised preconditioned (bi-)conjugate gradient, for both symmetric and asymmetric matrices.
- **PCG/PBiCG**: preconditioned (bi-)conjugate gradient, with PCG for symmetric matrices, PBiCG for asymmetric matrices.
- **smoothSolver**: solver that uses a smoother.
- **GAMG**: generalised geometric-algebraic multi-grid.
- **diagonal**: diagonal solver for explicit systems.

The solvers distinguish between symmetric matrices and asymmetric matrices. The symmetry of the matrix depends on the terms of the equation being solved, e.g. time derivatives and Laplacian terms form coefficients of a symmetric matrix, whereas an advective derivative introduces asymmetry. If the user specifies a symmetric solver for an asymmetric matrix, or vice versa, an error message will be written to advise the user accordingly, e.g.

```

--> FOAM FATAL IO ERROR : Unknown asymmetric matrix solver PCG
Valid asymmetric matrix solvers are :
3

```

```

(
PBiCG
smoothSolver
GAMG
)

```

4.5.1.1 Solution tolerances

The matrices are sparse, meaning they predominately include coefficients of 0, in segregated, decoupled, finite volume numerics. Consequently, the solvers are generally iterative, *i.e.* they are based on reducing the equation residual over successive solutions. The residual is ostensibly a measure of the error in the solution so that the smaller it is, the more accurate the solution. More precisely, the residual is evaluated by substituting the current solution into the equation and taking the magnitude of the difference between the left and right hand sides; it is also normalised to make it independent of the scale of the problem being analysed.

Before solving an equation for a particular field, the initial residual is evaluated based on the current values of the field. After each solver iteration the residual is re-evaluated. The solver stops if *any one* of the following conditions are reached:

- the residual falls below the *solver tolerance*, **tolerance**;
- the ratio of current to initial residuals falls below the *solver relative tolerance*, **relTol**;
- the number of iterations exceeds a *maximum number of iterations*, **maxIter**;

The solver tolerance should represent the level at which the residual is small enough that the solution can be deemed sufficiently accurate. The solver relative tolerance limits the relative improvement from initial to final solution. In transient simulations, it is usual to set the solver relative tolerance to 0 to force the solution to converge to the solver tolerance in each time step. The tolerances, **tolerance** and **relTol** must be specified in the dictionaries for all solvers; **maxIter** is optional and defaults to a value of 1000.

Equations are very often solved multiple times within one solution step, or time step. For example, when using the PISO algorithm, a pressure equation is solved according to the number specified by **nCorrectors**, as described in section 4.5.3. Where this occurs, the solver is very often set up to use different settings when solving the particular equation for the final time, specified by a keyword that adds **Final** to the field name. For example, in the **cavity** tutorial in section 2.1, the solver settings for pressure are as follows.

```

p
{
    solver          PCG;
    preconditioner DIC;
    tolerance       1e-06;
    relTol         0.05;
}

pFinal
{

```

```

$p;
relTol      0;
}

```

If the case is specified to solve pressure 4 times within one time step, then the first 3 solutions would use the settings for `p` with `relTol` of 0.05, so that the cost of solving each equation is relatively low. Only when the equation is solved the final (4th) time, it solves to a residual level specified by `tolerance` (since `relTol` is 0, effectively deactivating it) for greater accuracy, but at greater cost.

4.5.1.2 Preconditioned conjugate gradient solvers

There are a range of options for preconditioning of matrices in the conjugate gradient solvers, represented by the `preconditioner` keyword in the solver dictionary, listed below. Note that the `DIC/DILU` preconditioners are exclusively specified in the tutorials in OpenFOAM.

- `DIC/DILU`: diagonal incomplete-Cholesky (symmetric) and incomplete-LU (asymmetric)
- `FDIC`: faster diagonal incomplete-Cholesky (`DIC` with caching, symmetric)
- `diagonal`: diagonal preconditioning.
- `GAMG`: geometric-algebraic multi-grid.
- `none`: no preconditioning.

4.5.1.3 Smooth solvers

The solvers that use a smoother require the choice of smoother to be specified. The smoother options are listed below. The `symGaussSeidel` and `GaussSeidel` smoothers are preferred in the tutorials.

- `GaussSeidel`: Gauss-Seidel.
- `symGaussSeidel`: symmetric Gauss-Seidel.
- `DIC/DILU`: diagonal incomplete-Cholesky (symmetric), incomplete-LU (asymmetric).
- `DICGaussSeidel`: diagonal incomplete-Cholesky/LU with Gauss-Seidel (symmetric/asymmetric).

When using the smooth solvers, the user can optionally specify the number of sweeps, by the `nSweeps` keyword, before the residual is recalculated. Without setting it, it reverts to a default value of 1.

4.5.1.4 Geometric-algebraic multi-grid solvers

The generalised method of geometric-algebraic multi-grid (GAMG) uses the principle of: generating a quick solution on a mesh with a small number of cells; mapping this solution onto a finer mesh; using it as an initial guess to obtain an accurate solution on the fine mesh. GAMG is faster than standard methods when the increase in speed by solving first on coarser meshes outweighs the additional costs of mesh refinement and mapping of field data. In practice, GAMG starts with the mesh specified by the user and coarsens/refines the mesh in stages. The user is only required to specify an approximate mesh size at the most coarse level in terms of the number of cells

The agglomeration of cells is performed by the method specified by the `agglomerator` keyword. The tutorials all use the default `faceAreaPair` method, although the `MGridGen` option is an alternative method that requires an additional entry specifying the shared object library for `MGridGen`:

```
geometricGamAgglomerationLibs ("libMGridGenGamAgglomeration.so");
```

The agglomeration can be controlled using the following optional entries, most of which default in the tutorials.

- `cacheAgglomeration`: switch specifying caching of the agglomeration strategy (default `true`).
- `nCellsInCoarsestLevel`: approximate mesh size at the most coarse level in terms of the number of cells (default 10).
- `directSolveCoarsen`: use a direct solver at the coarsest level (default `false`).
- `mergeLevels`: keyword controls the speed at which coarsening or refinement is performed; the default is 1, which is safest, but for simple meshes, the solution speed can be increased by coarsening/refining 2 levels at a time, *i.e.* setting `mergeLevels` 2.

Smoothing is specified by the `smoother` as described in section 4.5.1.3. The number of sweeps used by the smoother at different levels of mesh density are specified by the following optional entries.

- `nPreSweeps`: number of sweeps as the algorithm is coarsening (default 0).
- `preSweepsLevelMultiplier`: multiplier for the the number of sweeps between each coarsening level (default 1).
- `maxPreSweeps`: maximum number of sweeps as the algorithm is coarsening (default 4).
- `nPostSweeps`: number of sweeps as the algorithm is refining (default 2).
- `postSweepsLevelMultiplier`: multiplier for the the number of sweeps between each refinement level (default 1).
- `maxPostSweeps`: maximum number of sweeps as the algorithm is refining (default 4).
- `nFinestSweeps`: number of sweeps at finest level (default 2).

4.5.2 Solution under-relaxation

A second sub-dictionary of *fvSolution* that is often used in OpenFOAM is *relaxationFactors* which controls under-relaxation, a technique used for improving stability of a computation, particularly in solving steady-state problems. Under-relaxation works by limiting the amount which a variable changes from one iteration to the next, either by modifying the solution matrix and source prior to solving for a field or by modifying the field directly. An under-relaxation factor $\alpha, 0 < \alpha \leq 1$ specifies the amount of under-relaxation, as described below.

- No specified α : no under-relaxation.
- $\alpha = 1$: guaranteed matrix diagonal equality/dominance.
- α decreases, under-relaxation increases.
- $\alpha = 0$: solution does not change with successive iterations.

An optimum choice of α is one that is small enough to ensure stable computation but large enough to move the iterative process forward quickly; values of α as high as 0.9 can ensure stability in some cases and anything much below, say, 0.2 are prohibitively restrictive in slowing the iterative process.

Relaxation factors for under-relaxation of fields are specified within a *field* sub-dictionary; relaxation factors for equation under-relaxation are within a *equations* sub-dictionary. An example is shown below from tutorial example of *simpleFoam*, showing typical settings for an incompressible steady-state solver. The factors are specified for pressure *p*, pressure *U*, and turbulent fields grouped using a regular expression.

```

54
55     relaxationFactors
56     {
57         fields
58         {
59             p           0.3;
60         }
61         equations
62         {
63             U           0.7;
64             "(k|omega|epsilon).*" 0.7;
65         }
66     }
67 // ****
68

```

Another example for *pimpleFoam*, a transient incompressible solver, just uses under-relaxation to ensure matrix diagonal equality, typical of transient simulations.

```

60
61     relaxationFactors
62     {
63         equations
64         {
65             "U.*"          1;
66             "k.*"          1;
67             "epsilon.*"    1;
68         }
69 // ****

```

4.5.3 PISO, SIMPLE and PIMPLE algorithms

Most fluid dynamics solver applications in OpenFOAM use either the pressure-implicit split-operator (PISO), the semi-implicit method for pressure-linked equations (SIMPLE) algorithms, or a combined PIMPLE algorithm. These algorithms are iterative procedures for coupling equations for momentum and mass conservation, PISO and PIMPLE being used for transient problems and SIMPLE for steady-state.

Within in time, or solution, step, both algorithms solve a pressure equation, to enforce mass conservation, with an explicit correction to velocity to satisfy momentum conservation. They optionally begin each step by solving the momentum equation — the so-called momentum predictor.

While all the algorithms solve the same governing equations (albeit in different forms), the algorithms principally differ in how they loop over the equations. The looping is controlled by input parameters that are listed below. They are set in a dictionary named after the algorithm, *i.e.* *SIMPLE*, *PISO* or *PIMPLE*.

- **nCorrectors:** used by PISO, and PIMPLE, sets the number of times the algorithm solves the pressure equation and momentum corrector in each step; typically set to 2 or 3.
- **nNonOrthogonalCorrectors:** used by all algorithms, specifies repeated solutions of the pressure equation, used to update the explicit non-orthogonal correction, described in section 4.4.4, of the Laplacian term $\nabla \cdot ((1/A)\nabla p)$; typically set to 0 (particularly for steady-state) or 1.
- **nOuterCorrectors:** used by PIMPLE, it enables looping over the entire system of equations within one time step, representing the total number of times the system is solved; must be ≥ 1 and is typically set to 1, replicating the PISO algorithm.
- **momentumPredictor:** switch that controls solving of the momentum predictor; typically set to `off` for some flows, including low Reynolds number and multiphase

4.5.4 Pressure referencing

In a closed incompressible system, pressure is relative: it is the pressure range that matters not the absolute values. In these cases, the solver sets a reference level of `pRefValue` in cell `pRefCell`. These entries are generally stored in the *SIMPLE*, *PISO* or *PIMPLE* sub-dictionary and are used by those solvers that require them when the case demands it.

4.5.5 Other parameters

The `fvSolutions` dictionaries in the majority of standard OpenFOAM solver applications contain no other entries than those described so far in this section. However, in general the `fvSolution` dictionary may contain any parameters to control the solvers, algorithms, or in fact anything. If any parameter or sub-dictionary is missing when a solver is run, it will terminate, printing a detailed error message. The user can then add missing parameters accordingly.

4.6 Case management tools

There are a set of applications and scripts that help with managing case files and help the user find and set keyword data entries in case files. The tools are described in the following sections.

4.6.1 File management scripts

The following tools help manage case files.

`foamListTimes` lists the time directories for a case, omitting the `0` directory by default; the `-rm` option deletes the listed time directories, so that a case can be cleaned of time directories with results by the following command.

```
foamListTimes -rm
```

`foamCloneCase` creates a new case, by copying the `0`, `system` and `constant` directories from an existing case; executed simply by the following command, where `oldCase` refers to an existing case directory.

```
foamCloneCase oldCase newCase
```

`foamCleanPolyMesh` deletes mesh files for a case; useful to execute before regenerating a mesh, particularly with `snappyHexMesh` which generates refinement history and other files that need removing when re-meshing.

4.6.2 The foamDictionary tool

The `foamDictionary` utility offer several options for writing, editing and adding keyword entries in case files. The utility is executed with an OpenFOAM case dictionary file as an argument, *e.g.* from within a case directory on the `fvSchemes` file.

```
foamDictionary system/fvSchemes
```

Without options, the utility lists all the keyword entries in the file, *e.g.* as follows for the `fvSchemes` file in the pitzDaily tutorial case for `simpleFoam`.

```
{
FoamFile
{
    version      2;
    format       ascii;
    class        dictionary;
    location     "system";
    object       fvSchemes;
}
ddtSchemes
{
    default      steadyState;
}
gradSchemes
{
    default      Gauss linear;
}
divSchemes
{
    default      none;
    div(phi,U)   bounded Gauss linearUpwind grad(U);
```

```

        div(phi,k)      bounded Gauss limitedLinear 1;
        div(phi,epsilon) bounded Gauss limitedLinear 1;
        div(phi,omega)   bounded Gauss limitedLinear 1;
        div(phi,v2)     bounded Gauss limitedLinear 1;
        div((nuEff*dev2(T(grad(U))))) Gauss linear;
        div(nonlinearStress) Gauss linear;
    }
    laplacianSchemes
    {
        default          Gauss linear corrected;
    }
    interpolationSchemes
    {
        default          linear;
    }
    snGradSchemes
    {
        default          corrected;
    }
    wallDist
    {
        method          meshWave;
    }
}

```

The **-entry** option allows the user to print the entry for a particular keyword, *e.g.* **divSchemes** in the example below

```
foamDictionary -entry divSchemes system/fvSchemes
```

The example clearly extracts the **divSchemes** dictionary.

```

divSchemes
{
    default      none;
    div(phi,U)   bounded Gauss linearUpwind grad(U);
    div(phi,k)   bounded Gauss limitedLinear 1;
    div(phi,epsilon) bounded Gauss limitedLinear 1;
    div(phi,omega)   bounded Gauss limitedLinear 1;
    div(phi,v2)     bounded Gauss limitedLinear 1;
    div((nuEff*dev2(T(grad(U))))) Gauss linear;
    div(nonlinearStress) Gauss linear;
}

```

The “.” syntax allows access to keywords with levels of sub-dictionary. For example, the **div(phi,U)** keyword can be accessed within the **divSchemes** sub-dictionary by the following command.

```
foamDictionary -entry "divSchemes.div(phi,U)" system/fvSchemes
```

The example returns the single **divSchemes.div(phi,U)** entry.

```
div(phi,U)   bounded Gauss linearUpwind grad(U);
```

The **-value** option causes only the entry to be written.

```
foamDictionary -entry "divSchemes.div(phi,U)" -value system/fvSchemes
```

The example removes the keyword and terminating semicolon, leaving just the data.

```
bounded Gauss linearUpwind grad(U)
```

The `-keywords` option causes only the keywords to be written.

```
foamDictionary -entry divSchemes -keywords system/fvSchemes
```

The example produces a list of keywords inside the `divSchemes` dictionary.

```
default
div(phi,U)
div(phi,k)
div(phi,epsilon)
div(phi,omega)
div(phi,v2)
div((nuEff*dev2(T(grad(U)))))
```

The example removes the keyword and terminating semicolon, leaving just the data.

```
bounded Gauss linearUpwind grad(U)
```

`foamDictionary` can set entries with the `-set` option. If the user wishes to change the `div(phi,U)` to the upwind scheme, they can enter the following.

```
foamDictionary -entry "divSchemes.div(phi,U)" \
-set "bounded Gauss upwind" system/fvSchemes
```

`foamDictionary` can add entries with the `-add` option. If the user wishes to add an entry named `turbulence` to `divSchemes` with the upwind scheme, they can enter the following.

```
foamDictionary -entry "divSchemes.turbulence" \
-add "bounded Gauss upwind" system/fvSchemes
```

The `foamSearch` script, demonstrated extensively in section 4.4, uses `foamDictionary` functionality to extract and sort keyword entries from all files of a specified name in a specified dictionary. The `-c` option counts the number of entries of each type, *e.g.* the user could search for the choice of `solver` for the `p` equation in all the `fvSolution` files in the tutorials.

```
foamSearch -c $FOAM_TUTORIALS fvSolution solvers.p.solver
```

The search shows `GAMG` to be the most common choice in all the tutorials.

```
59 solver      GAMG;
 3 solver      PBiCG;
18 solver      PCG;
 5 solver      smoothSolver;
```

Chapter 5

Mesh generation and conversion

This chapter describes all topics relating to the creation of meshes in OpenFOAM: section 5.1 gives an overview of the ways a mesh may be described in OpenFOAM; section 5.3 covers the `blockMesh` utility for generating simple meshes of blocks of hexahedral cells; section 5.4 covers the `snappyHexMesh` utility for generating complex meshes of hexahedral and split-hexahedral cells automatically from triangulated surface geometries; section 5.5 describes the options available for conversion of a mesh that has been generated by a third-party product into a format that OpenFOAM can read.

5.1 Mesh description

This section provides a specification of the way the OpenFOAM C++ classes handle a mesh. The mesh is an integral part of the numerical solution and must satisfy certain criteria to ensure a valid, and hence accurate, solution. During any run, OpenFOAM checks that the mesh satisfies a fairly stringent set of validity constraints and will cease running if the constraints are not satisfied.

By default OpenFOAM defines a mesh of arbitrary polyhedral cells in 3-D, bounded by arbitrary polygonal faces, *i.e.* the cells can have an unlimited number of faces where, for each face, there is no limit on the number of edges nor any restriction on its alignment. A mesh with this general structure is known in OpenFOAM as a `polyMesh`. This type of mesh offers great freedom in mesh generation and manipulation in particular when the geometry of the domain is complex or changes over time.

5.1.1 Mesh specification and validity constraints

Before describing the OpenFOAM mesh format, we will first set out the validity constraints used in OpenFOAM. The conditions that a mesh must satisfy are:

5.1.1.1 Points

A point is a location in 3-D space, defined by a vector in units of metres (m). The points are compiled into a list and each point is referred to by a label, which represents its position in the list, starting from zero. *The point list cannot contain two different points at an exactly identical position nor any point that is not part at least one face.*

5.1.1.2 Faces

A face is an ordered list of points, where a point is referred to by its label. The ordering of point labels in a face is such that each two neighbouring points are connected by an edge, *i.e.* you follow points as you travel around the circumference of the face. Faces are compiled into a list and each face is referred to by its label, representing its position in the list. The direction of the face normal vector is defined by the right-hand rule, *i.e.* looking towards a face, if the numbering of the points follows an anti-clockwise path, the normal vector points towards you, as shown in Figure 5.1.

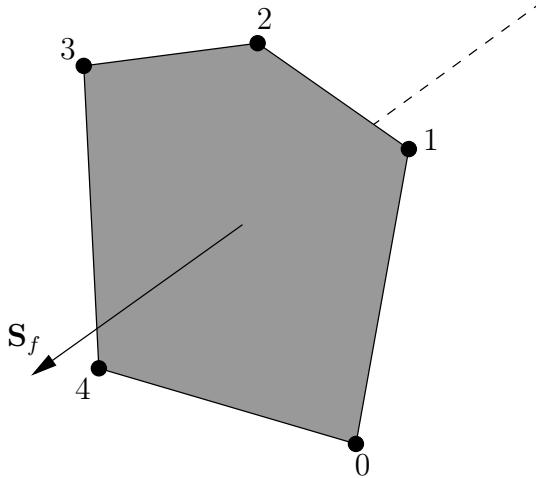


Figure 5.1: Face area vector from point numbering on the face

There are two types of face:

Internal faces Those faces that connect two cells (and it can never be more than two).

For each internal face, the ordering of the point labels is such that the face normal points into the cell with the larger label, *i.e.* for cells 2 and 5, the normal points into 5;

Boundary faces Those belonging to one cell since they coincide with the boundary of the domain. A boundary face is therefore addressed by one cell(only) and a boundary patch. The ordering of the point labels is such that the face normal points outside of the computational domain.

Faces are generally expected to be convex; at the very least the face centre needs to be inside the face. Faces are allowed to be warped, *i.e.* not all points of the face need to be coplanar.

5.1.1.3 Cells

A cell is a list of faces in arbitrary order. Cells must have the properties listed below.

Contiguous The cells must completely cover the computational domain and must not overlap one another.

Convex Every cell must be convex and its cell centre inside the cell.

Closed Every cell must be *closed*, both geometrically and topologically where:

- geometrical closedness requires that when all face area vectors are oriented to point outwards of the cell, their sum should equal the zero vector to machine accuracy;
- topological closedness requires that all the edges in a cell are used by exactly two faces of the cell in question.

5.1.1.4 Boundary

A boundary is a list of patches, each of which is associated with a boundary condition. A patch is a list of face labels which clearly must contain only boundary faces and no internal faces. The boundary is required to be closed, *i.e.* the sum all boundary face area vectors equates to zero to machine tolerance.

5.1.2 The polyMesh description

The *constant* directory contains a full description of the case **polyMesh** in a subdirectory **polyMesh**. The **polyMesh** description is based around faces and, as already discussed, internal faces connect 2 cells and boundary faces address a cell and a boundary patch. Each face is therefore assigned an ‘owner’ cell and ‘neighbour’ cell so that the connectivity across a given face can simply be described by the owner and neighbour cell labels. In the case of boundaries, the connected cell is the owner and the neighbour is assigned the label ‘-1’. With this in mind, the I/O specification consists of the following files:

points a list of vectors describing the cell vertices, where the first vector in the list represents vertex 0, the second vector represents vertex 1, *etc.*;

faces a list of faces, each face being a list of indices to vertices in the points list, where again, the first entry in the list represents face 0, *etc.*;

owner a list of owner cell labels, the index of entry relating directly to the index of the face, so that the first entry in the list is the owner label for face 0, the second entry is the owner label for face 1, *etc.*;

neighbour a list of neighbour cell labels;

boundary a list of patches, containing a dictionary entry for each patch, declared using the patch name, *e.g.*

```
movingWall
{
    type patch;
    nFaces 20;
    startFace 760;
}
```

The **startFace** is the index into the face list of the first face in the patch, and **nFaces** is the number of faces in the patch.

5.1.3 Cell shapes

While OpenFOAM supports any shapes of cell, other tools and software generally do not. Therefore conversion of meshes to and from OpenFOAM format often requires the use of defined cell shapes, such as tetrahedra, hexahedra, *etc.* The functionality is available in a `cellShape` class that uses a set of shapes defined in a `cellModels` file in the `$FOAM_ETCH` directory.

Cells of a given shape are then defined by the ordering of point labels in accordance with the numbering scheme contained in the shape model. For reference, the ordering schemes for points, faces and edges are shown in Table 5.1. Any cell description then consists of two parts: the name of a cell model and the ordered list of labels. Thus, using the following list of points

```
(  
    (0 0 0)  
    (1 0 0)  
    (1 1 0)  
    (0 1 0)  
    (0 0 0.5)  
    (1 0 0.5)  
    (1 1 0.5)  
    (0 1 0.5)  
)
```

A hexahedral cell would be written as follows using the keyword `hex`.

```
(hex 8(0 1 2 3 4 5 6 7))
```

This forms the basis for the input syntax for the `blockMesh` mesh generator, described in section 5.3.

5.1.4 1- and 2-dimensional and axi-symmetric problems

OpenFOAM is designed as a code for 3-dimensional space and defines all meshes as such. However, 1- and 2- dimensional and axi-symmetric problems can be simulated in OpenFOAM by generating a mesh in 3 dimensions and applying special boundary conditions on any patch in the plane(s) normal to the direction(s) of interest. More specifically, 1- and 2- dimensional problems use the `empty` patch type and axi-symmetric problems use the `wedge` type. The use of both are described in section 5.2.2 and the generation of wedge geometries for axi-symmetric problems is discussed in section 5.3.5.

5.2 Boundaries

In this section we discuss the way in which boundaries are treated in OpenFOAM. The subject of boundaries is quite complex because their role in modelling is not simply that of a geometric entity but an integral part of the solution and numerics through boundary conditions or inter-boundary ‘connections’. A discussion of boundaries sits uncomfortably between a discussion on meshes, fields, discretisation, computational processing *etc.*

We first need to consider that, for the purpose of applying boundary conditions, a boundary is generally broken up into a set of *patches*. One patch may include one or more enclosed

Cell type	Keyword	Vertex numbering	Face numbering	Edge numbering
Hexahedron	hex			
Wedge	wedge			
Prism	prism			
Pyramid	pyr			
Tetrahedron	tet			
Tet-wedge	tetWedge			

Table 5.1: Vertex, face and edge numbering for cellShapes.

areas of the boundary surface which do not necessarily need to be physically connected. A `type` is assigned to every patch as part of the mesh description, as part of the `boundary` file described in section 5.1.2. It describes the type of patch in terms of geometry or a data ‘communication link’. An example `boundary` file is shown below for a `sonicFoam` case. A `type` entry is clearly included for every patch (`inlet`, `outlet`, *etc.*), with types assigned that include `patch`, `symmetryPlane` and `empty`.

```

17   6
18   (
19     inlet
20     {
21       type          patch;
22       nFaces       50;
23       startFace    10325;
24     }
25     outlet
26     {
27       type          patch;
28       nFaces       40;
29       startFace    10375;
30     }
31     bottom
32     {
33       type          symmetryPlane;
34       inGroups     1(symmetryPlane);
35       nFaces       25;
36       startFace    10415;
37     }
38     top
39     {
40       type          symmetryPlane;
41       inGroups     1(symmetryPlane);
42       nFaces       125;
43       startFace    10440;
44     }
45     obstacle
46     {
47       type          patch;
48       nFaces       110;
49       startFace    10565;
50     }
51     defaultFaces
52     {
53       type          empty;
54       inGroups     1(empty);
55       nFaces       10500;
56       startFace    10675;
57     }
58   )
59   // ****
60
61 // ****

```

The user can scan the tutorials for mesh generation configuration files, *e.g.* `blockMeshDict` for `blockMesh` (see section 5.3) and `snappyHexMeshDict` for `snappyHexMesh` (see section 5.4, for examples of different types being used). The following example searches for `blockMeshDict` files that specify the `symmetryPlane` condition.

```
find $FOAM_TUTORIALS -name blockMeshDict | \
xargs grep -El "type[\t]*symmetryPlane"
```

The next example searches for `snappyHexMeshDict` files that specify the `wall` condition.

```
find $FOAM_TUTORIALS -name snappyHexMeshDict | \
xargs grep -El "type[\t]*wall"
```

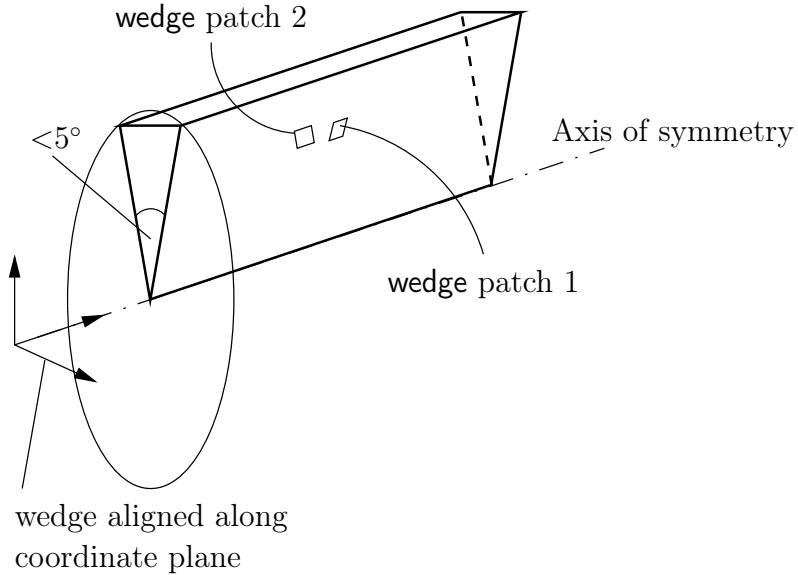


Figure 5.2: Axi-symmetric geometry using the wedge patch type.

5.2.1 Geometric (constraint) patch types

The main geometric types available in OpenFOAM are summarised below. This is not a complete list; for all types see `$FOAM_SRC/finiteVolume/fields/fvPatchFields/constraint`.

- **patch**: generic type containing no geometric or topological information about the mesh, *e.g.* used for an inlet or an outlet.
- **wall**: for patch that coincides with a solid wall, required for some physical modelling, *e.g.* wall functions in turbulence modelling.
- **symmetryPlane**: for a planar patch which is a symmetry plane.
- **symmetry**: for any (non-planar) patch which uses the symmetry plane (slip) condition.
- **empty**: for solutions in 2 (or 1) dimensions (2D/1D), the type used on each patch whose plane is normal to the 3rd (and 2nd) dimension for which no solution is required.
- **wedge**: for 2 dimensional axi-symmetric cases, *e.g.* a cylinder, the geometry is specified as a wedge of small angle (*e.g.* 1°) and 1 cell thick, running along the centre line, straddling one of the coordinate planes, as shown in Figure 5.2; the axi-symmetric wedge planes must be specified as separate patches of `wedge` type.
- **cyclic**: enables two patches to be treated as if they are physically connected; used for repeated geometries; one `cyclic` patch is linked to another through a `neighbourPatch` keyword in the `boundary` file; each pair of connecting faces must have similar area to within a tolerance given by the `matchTolerance` keyword in the `boundary` file.
- **cyclicAMI**: like `cyclic`, but for 2 patches whose faces are non matching; used for sliding interface in rotating geometry cases.
- **processor**: the type that describes inter-processor boundaries for meshes decomposed for parallel running.

5.2.2 Basic boundary conditions

Boundary conditions are specified in field files, *e.g.* p , U , in time directories as described in section 4.2.8. An example pressure field file, p , is shown below for the `sonicFoam` case corresponding to the `boundary` file presented in section 5.2.1.

```

17 dimensions      [1 -1 -2 0 0 0];
18 internalField   uniform 1;
20
21 boundaryField
22 {
23     inlet
24     {
25         type          fixedValue;
26         value         uniform 1;
27     }
28
29     outlet
30     {
31         type          waveTransmissive;
32         field         p;
33         psi           thermo:psi;
34         gamma         1.4;
35         fieldInf     1;
36         lInf          3;
37         value         uniform 1;
38     }
39
40     bottom
41     {
42         type          symmetryPlane;
43     }
44
45     top
46     {
47         type          symmetryPlane;
48     }
49
50     obstacle
51     {
52         type          zeroGradient;
53     }
54
55     defaultFaces
56     {
57         type          empty;
58     }
59 }
60 // ****
61

```

Every patch includes a `type` entry that specifies the type of boundary condition. They range from a basic `fixedValue` condition applied to the `inlet`, to a complex `waveTransmissive` condition applied to the `outlet`. The patches with non-generic types, *e.g.* `symmetryPlane`, defined in `boundary`, use consistent boundary condition types in the `p` file.

The main basic boundary condition types available in OpenFOAM are summarised below using a patch field named `Q`. This is not a complete list; for all types see `$FOAM_SRC/finiteVolume/fields/fvPatchFields/basic`.

- `fixedValue`: value of `Q` is specified by `value`.
- `fixedGradient`: normal gradient of `Q` ($\partial \mathbf{Q} / \partial n$) is specified by `gradient`.
- `zeroGradient`: normal gradient of `Q` is zero.
- `calculated`: patch field `Q` calculated from other patch fields.

- mixed: mixed `fixedValue/ fixedGradient` condition depending on `valueFraction` ($0 \leq \text{valueFraction} \leq 1$) where

$$\text{valueFraction} = \begin{cases} 1 & \text{corresponds to } \mathbf{Q} = \text{refValue}, \\ 0 & \text{corresponds to } \partial\mathbf{Q}/\partial n = \text{refGradient}. \end{cases} \quad (5.1)$$

- directionMixed: mixed condition with tensorial `valueFraction`, to allow different conditions in normal and tangential directions of a vector patch field, e.g. `fixedValue` in the tangential direction, `zeroGradient` in the normal direction.

5.2.3 Derived types

There are numerous more complex boundary conditions derived from the basic conditions. For example, many complex conditions are derived from `fixedValue`, where the value is calculated by a function of other patch fields, time, geometric information, etc. Some other conditions derived from `mixed/directionMixed` switch between `fixedValue` and `fixedGradient` (usually `zeroGradient`).

There are a number of ways the user can list the available boundary conditions in OpenFOAM, with the `-listScalarBCs` and `-listVectorBCs` utility being the quickest. The boundary conditions for scalar fields and vector fields, respectively, can be listed for a given solver, e.g. `simpleFoam`, as follows.

```
simpleFoam -listScalarBCs -listVectorBCs
```

These produce long lists which the user can scan through. If the user wants more information of a particular condition, they can try searching for examples in the tutorials, or look to the source code, where information is usually provided in the `Description`. For example, for the `totalPressure` boundary condition, examples in the tutorials can be found by the following.

```
find $FOAM_TUTORIALS -type f | xargs grep -l totalPressure
```

The source code files can be located by the following.

```
find $FOAM_SRC -name totalPressure
```

In the following sections we will highlight some particularly important, commonly used boundary conditions.

5.2.3.1 The inlet/outlet condition

The `inletOutlet` condition is one derived from `mixed`, which switches between `zeroGradient` when the fluid flows out of the domain at a patch face, and `fixedValue`, when the fluid is flowing into the domain. For inflow, the inlet value is specified by an `inletValue` entry. A good example of its use can be seen in the `damBreak` tutorial, where it is applied to the phase fraction on the upper `atmosphere` boundary. Where there is outflow, the condition is well posed, where there is inflow, the phase fraction is fixed with a value of 0, corresponding to 100% air.

```

17 dimensions      [0 0 0 0 0 0];
18 internalField   uniform 0;
20 boundaryField
21 {
22     leftWall
23     {
24         type          zeroGradient;
25     }
26
27     rightWall
28     {
29         type          zeroGradient;
30     }
31
32     lowerWall
33     {
34         type          zeroGradient;
35     }
36
37     atmosphere
38     {
39         type          inletOutlet;
40         inletValue    uniform 0;
41         value        uniform 0;
42     }
43
44     defaultFaces
45     {
46         type          empty;
47     }
48 }
49
50 // ****
51

```

5.2.3.2 Entrainment boundary conditions

The combination of the `totalPressure` condition on pressure and `pressureInletOutletVelocity` on velocity is extremely common for patches where some inflow occurs and the inlet flow velocity is not known. That includes the `atmosphere` boundary in the `damBreak` tutorial, inlet conditions where only pressure is known, outlets where flow reversal may occur, and where flow is entrained, *e.g.* on boundaries surrounding a jet through a nozzle.

The `totalPressure` condition specifies

$$p = \begin{cases} p_0 & \text{for outflow} \\ p_0 - \frac{1}{2}|\mathbf{U}^2| & \text{for inflow (incompressible, subsonic)} \end{cases} \quad (5.2)$$

where the user specifies p_0 through the `p0` keyword. The `pressureInletOutletVelocity` condition specifies `zeroGradient` at all times, except on the tangential component which is set to `fixedValue` for inflow, with the `tangentialVelocity` defaulting to 0.

The idea behind this combination is that the condition is a standard combination in the case of outflow, but for inflow the normal velocity is allowed to find its own value. Under these circumstances, a rapid rise in velocity presents a risk of instability, but the rise is moderated by the reduction of inlet pressure, and hence driving pressure gradient, as the inflow velocity increases.

The specification of these boundary conditions in the `U` and `p_rgh` files, in the `damBreak` case, are shown below.

```

17 dimensions      [0 1 -1 0 0 0];
18 internalField   uniform (0 0 0);
20 boundaryField
21

```

```

22 {
23     leftWall
24     {
25         type          noSlip;
26     }
27     rightWall
28     {
29         type          noSlip;
30     }
31     lowerWall
32     {
33         type          noSlip;
34     }
35     atmosphere
36     {
37         type          pressureInletOutletVelocity;
38         value         uniform (0 0 0);
39     }
40     defaultFaces
41     {
42         type          empty;
43     }
44 }
45 // ****
46

17 dimensions      [1 -1 -2 0 0 0];
18 internalField   uniform 0;
19 boundaryField
20 {
21     leftWall
22     {
23         type          fixedFluxPressure;
24         value         uniform 0;
25     }
26     rightWall
27     {
28         type          fixedFluxPressure;
29         value         uniform 0;
30     }
31     lowerWall
32     {
33         type          fixedFluxPressure;
34         value         uniform 0;
35     }
36     atmosphere
37     {
38         type          totalPressure;
39         p0           uniform 0;
40     }
41     defaultFaces
42     {
43         type          empty;
44     }
45 }
46 // ****
47
48
49
50
51
52
53 // ****

```

5.2.3.3 Fixed flux pressure

In the above example, it can be seen that all the wall boundaries use a boundary condition named `fixedFluxPressure`. This boundary condition is used for pressure in situations where `zeroGradient` is generally used, but where body forces such as gravity and surface tension are present in the solution equations. The condition adjusts the gradient accordingly.

5.2.3.4 Time-varying boundary conditions

There are several boundary conditions for which some input parameters are specified by a function of time (using `Function1` functionality) class. They can be searched by the following command.

```
find $FOAM_SRC/finiteVolume/fields/fvPatchFields -type f | \
xargs grep -l Function1 | xargs dirname | sort -u
```

They include conditions such as `uniformFixedValue`, which is a `fixedValue` condition which applies a single value which is a function of time through a `uniformValue` keyword entry.

The `Function1` is specified by a keyword following the `uniformValue` entry, followed by parameters that relate to the particular function. The `Function1` options are listed below.

- `constant`: constant value.
- `table`: inline list of (`time value`) pairs; interpolates values linearly between times.
- `tableFile`: as above, but with data supplied in a separate file.
- `csvFile`: time-value data supplied in a file in CSV format.
- `square`: square-wave function.
- `sine`: sine function.
- `one` and `zero`: constant one and zero values.
- `polynomial`: polynomial function using a list (`coeff exponent`) pairs.
- `scale`: scales a given `value` function by a scalar `scale` function; both entries can be themselves `Function1`; `scale` function is often a ramp function (below), with `value` controlling the ramp value.
- `linearRamp`, `quadraticRamp`, `halfCosineRamp`, `quarterCosineRamp` and `quarterSineRamp`: monotonic ramp functions which ramp from 0 to 1 over specified duration.

Examples of a time-varying inlet for a scalar are shown below.

```
inlet
{
    type      uniformFixedValue;
    uniformValue constant 2;
}

inlet
{
    type      uniformFixedValue;
    uniformValue table ((0 0) (10 2));
}

inlet
{
    type      uniformFixedValue;
    uniformValue tableFile;
    uniformValueCoeffs
    {
        fileName      "dataTable.txt";
    }
}
```

```

}

inlet
{
    type      uniformFixedValue;
    uniformValue csvFile;
    uniformValueCoeffs
    {
        nHeaderLine      4;           // number of header lines
        refColumn        0;           // time column index
        componentColumns (1);       // data column index
        separator        ",";         // optional (defaults to ",")
        mergeSeparators no;          // merge multiple separators
        fileName         "dataTable.csv";
    }
}

inlet
{
    type      uniformFixedValue;
    uniformValue square;
    uniformValueCoeffs
    {
        frequency     10;
        amplitude     1;
        scale         2; // Scale factor for wave
        level         1; // Offset
    }
}

inlet
{
    type      uniformFixedValue;
    uniformValue sine;
    uniformValueCoeffs
    {
        frequency     10;
        amplitude     1;
        scale         2; // Scale factor for wave
        level         1; // Offset
    }
}

inlet
{
    type      uniformFixedValue;
    uniformValue polynomial ((1 0) (2 2)); // = 1*t^0 + 2*t^2
}

input // ramp from 0 -> 2, from t = 0 -> 0.4
{
    type      uniformFixedValue;
    uniformValue scale;
    scale
    {
        type      linearRamp;
        start    0;
        duration 0.4;
    }
    value     2;
}

```

5.3 Mesh generation with the `blockMesh` utility

This section describes the mesh generation utility, `blockMesh`, supplied with OpenFOAM. The `blockMesh` utility creates parametric meshes with grading and curved edges.

The mesh is generated from a dictionary file named `blockMeshDict` located in the `system`

(or *constant/polyMesh*) directory of a case. **blockMesh** reads this dictionary, generates the mesh and writes out the mesh data to *points* and *faces*, *cells* and *boundary* files in the same directory.

The principle behind **blockMesh** is to decompose the domain geometry into a set of 1 or more three dimensional, hexahedral blocks. Edges of the blocks can be straight lines, arcs or splines. The mesh is ostensibly specified as a number of cells in each direction of the block, sufficient information for **blockMesh** to generate the mesh data.

Each block of the geometry is defined by 8 vertices, one at each corner of a hexahedron. The vertices are written in a list so that each vertex can be accessed using its label, remembering that OpenFOAM always uses the C++ convention that the first element of the list has label ‘0’. An example block is shown in Figure 5.3 with each vertex numbered according to the list. The edge connecting vertices 1 and 5 is curved to remind the reader that curved edges can be specified in **blockMesh**.

It is possible to generate blocks with less than 8 vertices by collapsing one or more pairs of vertices on top of each other, as described in section 5.3.5.

Each block has a local coordinate system (x_1, x_2, x_3) that must be right-handed. 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.

The local coordinate system is defined by the order in which the vertices are presented in the block definition according to:

- the axis origin is the first entry in the block definition, vertex 0 in our example;
- the x_1 direction is described by moving from vertex 0 to vertex 1;
- the x_2 direction is described by moving from vertex 1 to vertex 2;
- vertices 0, 1, 2, 3 define the plane $x_3 = 0$;
- vertex 4 is found by moving from vertex 0 in the x_3 direction;
- vertices 5,6 and 7 are similarly found by moving in the x_3 direction from vertices 1,2 and 3 respectively.

5.3.1 Writing a *blockMeshDict* file

The *blockMeshDict* file is a dictionary using keywords described below.

- **convertToMeters**: scaling factor for the vertex coordinates, e.g. 0.001 scales to mm.
- **vertices**: list of vertex coordinates, see section 5.3.1.1.
- **edges**: used to describe curved geometry, see section 5.3.1.2.
- **block**: ordered list of vertex labels and mesh size, see section 5.3.1.3.
- **boundary**: sub-dictionary of boundary patches, see section 5.3.1.5.
- **mergePatchPairs**: list of patches to be merged, see section 5.3.2.

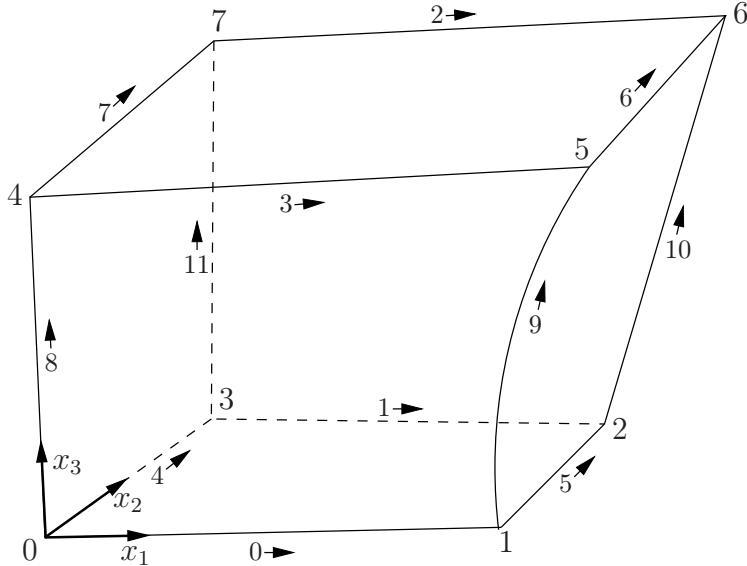


Figure 5.3: A single block

Keyword	Description	Example/selection
<code>convertToMeters</code>	Scaling factor for the vertex coordinates	0.001 scales to mm
<code>vertices</code>	List of vertex coordinates	(0 0 0)
<code>edges</code>	Used to describe <code>arc</code> or <code>spline</code> edges	<code>arc 1 4 (0.939 0.342 -0.5)</code>
<code>block</code>	Ordered list of vertex labels and mesh size	<code>hex (0 1 2 3 4 5 6 7) (10 10 1)</code>
<code>patches</code>	List of patches	<code>simpleGrading (1.0 1.0 1.0)</code> <code>symmetryPlane base ((0 1 2 3))</code>
<code>mergePatchPairs</code>	List of patches to be merged	see section 5.3.2

Table 5.2: Keywords used in *blockMeshDict*.

The `convertToMeters` keyword specifies a scaling factor by which all vertex coordinates in the mesh description are multiplied. For example,

```
convertToMeters    0.001;
```

means that all coordinates are multiplied by 0.001, *i.e.* the values quoted in the *blockMeshDict* file are in mm.

5.3.1.1 The vertices

The vertices of the blocks of the mesh are given next as a standard list named `vertices`, *e.g.* for our example block in Figure 5.3, the vertices are:

```
vertices
(
```

```

( 0    0    0 )    // vertex number 0
( 1    0    0.1)   // vertex number 1
( 1.1  1    0.1)   // vertex number 2
( 0    1    0.1)   // vertex number 3
(-0.1 -0.1  1 )   // vertex number 4
( 1.3  0    1.2)   // vertex number 5
( 1.4  1.1  1.3)   // vertex number 6
( 0    1    1.1)   // vertex number 7
);

```

5.3.1.2 The edges

Each edge joining 2 vertex points is assumed to be straight by default. However any edge may be specified to be curved by entries in a list named `edges`. The list is optional; if the geometry contains no curved edges, it may be omitted.

Each entry for a curved edge begins with a keyword specifying the type of curve from those listed in Table 5.3.

Keyword selection	Description	Additional entries
<code>arc</code>	Circular arc	Single interpolation point
<code>spline</code>	Spline curve	List of interpolation points
<code>polyLine</code>	Set of lines	List of interpolation points
<code>BSpline</code>	B-spline curve	List of interpolation points
<code>line</code>	Straight line	—

Table 5.3: Edge types available in the *blockMeshDict* dictionary.

The keyword is then followed by the labels of the 2 vertices that the edge connects. Following that, interpolation points must be specified through which the edge passes. For a `arc`, a single interpolation point is required, which the circular arc will intersect. For `spline`, `polyLine` and `BSpline`, a list of interpolation points is required. The `line` edge is directly equivalent to the option executed by default, and requires no interpolation points. Note that there is no need to use the `line` edge but it is included for completeness. For our example block in Figure 5.3 we specify an `arc` edge connecting vertices 1 and 5 as follows through the interpolation point (1.1, 0.0, 0.5):

```

edges
(
    arc 1 5 (1.1 0.0 0.5)
);

```

5.3.1.3 The blocks

The block definitions are contained in a list named `blocks`. Each block definition is a compound entry consisting of a list of vertex labels whose order is described in section 5.3, a vector giving the number of cells required in each direction, the type and list of cell expansion ratio in each direction.

Then the blocks are defined as follows:

```

blocks
(
    hex (0 1 2 3 4 5 6 7)      // vertex numbers
    (10 10 10)                 // numbers of cells in each direction
    simpleGrading (1 2 3)       // cell expansion ratios
);

```

The definition of each block is as follows:

Vertex numbering The first entry is the shape identifier of the block, as defined in the *.OpenFOAM-5.0/cellModels* file. The shape is always `hex` since the blocks are always hexahedra. There follows a list of vertex numbers, ordered in the manner described on page U-148.

Number of cells The second entry gives the number of cells in each of the x_1 x_2 and x_3 directions for that block.

Cell expansion ratios The third entry gives the cell expansion ratios for each direction in the block. The expansion ratio enables the mesh to be graded, or refined, in specified directions. The ratio is that of the width of the end cell δ_e along one edge of a block to the width of the start cell δ_s along that edge, as shown in Figure 5.4. Each of the following keywords specify one of two types of grading specification available in `blockMesh`.

`simpleGrading` The simple description specifies uniform expansions in the local x_1 , x_2 and x_3 directions respectively with only 3 expansion ratios, e.g.

`simpleGrading (1 2 3)`
`edgeGrading` The full cell expansion description gives a ratio for each edge of the block, numbered according to the scheme shown in Figure 5.3 with the arrows representing the direction ‘from first cell...to last cell’ e.g. something like

`edgeGrading (1 1 1 1 2 2 2 2 3 3 3 3)`
This means the ratio of cell widths along edges 0-3 is 1, along edges 4-7 is 2 and along 8-11 is 3 and is directly equivalent to the `simpleGrading` example given above.

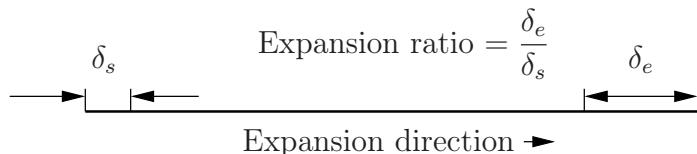


Figure 5.4: Mesh grading along a block edge

5.3.1.4 Multi-grading of a block

Using a single expansion ratio to describe mesh grading permits only “one-way” grading within a mesh block. In some cases, it reduces complexity and effort to be able to control grading within separate divisions of a single block, rather than have to define several blocks with one grading per block. For example, to mesh a channel with two opposing walls and

grade the mesh towards the walls requires three regions: two with grading to the wall with one in the middle without grading.

OpenFOAM v2.4+ includes multi-grading functionality that can divide a block in a given direction and apply different grading within each division. This multi-grading is specified by replacing any single value expansion ratio in the grading specification of the block, *e.g.* “1”, “2”, “3” in

```
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 300 100)
        simpleGrading (1 2 3);
);
```

We will present multi-grading for the following example:

- split the block into 3 divisions in the *y*-direction, representing 20%, 60% and 20% of the block length;
- include 30% of the total cells in the *y*-direction (300) in *each* divisions 1 and 3 and the remaining 40% in division 2;
- apply 1:4 expansion in divisions 1 and 3, and zero expansion in division 2.

We can specify this by replacing the *y*-direction expansion ratio “2” in the example above with the following:

```
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 300 100)
        simpleGrading
        (
            1           // x-direction expansion ratio
            (
                (0.2 0.3 4)   // 20% y-dir, 30% cells, expansion = 4
                (0.6 0.4 1)   // 60% y-dir, 40% cells, expansion = 1
                (0.2 0.3 0.25) // 20% y-dir, 30% cells, expansion = 0.25 (1/4)
            )
            3           // z-direction expansion ratio
        );
);
```

Both the fraction of the block and the fraction of the cells are normalized automatically. They can be specified as percentages, fractions, absolute lengths, *etc.* and do not need to sum to 100, 1, *etc.* The example above can be specified using percentages, *e.g.*

```
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 300 100)
        simpleGrading
```

```

(
 1
 (
  (20 30 4) // 20%, 30%...
  (60 40 1)
  (20 30 0.25)
 )
 3
)
);

```

5.3.1.5 The boundary

The boundary of the mesh is given in a list named `boundary`. The boundary is broken into patches (regions), where each patch in the list has its name as the keyword, which is the choice of the user, although we recommend something that conveniently identifies the patch, *e.g.*`inlet`; the name is used as an identifier for setting boundary conditions in the field data files. The patch information is then contained in sub-dictionary with:

- **type**: the patch type, either a generic `patch` on which some boundary conditions are applied or a particular geometric condition, as listed in section 5.2.1;
- **faces**: a list of block faces that make up the patch and whose name is the choice of the user, although we recommend something that conveniently identifies the patch, *e.g.*`inlet`; the name is used as an identifier for setting boundary conditions in the field data files.

`blockMesh` collects faces from any boundary patch that is omitted from the `boundary` list and assigns them to a default patch named `defaultFaces` of type `empty`. This means that for a 2 dimensional geometry, the user has the option to omit block faces lying in the 2D plane, knowing that they will be collected into an `empty` patch as required.

Returning to the example block in Figure 5.3, if it has an inlet on the left face, an output on the right face and the four other faces are walls then the patches could be defined as follows:

```

boundary          // keyword
(
  inlet           // patch name
  {
    type patch; // patch type for patch 0
    faces
    (
      (0 4 7 3) // block face in this patch
    );
  }              // end of 0th patch definition

  outlet          // patch name
  {
    type patch; // patch type for patch 1
  }
);

```

```

faces
(
    (1 2 6 5)
);
}

walls
{
    type wall;
    faces
    (
        (0 1 5 4)
        (0 3 2 1)
        (3 7 6 2)
        (4 5 6 7)
    );
}
);

```

Each block face is defined by a list of 4 vertex numbers. The order in which the vertices are given **must** be such that, looking from inside the block and starting with any vertex, the face must be traversed in a clockwise direction to define the other vertices.

When specifying a **cyclic** patch in **blockMesh**, the user must specify the name of the related cyclic patch through the **neighbourPatch** keyword. For example, a pair of cyclic patches might be specified as follows:

```

left
{
    type      cyclic;
    neighbourPatch right;
    faces     ((0 4 7 3));
}
right
{
    type      cyclic;
    neighbourPatch left;
    faces     ((1 5 6 2));
}

```

5.3.2 Multiple blocks

A mesh can be created using more than 1 block. In such circumstances, the mesh is created as has been described in the preceding text; the only additional issue is the connection between blocks, in which there are two distinct possibilities:

face matching the set of faces that comprise a patch from one block are formed from *the same set of vertices* as a set of faces patch that comprise a patch from another block;

face merging a group of faces from a patch from one block are connected to another group of faces from a patch from another block, to create a new set of internal faces connecting the two blocks.

To connect two blocks with **face matching**, the two patches that form the connection should simply be ignored from the `patches` list. `blockMesh` then identifies that the faces do not form an external boundary and combines each collocated pair into a single internal faces that connects cells from the two blocks.

The alternative, **face merging**, requires that the block patches to be merged are first defined in the `patches` list. Each pair of patches whose faces are to be merged must then be included in an optional list named `mergePatchPairs`. The format of `mergePatchPairs` is:

```
mergePatchPairs
(
    ( <masterPatch> <slavePatch> ) // merge patch pair 0
    ( <masterPatch> <slavePatch> ) // merge patch pair 1
    ...
)
```

The pairs of patches are interpreted such that the first patch becomes the *master* and the second becomes the *slave*. The rules for merging are as follows:

- the faces of the master patch remain as originally defined, with all vertices in their original location;
- the faces of the slave patch are projected onto the master patch where there is some separation between slave and master patch;
- the location of any vertex of a slave face might be adjusted by `blockMesh` to eliminate any face edge that is shorter than a minimum tolerance;
- if patches overlap as shown in Figure 5.5, each face that does not merge remains as an external face of the original patch, on which boundary conditions must then be applied;
- if all the faces of a patch are merged, then the patch itself will contain no faces and is removed.

The consequence is that the original geometry of the slave patch will not necessarily be completely preserved during merging. Therefore in a case, say, where a cylindrical block is being connected to a larger block, it would be wise to assign the master patch to the cylinder, so that its cylindrical shape is correctly preserved. There are some additional recommendations to ensure successful merge procedures:

- in 2 dimensional geometries, the size of the cells in the third dimension, *i.e.* out of the 2D plane, should be similar to the width/height of cells in the 2D plane;
- it is inadvisable to merge a patch twice, *i.e.* include it twice in `mergePatchPairs`;
- where a patch to be merged shares a common edge with another patch to be merged, both should be declared as a master patch.

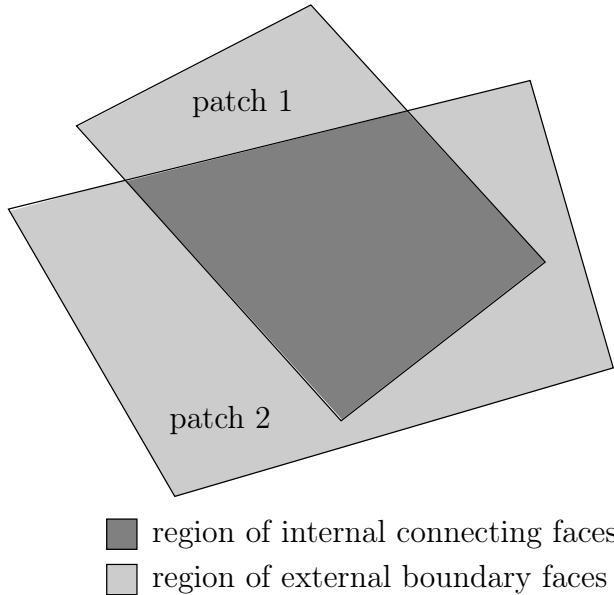


Figure 5.5: Merging overlapping patches

5.3.3 Projection of vertices, edges and faces

`blockMesh` can be configured to create body fitted meshes using projection of vertices, edges and/or faces onto specified geometry. The functionality can be used to mesh spherical and cylindrical geometries such as pipes and vessels conveniently. The user can specify within the `blockMeshDict` file within an optional `geometry` dictionary with the same format as used in the `snappyHexMeshDict` file. For example to specify a cylinder using the built in geometric type the user could configure with the following:

```
geometry
{
    cylinder
    {
        type searchableCylinder;
        point1 (0 -4 0);
        point2 (0 4 0);
        radius 0.7;
    }
};
```

The user can then project vertices, edges and/or faces onto the cylinder surface with the `project` keyword using example syntax shown below:

```
vertices
(
    project (-1 -0.1 -1) (cylinder)
    project ( 1 -0.1 -1) (cylinder)
    ...
);

edges
(
    project 0 1 (cylinder)
    ...
);

faces
(
    project (0 4 7 3) cylinder
    ...
);
```

The use of this functionality is demonstrated in tutorials which can be located by searching for the `project` keyword in all the `blockMeshDict` files by:

```
find $FOAM_TUTORIALS -name blockMeshDict | xargs grep -l project
```

5.3.4 Naming vertices, edges, faces and blocks

Vertices, edges, faces and blocks can be named in the configuration of a `blockMeshDict` file, which can make it easier to manage more complex examples. It is done simply using the `name` keyword. The following syntax shows naming using the example for projection in the previous subsection:

```
vertices
(
    name v0 project (-1 -0.1 -1) (cylinder)
    name v1 project ( 1 -0.1 -1) (cylinder)
    ...
);

edges
(
    project v0 v1 (cylinder)
    ...
);
```

When a name is provided for a given entity, it can be used to replace the index. In the example about, rather than specify the edge using vertex indices 0 and 1, the names `v0` and `v1` are used.

5.3.5 Creating blocks with fewer than 8 vertices

It is possible to collapse one or more pair(s) of vertices onto each other in order to create a block with fewer than 8 vertices. The most common example of collapsing vertices is when creating a 6-sided wedge shaped block for 2-dimensional axi-symmetric cases that use the `wedge` patch type described in section 5.2.2. The process is best illustrated by using a simplified version of our example block shown in Figure 5.6. Let us say we wished to create a wedge shaped block by collapsing vertex 7 onto 4 and 6 onto 5. This is simply done by exchanging the vertex number 7 by 4 and 6 by 5 respectively so that the block numbering would become:

```
hex (0 1 2 3 4 5 5 4)
```

The same applies to the patches with the main consideration that the block face containing the collapsed vertices, previously (4 5 6 7) now becomes (4 5 5 4). This is a block face of zero area which creates a patch with no faces in the `polyMesh`, as the user can see in a `boundary` file for such a case. The patch should be specified as `empty` in the `blockMeshDict` and the boundary condition for any fields should consequently be `empty` also.

5.3.6 Running `blockMesh`

As described in section 3.3, the following can be executed at the command line to run `blockMesh` for a case in the `<case>` directory:

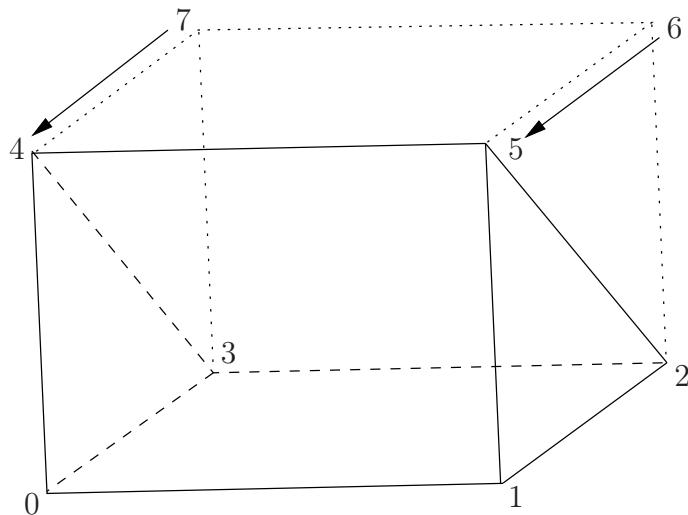


Figure 5.6: Creating a wedge shaped block with 6 vertices

```
blockMesh -case <case>
```

The *blockMeshDict* file must exist in the *system* (or *constant/polyMesh*) directory.

5.4 Mesh generation with the `snappyHexMesh` utility

This section describes the mesh generation utility, `snappyHexMesh`, supplied with OpenFOAM. The `snappyHexMesh` utility generates 3-dimensional meshes containing hexahedra (hex) and split-hexahedra (split-hex) automatically from triangulated surface geometries, or tri-surfaces, in Stereolithography (STL) or Wavefront Object (OBJ) format. The mesh approximately conforms to the surface by iteratively refining a starting mesh and morphing the resulting split-hex mesh to the surface. An optional phase will shrink back the resulting mesh and insert cell layers. The specification of mesh refinement level is very flexible and the surface handling is robust with a pre-specified final mesh quality. It runs in parallel with a load balancing step every iteration.

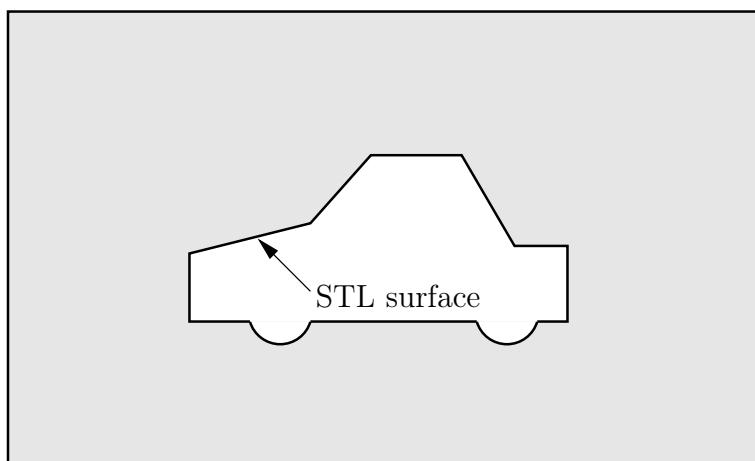


Figure 5.7: Schematic 2D meshing problem for `snappyHexMesh`

5.4.1 The mesh generation process of `snappyHexMesh`

The process of generating a mesh using `snappyHexMesh` will be described using the schematic in Figure 5.7. The objective is to mesh a rectangular shaped region (shaded grey in the figure) surrounding an object described by a tri-surface, *e.g.* typical for an external aerodynamics simulation. Note that the schematic is 2-dimensional to make it easier to understand, even though the `snappyHexMesh` is a 3D meshing tool.

In order to run `snappyHexMesh`, the user requires the following:

- one or more tri-surface files located in a `constant/triSurface` sub-directory of the case directory;
- a background hex mesh which defines the extent of the computational domain and a base level mesh density; typically generated using `blockMesh`, discussed in section 5.4.2.
- a `snappyHexMeshDict` dictionary, with appropriate entries, located in the `system` sub-directory of the case.

The `snappyHexMeshDict` dictionary includes: switches at the top level that control the various stages of the meshing process; and, individual sub-directories for each process. The entries are listed below.

- `castellatedMesh`: to switch on creation of the castellated mesh.
- `snap`: to switch on surface snapping stage.
- `addLayers`: to switch on surface layer insertion.
- `mergeTolerance`: merge tolerance as fraction of bounding box of initial mesh.
- `geometry`: sub-dictionary of all surface geometry used.
- `castellatedMeshControls`: sub-dictionary of controls for castellated mesh.
- `snapControls`: sub-dictionary of controls for surface snapping.
- `addLayersControls`: sub-dictionary of controls for layer addition.
- `meshQualityControls`: sub-dictionary of controls for mesh quality.

All the geometry used by `snappyHexMesh` is specified in a `geometry` sub-dictionary in the `snappyHexMeshDict` dictionary. The geometry can be specified through a tri-surface or bounding geometry entities in OpenFOAM. An example is given below:

```
geometry
{
    sphere1      // User defined region name
    {
        type      triSurfaceMesh;
        file      "sphere1.obj"; // surface geometry OBJ file
        regions
        {
            secondSolid          // Named region in the OBJ file
            {
                name mySecondPatch; // User-defined patch name
            }                      // otherwise given sphere1_secondSolid
        }
    }
}
```

```

box1x1x1    // User defined region name
{
    type    searchableBox;      // region defined by bounding box
    min     (1.5 1 -0.5);
    max     (3.5 2 0.5);
}

sphere2      // User defined region name
{
    type    searchableSphere;   // region defined by bounding sphere
    centre  (1.5 1.5 1.5);
    radius  1.03;
}
;

```

5.4.2 Creating the background hex mesh

Before `snappyHexMesh` is executed the user must create a background mesh of hexahedral cells that fills the entire region within by the external boundary as shown in Figure 5.8. This can be done simply using `blockMesh`. The following criteria must be observed when

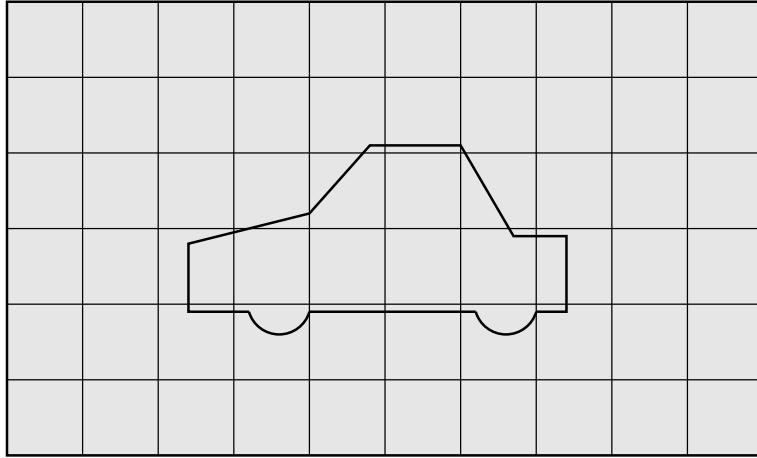


Figure 5.8: Initial mesh generation in `snappyHexMesh` meshing process

creating the background mesh:

- the mesh must consist purely of hexes;
- the cell aspect ratio should be approximately 1, at least near surfaces at which the subsequent snapping procedure is applied, otherwise the convergence of the snapping procedure is slow, possibly to the point of failure;
- there must be at least one intersection of a cell edge with the tri-surface, *i.e.* a mesh of one cell will not work.

5.4.3 Cell splitting at feature edges and surfaces

Cell splitting is performed according to the specification supplied by the user in the `castellatedMeshControls` sub-dictionary in the `snappyHexMeshDict`. The entries for `castellatedMeshControls` are presented below.

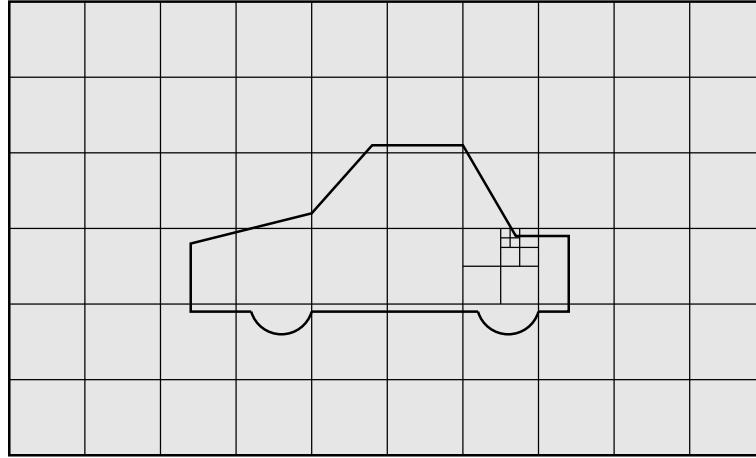


Figure 5.9: Cell splitting by feature edge in `snappyHexMesh` meshing process

- `locationInMesh`: location vector inside the region to be meshed; vector must not coincide with a cell face either before or during refinement.
- `maxLocalCells`: max number of cells per processor during refinement.
- `maxGlobalCells`: overall cell limit during refinement (*i.e.* before removal).
- `minRefinementCells`: if `minRefinementCells` \geq number of cells to be refined, surface refinement stops.
- `nCellsBetweenLevels`: number of buffer layers of cells between successive levels of refinement (typically set to 3).
- `resolveFeatureAngle`: applies maximum level of refinement to cells that can see intersections whose angle exceeds `resolveFeatureAngle` (typically set to 30).
- `features`: list of features for refinement.
- `refinementSurfaces`: dictionary of surfaces for refinement.
- `refinementRegions`: dictionary of regions for refinement.

The splitting process begins with cells being selected according to specified edge features first within the domain as illustrated in Figure 5.9. The `features` list in the `castellatedMeshControls` sub-dictionary permits dictionary entries containing a name of an `edgeMesh` file and the `level` of refinement, *e.g.*:

```
features
(
    {
        file "features.eMesh"; // file containing edge mesh
        level 2; // level of refinement
    }
);
```

The `edgeMesh` containing the features can be extracted from the tri-surface file using `surfaceFeatureExtract` which specifies the tri-surface and controls such as included angle through a `surfaceFeatureExtractDict` configuration file, examples of which can be found in several tutorials and the `$FOAM_UTILITIES/surface/surfaceFeatureExtract` directory in the OpenFOAM installation. The utility is simply run by executing the following in a terminal

surfaceFeatureExtract

Following feature refinement, cells are selected for splitting in the locality of specified surfaces as illustrated in Figure 5.10. The `refinementSurfaces` dictionary in `castellatedMeshControls` requires dictionary entries for each STL surface and a default level specification of the minimum and maximum refinement in the form (`<min> <max>`). The minimum level is applied generally across the surface; the maximum level is applied to cells that can see intersections that form an angle in excess of that specified by `resolveFeatureAngle`.

The refinement can optionally be overridden on one or more specific region of an STL surface. The region entries are collected in a `regions` sub-dictionary. The keyword for each region entry is the name of the region itself and the refinement level is contained within a further sub-dictionary. An example is given below:

```
refinementSurfaces
{
    sphere1
    {
        level (2 2); // default (min max) refinement for whole surface
        regions
        {
            secondSolid
            {
                level (3 3); // optional refinement for secondSolid region
            }
        }
    }
}
```

5.4.4 Cell removal

Once the feature and surface splitting is complete a process of cell removal begins. Cell removal requires one or more regions enclosed entirely by a bounding surface within the domain. The region in which cells are retained are simply identified by a location vector within that region, specified by the `locationInMesh` keyword in `castellatedMeshControls`. Cells are retained if, approximately speaking, 50% or more of their volume lies within the region. The remaining cells are removed accordingly as illustrated in Figure 5.11.



Figure 5.10: Cell splitting by surface in `snappyHexMesh` meshing process

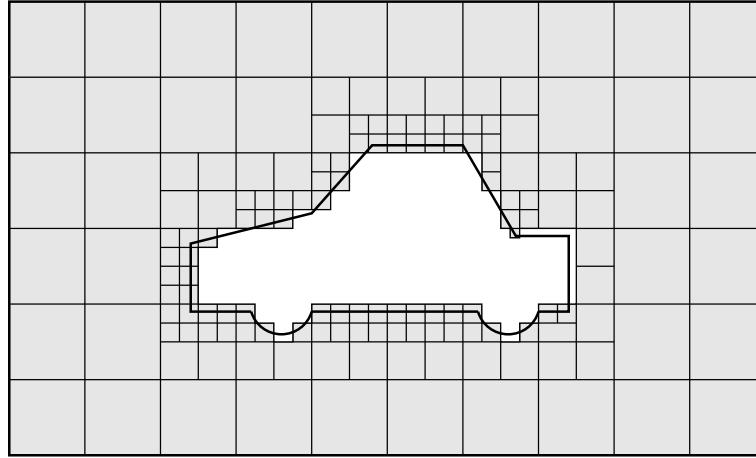


Figure 5.11: Cell removal in `snappyHexMesh` meshing process

5.4.5 Cell splitting in specified regions

Those cells that lie within one or more specified volume regions can be further split as illustrated in Figure 5.12 by a rectangular region shown by dark shading. The `refinementRegions` sub-dictionary in `castellatedMeshControls` contains entries for refinement of the volume regions specified in the `geometry` sub-dictionary. A refinement mode is applied to each region which can be:

- `inside` refines inside the volume region;
- `outside` refines outside the volume region
- `distance` refines according to distance to the surface; and can accommodate different levels at multiple distances with the `levels` keyword.

For the `refinementRegions`, the refinement level is specified by the `levels` list of entries with the format (`<distance> <level>`). In the case of `inside` and `outside` refinement, the `<distance>` is not required so is ignored (but it must be specified). Examples are shown below:

```
refinementRegions
{
    box1x1x1
    {
        mode inside;
        levels ((1.0 4));           // refinement level 4 (1.0 entry ignored)
    }

    sphere1
    {
        mode distance;            // refinement level 5 within 1.0 m
        levels ((1.0 5) (2.0 3)); // refinement level 3 within 2.0 m
        levels must be ordered nearest first
    }
}
```

5.4.6 Snapping to surfaces

The next stage of the meshing process involves moving cell vertex points onto surface geometry to remove the jagged castellated surface from the mesh. The process is:

1. displace the vertices in the castellated boundary onto the STL surface;
2. solve for relaxation of the internal mesh with the latest displaced boundary vertices;
3. find the vertices that cause mesh quality parameters to be violated;
4. reduce the displacement of those vertices from their initial value (at 1) and repeat from 2 until mesh quality is satisfied.

The method uses the settings in the *snapControls* sub-dictionary in *snappyHexMeshDict*, listed below.

- **nSmoothPatch**: number of patch smoothing iterations before finding correspondence to surface (typically 3).
- **tolerance**: ratio of distance for points to be attracted by surface feature point or edge, to local maximum edge length (typically 2.0).
- **nSolveIter**: number of mesh displacement relaxation iterations (typically 30-100).
- **nRelaxIter**: maximum number of snapping relaxation iterations (typically 5).

An example is illustrated in the schematic in Figure 5.13 (albeit with mesh motion that looks slightly unrealistic).

5.4.7 Mesh layers

The mesh output from the snapping stage may be suitable for the purpose, although it can produce some irregular cells along boundary surfaces. There is an optional stage of the meshing process which introduces additional layers of hexahedral cells aligned to the boundary surface as illustrated by the dark shaded cells in Figure 5.14.

The process of mesh layer addition involves shrinking the existing mesh from the boundary and inserting layers of cells, broadly as follows:

1. the mesh is projected back from the surface by a specified thickness in the direction normal to the surface;
2. solve for relaxation of the internal mesh with the latest projected boundary vertices;
3. check if validation criteria are satisfied otherwise reduce the projected thickness and return to 2; if validation cannot be satisfied for any thickness, do not insert layers;
4. if the validation criteria can be satisfied, insert mesh layers;
5. the mesh is checked again; if the checks fail, layers are removed and we return to 2.

The layer addition procedure uses the settings in the *addLayersControls* sub-dictionary in *snappyHexMeshDict*; entries are listed below. The user has the option of 4 different layer thickness parameters — **expansionRatio**, **finalLayerThickness**, **firstLayerThickness**, **thickness** — *from which they must specify 2 only*; more than 2, and the problem is over-specified.

- **layers**: dictionary specifying layers to be inserted.

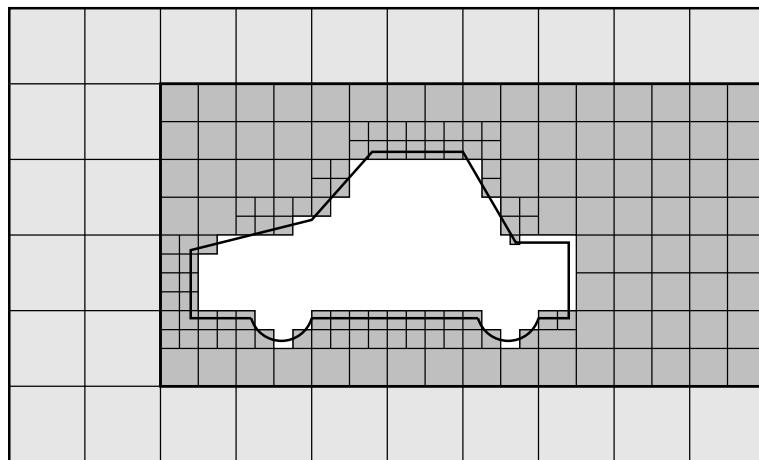


Figure 5.12: Cell splitting by region in `snappyHexMesh` meshing process

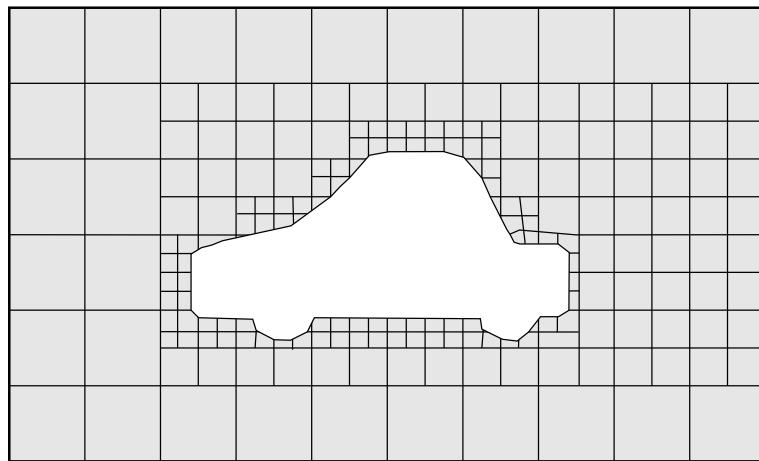


Figure 5.13: Surface snapping in `snappyHexMesh` meshing process

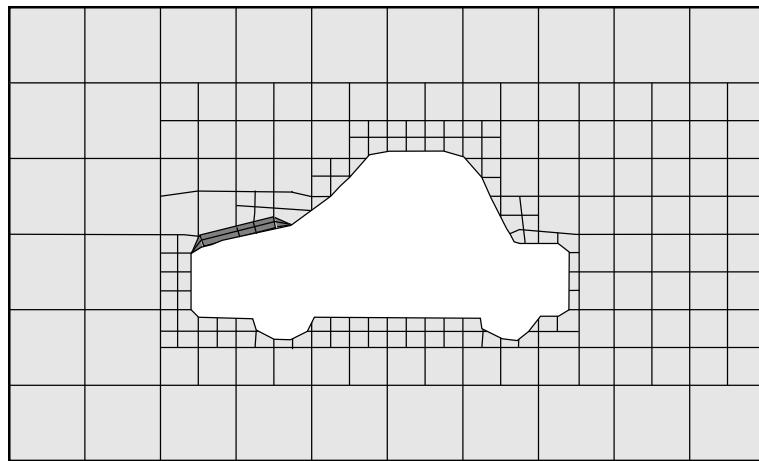


Figure 5.14: Layer addition in `snappyHexMesh` meshing process

- **relativeSizes**: switch that sets whether the specified layer thicknesses are relative to undistorted cell size outside layer or absolute.
- **expansionRatio**: expansion factor for layer mesh, increase in size from one layer to the next.
- **finalLayerThickness**: thickness of layer furthest from the wall, usually in combination with relative sizes according to the **relativeSizes** entry.
- **firstLayerThickness**: thickness of layer nearest the wall, usually in combination with absolute sizes according to the **relativeSizes** entry.
- **thickness**: total thickness of all layers of cells, usually in combination with absolute sizes according to the
- **relativeSizes** entry.
- **minThickness**: minimum thickness of cell layer, either relative or absolute (as above).
- **nGrow**: number of layers of connected faces that are not grown if points do not get extruded; helps convergence of layer addition close to features.
- **featureAngle**: angle above which surface is not extruded.
- **nRelaxIter**: maximum number of snapping relaxation iterations (typically 5).
- **nSmoothSurfaceNormals**: number of smoothing iterations of surface normals (typically 1).
- **nSmoothNormals**: number of smoothing iterations of interior mesh movement direction (typically 3).
- **nSmoothThickness**: smooth layer thickness over surface patches (typically 10).
- **maxFaceThicknessRatio**: stop layer growth on highly warped cells (typically 0.5).
- **maxThicknessToMedialRatio**: reduce layer growth where ratio thickness to medial distance is large (typically 0.3)
- **minMedianAxisAngle**: angle used to pick up medial axis points (typically 90).
- **nBufferCellsNoExtrude**: create buffer region for new layer terminations (typically 0).
- **nLayerIter**: overall max number of layer addition iterations (typically 50).
- **nRelaxedIter**: max number of iterations after which the controls in the **relaxed** sub dictionary of **meshQuality** are used (typically 20).

The **layers** sub-dictionary contains entries for each *patch* on which the layers are to be applied and the number of surface layers required. The patch name is used because the layers addition relates to the existing mesh, not the surface geometry; hence applied to a patch, not a surface region. An example **layers** entry is as follows:

```

layers
{
    sphere1_firstSolid
    {
        nSurfaceLayers 1;
    }
    maxY
    {
        nSurfaceLayers 1;
    }
}

```

5.4.8 Mesh quality controls

The mesh quality is controlled by the entries in the `meshQualityControls` sub-dictionary in `snappyHexMeshDict`; entries are listed below.

- `maxNonOrtho`: maximum non-orthogonality allowed (degrees, typically 65).
- `maxBoundarySkewness`: max boundary face skewness allowed (typically 20).
- `maxInternalSkewness`: max internal face skewness allowed (typically 4).
- `maxConcave`: max concaveness allowed (typically 80).
- `minFlatness`: ratio of minimum projected area to actual area (typically 0.5)
- `minTetQuality`: minimum quality of tetrahedral cells from cell decomposition; generally deactivated by setting large negative number since v5.0 when new barycentric tracking was introduced, which could handle negative tets.
- `minVol`: minimum cell pyramid volume (typically 1e-13, large negative number disables).
- `minArea`: minimum face area (typically -1).
- `minTwist`: minimum face twist (typically 0.05).
- `minDeterminant`: minimum normalised cell determinant; 1 = hex; ≤ 0 = illegal cell (typically 0.001).
- `minFaceWeight`: 0→0.5 (typically 0.05).
- `minVolRatio`: 0→1.0 (typically 0.01).
- `minTriangleTwist`: > 0 for Fluent compatibility (typically -1).
- `nSmoothScale`: number of error distribution iterations (typically 4).
- `errorReduction`: amount to scale back displacement at error points (typically 0.75).
- `relaxed`: sub-dictionary that can include modified values for the above keyword entries to be used when `nRelaxedIter` is exceeded in the layer addition process.

5.5 Mesh conversion

The user can generate meshes using other packages and convert them into the format that OpenFOAM uses. There are numerous mesh conversion utilities listed in section 3.6.3. Some of the more popular mesh converters are listed below and their use is presented in this section.

`fluentMeshToFoam` reads a `Fluent.msh` mesh file, working for both 2-D and 3-D cases;

`starToFoam` reads STAR-CD/PROSTAR mesh files.

`gambitToFoam` reads a `GAMBIT.neu` neutral file;

`ideasToFoam` reads an I-DEAS mesh written in ANSYS.`.ans` format;

`cfx4ToFoam` reads a CFX mesh written in `.geo` format;

5.5.1 fluentMeshToFoam

Fluent writes mesh data to a single file with a `.msh` extension. The file must be written in ASCII format, which is not the default option in Fluent. It is possible to convert single-stream Fluent meshes, including the 2 dimensional geometries. In OpenFOAM, 2 dimensional geometries are currently treated by defining a mesh in 3 dimensions, where the front and back plane are defined as the `empty` boundary patch type. When reading a 2 dimensional Fluent mesh, the converter automatically extrudes the mesh in the third direction and adds the empty patch, naming it `frontAndBackPlanes`.

The following features should also be observed.

- The OpenFOAM converter will attempt to capture the Fluent boundary condition definition as much as possible; however, since there is no clear, direct correspondence between the OpenFOAM and Fluent boundary conditions, the user should check the boundary conditions before running a case.
- Creation of axi-symmetric meshes from a 2 dimensional mesh is currently not supported but can be implemented on request.
- Multiple material meshes are not permitted. If multiple fluid materials exist, they will be converted into a single OpenFOAM mesh; if a solid region is detected, the converter will attempt to filter it out.
- Fluent allows the user to define a patch which is internal to the mesh, *i.e.* consists of the faces with cells on both sides. Such patches are not allowed in OpenFOAM and the converter will attempt to filter them out.
- There is currently no support for embedded interfaces and refinement trees.

The procedure of converting a `Fluent.msh` file is first to create a new OpenFOAM case by creating the necessary directories/files: the case directory containing a `controlDict` file in a `system` subdirectory. Then at a command prompt the user should execute:

```
fluentMeshToFoam <meshFile>
```

where `<meshFile>` is the name of the `.msh` file, including the full or relative path.

5.5.2 starToFoam

This section describes how to convert a mesh generated on the STAR-CD code into a form that can be read by OpenFOAM mesh classes. The mesh can be generated by any of the packages supplied with STAR-CD, *i.e.* PROSTAR, SAMM, ProAM and their derivatives. The converter accepts any single-stream mesh including integral and arbitrary couple matching and all cell types are supported. The features that the converter does not support are:

- multi-stream mesh specification;
- baffles, *i.e.* zero-thickness walls inserted into the domain;
- partial boundaries, where an uncovered part of a couple match is considered to be a boundary face;
- sliding interfaces.

For multi-stream meshes, mesh conversion can be achieved by writing each individual stream as a separate mesh and reassemble them in OpenFOAM.

OpenFOAM adopts a policy of only accepting input meshes that conform to the fairly stringent validity criteria specified in section 5.1. It will simply not run using invalid meshes and cannot convert a mesh that is itself invalid. The following sections describe steps that must be taken when generating a mesh using a mesh generating package supplied with STAR-CD to ensure that it can be converted to OpenFOAM format. To avoid repetition in the remainder of the section, the mesh generation tools supplied with STAR-CD will be referred to by the collective name STAR-CD.

5.5.2.1 General advice on conversion

We strongly recommend that the user run the STAR-CD mesh checking tools before attempting a `starToFoam` conversion and, after conversion, the `checkMesh` utility should be run on the newly converted mesh. Alternatively, `starToFoam` may itself issue warnings containing PROSTAR commands that will enable the user to take a closer look at cells with problems. Problematic cells and matches should be checked and fixed before attempting to use the mesh with OpenFOAM. Remember that an invalid mesh will not run with OpenFOAM, but it may run in another environment that does not impose the validity criteria.

Some problems of tolerance matching can be overcome by the use of a matching tolerance in the converter. However, there is a limit to its effectiveness and an apparent need to increase the matching tolerance from its default level indicates that the original mesh suffers from inaccuracies.

5.5.2.2 Eliminating extraneous data

When mesh generation is completed, remove any extraneous vertices and compress the cells boundary and vertex numbering, assuming that fluid cells have been created and all other cells are discarded. This is done with the following PROSTAR commands:

```
CSET NEWS FLUID
CSET INVE
```

The CSET should be empty. If this is not the case, examine the cells in CSET and adjust the model. If the cells are genuinely not desired, they can be removed using the PROSTAR command:

```
CDEL CSET
```

Similarly, vertices will need to be discarded as well:

```
CSET NEWS FLUID
VSET NEWS CSET
VSET INVE
```

Before discarding these unwanted vertices, the unwanted boundary faces have to be collected before purging:

```
CSET NEWS FLUID
VSET NEWS CSET
BSET NEWS VSET ALL
BSET INVE
```

If the BSET is not empty, the unwanted boundary faces can be deleted using:

```
BDEL BSET
```

At this time, the model should contain only the fluid cells and the supporting vertices, as well as the defined boundary faces. All boundary faces should be fully supported by the vertices of the cells, if this is not the case, carry on cleaning the geometry until everything is clean.

5.5.2.3 Removing default boundary conditions

By default, STAR-CD assigns wall boundaries to any boundary faces not explicitly associated with a boundary region. The remaining boundary faces are collected into a **default** boundary region, with the assigned boundary type 0. OpenFOAM deliberately does not have a concept of a **default** boundary condition for undefined boundary faces since it invites human error, *e.g.* there is no means of checking that we meant to give all the unassociated faces the default condition.

Therefore **all** boundaries for each OpenFOAM mesh must be specified for a mesh to be successfully converted. The **default** boundary needs to be transformed into a real one using the procedure described below:

1. Plot the geometry with **Wire Surface** option.
2. Define an extra boundary region with the same parameters as the **default** region 0 and add all visible faces into the new region, say 10, by selecting a zone option in the boundary tool and drawing a polygon around the entire screen draw of the model. This can be done by issuing the following commands in PROSTAR:

```
RDEF 10 WALL
BZON 10 ALL
```

3. We shall remove all previously defined boundary types from the set. Go through the boundary regions:

```
BSET NEWS REGI 1
BSET NEWS REGI 2
... 3, 4, ...
```

Collect the vertices associated with the boundary set and then the boundary faces associated with the vertices (there will be twice as many of them as in the original set).

```
BSET NEWS REGI 1
VSET NEWS BSET
BSET NEWS VSET ALL
BSET DELE REGI 1
REPL
```

This should give the faces of boundary Region 10 which have been defined on top of boundary Region 1. Delete them with BDEL BSET. Repeat these for all regions.

5.5.2.4 Renumbering the model

Renumber and check the model using the commands:

```
CSET NEW FLUID
CCOM CSET

VSET NEWS CSET
VSET INVE (Should be empty!)
VSET INVE
VCOM VSET

BSET NEWS VSET ALL
BSET INVE (Should be empty also!)
BSET INVE
BCOM BSET

CHECK ALL
GEOM
```

Internal PROSTAR checking is performed by the last two commands, which may reveal some other unforeseeable error(s). Also, take note of the scaling factor because PROSTAR only applies the factor for STAR-CD and not the geometry. If the factor is not 1, use the scalePoints utility in OpenFOAM.

5.5.2.5 Writing out the mesh data

Once the mesh is completed, place all the integral matches of the model into the couple type 1. All other types will be used to indicate arbitrary matches.

```
CPSET NEWS TYPE INTEGRAL
CPMOD CPSET 1
```

The components of the computational grid must then be written to their own files. This is done using PROSTAR for boundaries by issuing the command

BWRITE

by default, this writes to a *.23* file (versions prior to 3.0) or a *.bnd* file (versions 3.0 and higher). For cells, the command

CWRITE

outputs the cells to a *.14* or *.cel* file and for vertices, the command

VWRITE

outputs to file a *.15* or *.vrt* file. The current default setting writes the files in ASCII format. If couples are present, an additional couple file with the extension *.cpl* needs to be written out by typing:

CPWRITE

After outputting to the three files, exit PROSTAR or close the files. Look through the panels and take note of all STAR-CD sub-models, material and fluid properties used – the material properties and mathematical model will need to be set up by creating and editing OpenFOAM dictionary files.

The procedure of converting the PROSTAR files is first to create a new OpenFOAM case by creating the necessary directories. The PROSTAR files must be stored within the same directory and the user must change the file extensions: from *.23*, *.14* and *.15* (below STAR-CD version 3.0), or *.pcs*, *.cls* and *.vtx* (STAR-CD version 3.0 and above); to *.bnd*, *.cel* and *.vrt* respectively.

5.5.2.6 Problems with the *.vrt* file

The *.vrt* file is written in columns of data of specified width, rather than free format. A typical line of data might be as follows, giving a vertex number followed by the coordinates:

```
19422      -0.105988957      -0.413711881E-02 0.000000000E+00
```

If the ordinates are written in scientific notation and are negative, there may be no space between values, *e.g.*:

```
19423      -0.953953117E-01-0.338810333E-02 0.000000000E+00
```

The `starToFoam` converter reads the data using spaces to delimit the ordinate values and will therefore object when reading the previous example. Therefore, OpenFOAM includes a simple script, `foamCorrectVrt` to insert a space between values where necessary, *i.e.* it would convert the previous example to:

```
19423      -0.953953117E-01 -0.338810333E-02 0.000000000E+00
```

The `foamCorrectVrt` script should therefore be executed if necessary before running the `starToFoam` converter, by typing:

```
foamCorrectVrt <file>.vrt
```

5.5.2.7 Converting the mesh to OpenFOAM format

The translator utility `starToFoam` can now be run to create the boundaries, cells and points files necessary for a OpenFOAM run:

```
starToFoam <meshFilePrefix>
```

where `<meshFilePrefix>` is the name of the the prefix of the mesh files, including the full or relative path. After the utility has finished running, OpenFOAM boundary types should be specified by editing the `boundary` file by hand.

5.5.3 gambitToFoam

GAMBIT writes mesh data to a single file with a `.neu` extension. The procedure of converting a GAMBIT.`.neu` file is first to create a new OpenFOAM case, then at a command prompt, the user should execute:

```
gambitToFoam <meshFile>
```

where `<meshFile>` is the name of the `.neu` file, including the full or relative path.

The GAMBIT file format does not provide information about type of the boundary patch, *e.g.* wall, symmetry plane, cyclic. Therefore all the patches have been created as type patch. Please reset after mesh conversion as necessary.

5.5.4 ideasToFoam

OpenFOAM can convert a mesh generated by I-DEAS but written out in ANSYS format as a `.ans` file. The procedure of converting the `.ans` file is first to create a new OpenFOAM case, then at a command prompt, the user should execute:

```
ideasToFoam <meshFile>
```

where `<meshFile>` is the name of the `.ans` file, including the full or relative path.

5.5.5 cfx4ToFoam

CFX writes mesh data to a single file with a `.geo` extension. The mesh format in CFX is block-structured, *i.e.* the mesh is specified as a set of blocks with glueing information and the vertex locations. OpenFOAM will convert the mesh and capture the CFX boundary condition as best as possible. The 3 dimensional ‘patch’ definition in CFX, containing information about the porous, solid regions *etc.* is ignored with all regions being converted into a single OpenFOAM mesh. CFX supports the concept of a ‘default’ patch, where each external face without a defined boundary condition is treated as a `wall`. These faces are collected by the converter and put into a `defaultFaces` patch in the OpenFOAM mesh and given the type `wall`; of course, the patch type can be subsequently changed.

Like, OpenFOAM 2 dimensional geometries in CFX are created as 3 dimensional meshes of 1 cell thickness. If a user wishes to run a 2 dimensional case on a mesh created by CFX, the boundary condition on the front and back planes should be set to `empty`; the user should

ensure that the boundary conditions on all other faces in the plane of the calculation are set correctly. Currently there is no facility for creating an axi-symmetric geometry from a 2 dimensional CFX mesh.

The procedure of converting a `CFX.geo` file is first to create a new OpenFOAM case, then at a command prompt, the user should execute:

```
cfx4ToFoam <meshFile>
```

where `<meshFile>` is the name of the `.geo` file, including the full or relative path.

5.6 Mapping fields between different geometries

The `mapFields` utility maps one or more fields relating to a given geometry onto the corresponding fields for another geometry. It is completely generalised in so much as there does not need to be any similarity between the geometries to which the fields relate. However, for cases where the geometries are consistent, `mapFields` can be executed with a special option that simplifies the mapping process.

For our discussion of `mapFields` we need to define a few terms. First, we say that the data is mapped from the *source* to the *target*. The fields are deemed *consistent* if the geometry *and* boundary types, or conditions, of both source and target fields are identical. The field data that `mapFields` maps are those fields within the time directory specified by `startFrom/startTime` in the `controlDict` of the target case. The data is read from the equivalent time directory of the source case and mapped onto the equivalent time directory of the target case.

5.6.1 Mapping consistent fields

A mapping of consistent fields is simply performed by executing `mapFields` on the (target) case using the `-consistent` command line option as follows:

```
mapFields <source dir> -consistent
```

5.6.2 Mapping inconsistent fields

When the fields are not consistent, as shown in Figure 5.15, `mapFields` requires a `mapFieldsDict` dictionary in the `system` directory of the target case. The following rules apply to the mapping:

- the field data is mapped from source to target wherever possible, *i.e.* in our example all the field data within the target geometry is mapped from the source, except those in the shaded region which remain unaltered;
- the patch field data is left unaltered unless specified otherwise in the `mapFieldsDict` dictionary.

The `mapFieldsDict` dictionary contain two lists that specify mapping of patch data. The first list is `patchMap` that specifies mapping of data between pairs of source and target patches that are geometrically coincident, as shown in Figure 5.15. The list contains each pair of

names of source and target patch. The second list is `cuttingPatches` that contains names of target patches whose values are to be mapped from the source internal field through which the target patch cuts. In the situation where the target patch only cuts through part of the source internal field, *e.g.* bottom left target patch in our example, those values within the internal field are mapped and those outside remain unchanged. An example `mapFieldsDict`

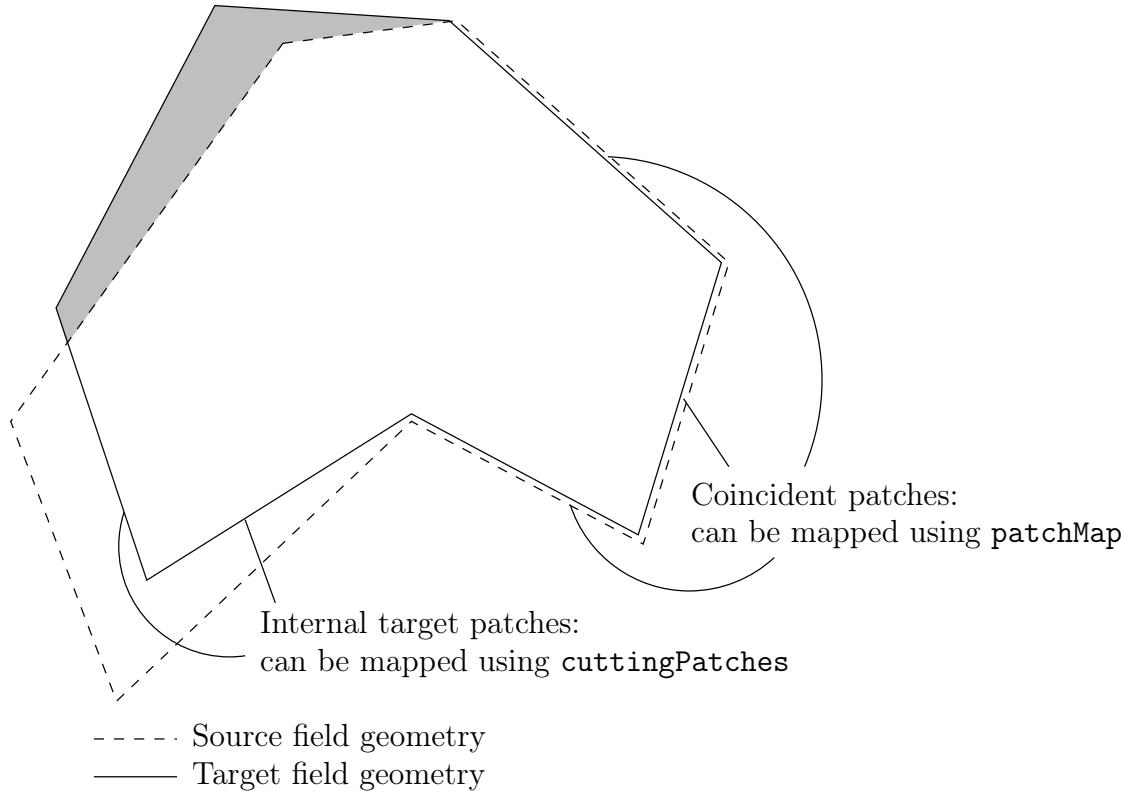


Figure 5.15: Mapping inconsistent fields

dictionary is shown below:

```

17   patchMap      (lid movingWall);
18
19   cuttingPatches  ();
20
21 // ****
mapFields <source dir>

```

5.6.3 Mapping parallel cases

If either or both of the source and target cases are decomposed for running in parallel, additional options must be supplied when executing `mapFields`:

- parallelSource if the source case is decomposed for parallel running;
- parallelTarget if the target case is decomposed for parallel running.

Chapter 6

Post-processing

This chapter describes options for post-processing with OpenFOAM. OpenFOAM is supplied with a post-processing utility `paraFoam` that uses `ParaView`, an open source visualisation application described in section 6.1.

Other methods of post-processing using third party products are offered, including `EnSight`, `Fieldview` and the post-processing supplied with `Fluent`.

6.1 `ParaView`/paraFoam graphical user interface (GUI)

The main post-processing tool provided with OpenFOAM is a reader module to run with `ParaView`, an open-source, visualization application. The module is compiled into 2 libraries, `PVFoamReader` and `vtkPVFoam` using version 5.4.0 of `ParaView` supplied with the OpenFOAM release. It is recommended that this version of `ParaView` is used, although it is possible that the latest binary release of the software will run adequately. Further details about `ParaView` can be found at <http://www.paraview.org>.

`ParaView` uses the Visualisation Toolkit (VTK) as its data processing and rendering engine and can therefore read any data in VTK format. OpenFOAM includes the `foamToVTK` utility to convert data from its native format to VTK format, which means that any VTK-based graphics tools can be used to post-process OpenFOAM cases. This provides an alternative means for using `ParaView` with OpenFOAM.

In summary, we recommend the reader module for `ParaView` as the primary post-processing tool for OpenFOAM. Alternatively OpenFOAM data can be converted into VTK format to be read by `ParaView` or any other VTK-based graphics tools.

6.1.1 Overview of `ParaView`/paraFoam

`paraFoam` is a script that launches `ParaView` using the reader module supplied with OpenFOAM. It is executed like any of the OpenFOAM utilities either by the single command from within the case directory or with the `-case` option with the case path as an argument, *e.g.:*

```
paraFoam -case <caseDir>
```

`ParaView` is launched and opens the window shown in Figure 6.1. The case is controlled from the left panel, which contains the following:

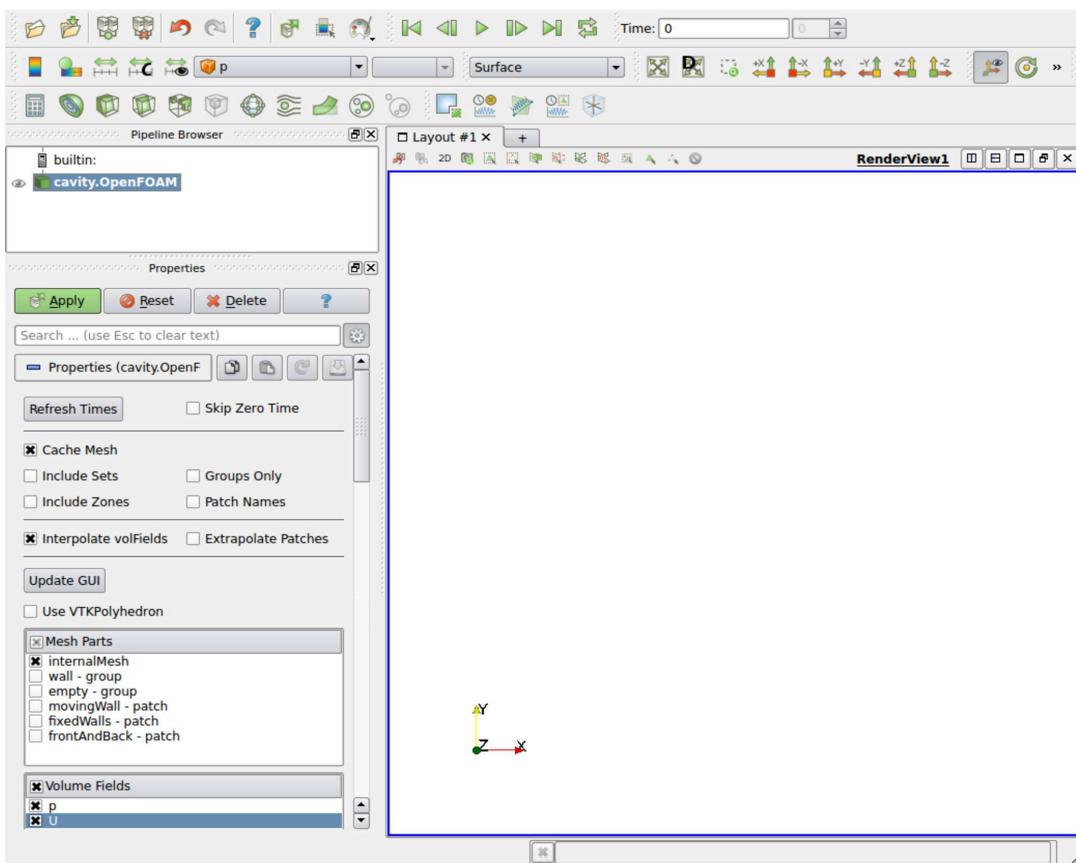


Figure 6.1: The paraFoam window

Pipeline Browser lists the *modules* opened in ParaView, where the selected modules are highlighted in blue and the graphics for the given module can be enabled/disabled by clicking the eye button alongside;

Properties panel contains the input selections for the case, such as times, regions and fields; it includes the **Display** panel that controls the visual representation of the selected module, *e.g.* colours;

Other panels can be selected from the **View** menu, including the **Information** panel which gives case statistics such as mesh geometry and size.

ParaView operates a tree-based structure in which data can be filtered from the top-level case module to create sets of sub-modules. For example, a contour plot of, say, pressure could be a sub-module of the case module which contains all the pressure data. The strength of ParaView is that the user can create a number of sub-modules and display whichever ones they feel to create the desired image or animation. For example, they may add some solid geometry, mesh and velocity vectors, to a contour plot of pressure, switching any of the items on and off as necessary.

The general operation of the system is based on the user making a selection and then clicking the green **Apply** button in the **Properties** panel. The additional buttons are: the **Reset** button which can be used to reset the GUI if necessary; and, the **Delete** button that will delete the active module.

6.1.2 The Parameters panel

The Properties window for the case module includes the Parameters panel that contains the settings for mesh, fields and global controls. The controls are described in Figure 6.2. The

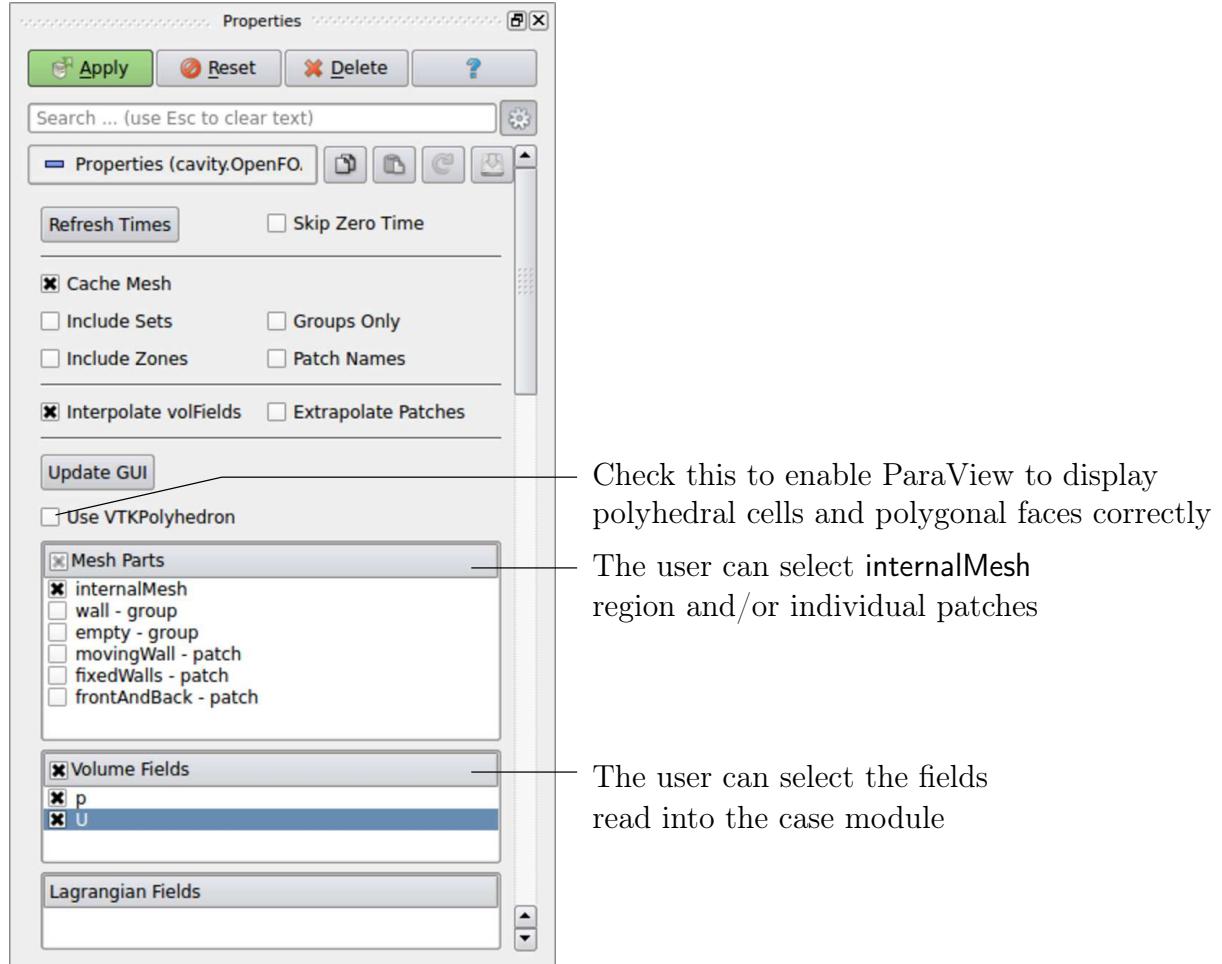


Figure 6.2: The Properties panel for the case module

user can select mesh and field data which is loaded for all time directories into ParaView. The buttons in the Current Time Controls and VCR Controls toolbars then select the time data to be displayed, as shown in section 6.1.4.

As with any operation in paraFoam, the user must click Apply after making any changes to any selections. The Apply button is highlighted in green to alert the user if changes have been made but not accepted. This method of operation has the advantage of allowing the user to make a number of selections before accepting them, which is particularly useful in large cases where data processing is best kept to a minimum.

If new data is written to time directories while the user is running ParaView, the user must load the additional time directories by checking the Refresh Times button. Where there are occasions when the case data changes on file and ParaView needs to load the changes, the user can also check the Update GUI button in the Parameters panel and apply the changes.

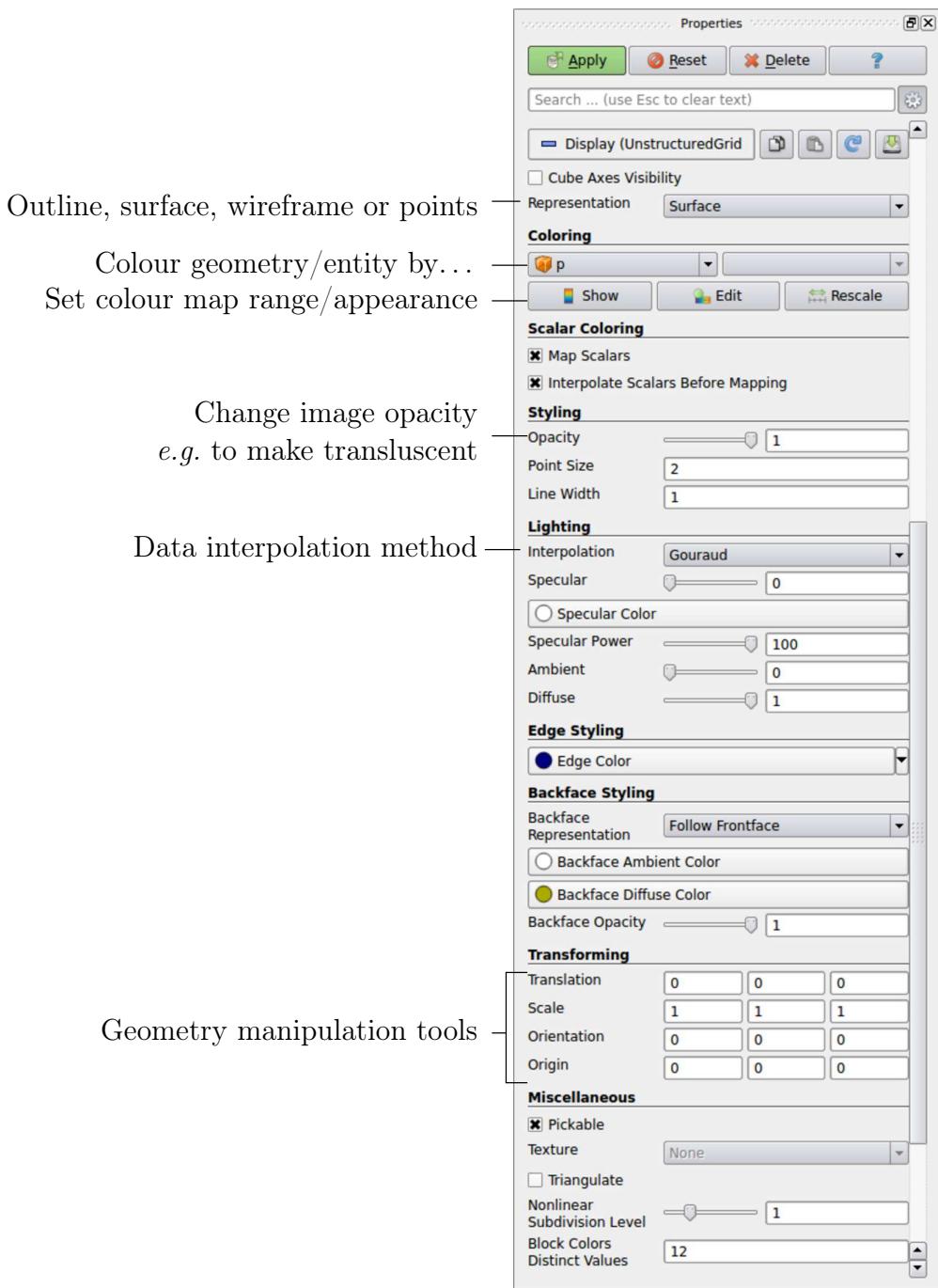


Figure 6.3: The Display panel

6.1.3 The Display panel

The Properties window contains the Display panel that includes the settings for visualising the data for a given case module. The following points are particularly important:

- the data range may not be automatically updated to the max/min limits of a field, so the user should take care to select Rescale at appropriate intervals, in particular after loading the initial case module;
- clicking the Edit Color Map button, brings up a window in which there are two panels:

1. The Color Scale panel in which the colours within the scale can be chosen. The standard blue to red colour scale for CFD can be selected by clicking Choose Preset and selecting Blue to Red Rainbow HSV.
 2. The Color Legend panel has a toggle switch for a colour bar legend and contains settings for the layout of the legend, *e.g.* font.
- the underlying mesh can be represented by selecting Wireframe in the Representation menu of the Style panel;
 - the geometry, *e.g.* a mesh (if Wireframe is selected), can be visualised as a single colour by selecting Solid Color from the Color By menu and specifying the colour in the Set Ambient Color window;
 - the image can be made translucent by editing the value in the Opacity text box (1 = solid, 0 = invisible) in the Style panel.

6.1.4 The button toolbars

ParaView duplicates functionality from pull-down menus at the top of the main window and the major panels, within the toolbars below the main pull-down menus. The displayed toolbars can be selected from Toolbars in the main View menu. The default layout with all toolbars is shown in Figure 6.4 with each toolbar labelled. The function of many of the buttons is clear from their icon and, with tooltips enabled in the Help menu, the user is given a concise description of the function of any button.

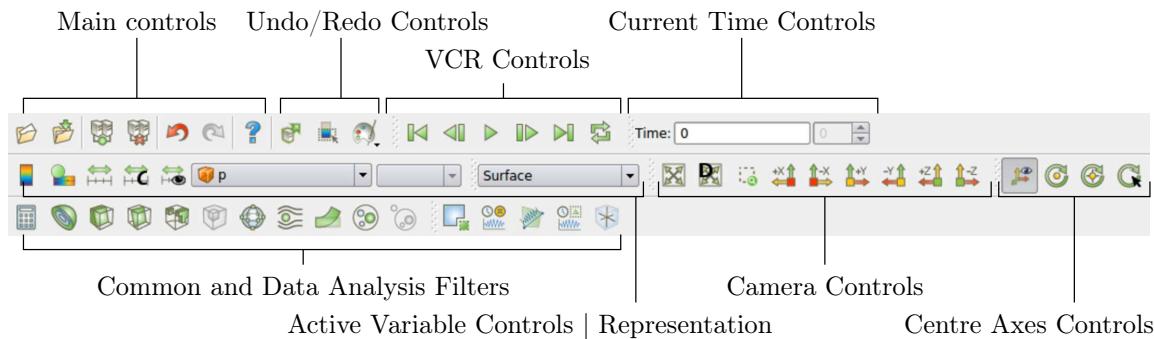


Figure 6.4: Toolbars in ParaView

6.1.5 Manipulating the view

This section describes operations for setting and manipulating the view of objects in paraFoam.

6.1.5.1 View settings

The View Settings are available in the Render View panel below the Display panel in the Properties window. Settings that are generally important only appear when the user checks the gearwheel button at the top of the Properties window, next to the search bar. These *advanced properties* include setting the background colour, where white is often a preferred choice for creating images for printed and website material.

The **Lights** button opens detailed lighting controls within the **Light Kit** panel. A separate **Headlight** panel controls the direct lighting of the image. Checking the **Headlight** button with white light colour of strength 1 seems to help produce images with strong bright colours, e.g. with an isosurface.

The **Camera Parallel Projection** is the usual choice for CFD, especially for 2D cases, and so should generally be checked. Other settings include **Cube Axes** which displays axes on the selected object to show its orientation and geometric dimensions.

6.1.5.2 General settings

The general **Settings** are selected from the **Edit** menu, which opens a general **Options** window with **General**, **Colors**, **Animations**, **Charts** and **Render View** menu items.

The **General** panel controls some default behaviour of **ParaView**. In particular, there is an **Auto Accept** button that enables **ParaView** to accept changes automatically without clicking the green **Apply** button in the **Properties** window. For larger cases, this option is generally not recommended: the user does not generally want the image to be re-rendered between each of a number of changes he/she selects, but be able to apply a number of changes to be re-rendered in their entirety once.

The **Render View** panel contains 3 sub-items: **General**, **Camera** and **Server**. The **General** panel includes the level of detail (LOD) which controls the rendering of the image while it is being manipulated, *e.g.* translated, resized, rotated; lowering the levels set by the sliders, allows cases with large numbers of cells to be re-rendered quickly during manipulation.

The **Camera** panel includes control settings for 3D and 2D movements. This presents the user with a map of rotation, translate and zoom controls using the mouse in combination with Shift- and Control-keys. The map can be edited to suit by the user.

6.1.6 Contour plots

A contour plot is created by selecting **Contour** from the **Filter** menu at the top menu bar. The filter acts on a given module so that, if the module is the 3D case module itself, the contours will be a set of 2D surfaces that represent a constant value, *i.e.* isosurfaces. The **Properties** panel for contours contains an **Isosurfaces** list that the user can edit, most conveniently by the **New Range** window. The chosen scalar field is selected from a pull down menu.

6.1.6.1 Introducing a cutting plane

Very often a user will wish to create a contour plot across a plane rather than producing isosurfaces. To do so, the user must first use the **Slice** filter to create the cutting plane, on which the contours can be plotted. The **Slice** filter allows the user to specify a cutting **Plane**, **Box** or **Sphere** in the **Slice Type** menu by a **center** and **normal/radius** respectively. The user can manipulate the cutting plane like any other using the mouse.

The user can then run the **Contour** filter on the cut plane to generate contour lines.

6.1.7 Vector plots

Vector plots are created using the **Glyph** filter. The filter reads the field selected in **Vectors** and offers a range of **Glyph Types** for which the **Arrow** provides a clear vector plot images.

Each glyph has a selection of graphical controls in a panel which the user can manipulate to best effect.

The remainder of the **Properties** panel contains mainly the **Scale Mode** menu for the glyphs. The most common options are **Scale Mode** are: **Vector**, where the glyph length is proportional to the vector magnitude; and, **Off** where each glyph is the same length. The **Set Scale Factor** parameter controls the base length of the glyphs.

6.1.7.1 Plotting at cell centres

Vectors are by default plotted on cell vertices but, very often, we wish to plot data at cell centres. This is done by first applying the **Cell Centers** filter to the case module, and then applying the **Glyph** filter to the resulting cell centre data.

6.1.8 Streamlines

Streamlines are created by first creating tracer lines using the **Stream Tracer** filter. The tracer **Seed** panel specifies a distribution of tracer points over a **Line Source** or **Point Cloud**. The user can view the tracer source, *e.g.* the line, but it is displayed in white, so they may need to change the background colour in order to see it.

The distance the tracer travels and the length of steps the tracer takes are specified in the text boxes in the main **Stream Tracer** panel. The process of achieving desired tracer lines is largely one of trial and error in which the tracer lines obviously appear smoother as the step length is reduced but with the penalty of a longer calculation time.

Once the tracer lines have been created, the **Tubes** filter can be applied to the **Tracer** module to produce high quality images. The tubes follow each tracer line and are not strictly cylindrical but have a fixed number of sides and given radius. When the number of sides is set above, say, 10, the tubes do however appear cylindrical, but again this adds a computational cost.

6.1.9 Image output

The simplest way to output an image to file from ParaView is to select **Save Screenshot** from the **File** menu. On selection, a window appears in which the user can select the resolution for the image to save. There is a button that, when clicked, locks the aspect ratio, so if the user changes the resolution in one direction, the resolution is adjusted in the other direction automatically. After selecting the pixel resolution, the image can be saved. To achieve high quality output, the user might try setting the pixel resolution to 1000 or more in the *x*-direction so that when the image is scaled to a typical size of a figure in an A4 or US letter document, perhaps in a PDF document, the resolution is sharp.

6.1.10 Animation output

To create an animation, the user should first select **Save Animation** from the **File** menu. A dialogue window appears in which the user can specify a number of things including the image resolution. The user should specify the resolution as required. The other noteworthy setting is number of frames per timestep. While this would intuitively be set to 1, it can be set to a larger number in order to introduce more frames into the animation artificially.

This technique can be particularly useful to produce a slower animation because some movie players have limited speed control, particularly over `mpeg` movies.

On clicking the **Save Animation** button, another window appears in which the user specifies a file name *root* and file format for a set of images. On clicking **OK**, the set of files will be saved according to the naming convention “`<fileRoot>_<imageNo>.<fileExt>`”, *e.g.* the third image of a series with the file root “`animation`”, saved in `jpg` format would be named “`animation_0002.jpg`” (`<imageNo>` starts at 0000).

Once the set of images are saved the user can convert them into a movie using their software of choice. One option is to use the built in `foamCreateVideo` script from the command line whose usage is shown with the `-help` option.

6.2 Post-processing command line interface (CLI)

In OpenFOAM version 4.0, the ‘non-GUI’ post-processing tools have been unified within a single command line interface (CLI). Post-processing functionality includes data processing, sampling (*e.g.* probes, graph plotting) visualisation, case control and run-time I/O. Functionality can be executed by:

- conventional *post-processing*, a data processing activity that occurs *after* a simulation has run;
- *run-time processing*, data processing that is performed *during* the running of a simulation.

Both approaches have advantages. Conventional post-processing allows the user to choose how to analyse data after the results are obtained. Run-time processing offers greater flexibility because it has access to *all* the data in the database of the run at all times, rather than just the data written during the simulation. It also allows the user to monitor processed data during a simulation and provides a greater level of convenience because the processed results can be available immediately to the user when the simulation ends.

There are 3 methods of post-processing that cover the options described above

- Every solver, *e.g.* `simpleFoam` can be configured to include run-time processing.
- The `postProcess` utility provides conventional post-processing of data written by a simulation.
- Every solver can be run with the `-postProcess` option, which *only* executes post-processing, but with additional access to data available on the database for the particular solver.

6.2.1 Post-processing functionality

All modes of post-processing access the same functionality implemented in OpenFOAM in the *function object* framework. Function objects can be listed by running a solver with the `-listFunctionObjects` option, *e.g.*

```
simpleFoam -listFunctionObjects
```

The list represents the underlying post-processing functionality. Almost all the functionality is packaged into a set of configured tools that are conveniently integrated within the post-processing CLI. Those tools are located in `$FOAM_ETC/caseDicts/postProcessing` and are listed by running `postProcess` with the `-list` option.

```
postProcess -list
```

This produces a list of tools that are described in the following sections.

6.2.1.1 Field calculation

`CourantNo` Calculates the Courant Number field from the flux field.

`Lambda2` Calculates and writes the second largest eigenvalue of the sum of the square of the symmetrical and anti-symmetrical parts of the velocity gradient tensor.

`MachNo` Calculates the Mach Number field from the velocity field.

`PecletNo` Calculates the Peclet Number field from the flux field.

`Q` Calculates the second invariant of the velocity gradient tensor.

`R` Calculates the Reynolds stress tensor field and stores it on the database.

`XiReactionRate` Writes the turbulent flame-speed and reaction-rate `volScalarFields` for the Xi-based combustion models.

`add` Add a list of fields.

`components` Writes the component scalar fields (*e.g.* `Ux`, `Uy`, `Uz`) of a field (*e.g.* `U`).

`div` Calculates the divergence of a field.

`enstrophy` Calculates the enstrophy of the velocity field.

`flowType` Calculates and writes the `flowType` of velocity field where: `-1` = rotational flow; `0` = simple shear flow; `+1` = planar extensional flow.

`grad` Calculates the gradient of a field.

`mag` Calculates the magnitude of a field.

`magSqr` Calculates the magnitude-squared of a field.

`randomise` Adds a random component to a field, with a specified perturbation magnitude.

`streamFunction` Writes the stream-function `pointScalarField`, calculated from the specified flux `surfaceScalarField`.

`subtract` From the first field, subtracts the remaining fields in the list.

`turbulenceFields` Calculates specified turbulence fields and stores it on the database.

`vorticity` Calculates the vorticity field, i.e. the curl of the velocity field.

wallShearStress Calculates the shear stress at wall patches, outputting the data as a **volVectorField**.

wallHeatFlux Calculates the heat flux at wall patches, outputting the data as a **volVectorField**.

writeCellCentres Writes the cell-centres **volVectorField** and the three component fields as **volScalarFields**; useful for post-processing thresholding.

writeCellVolumes Writes the cell-volumes **volScalarField**

writeObjects Writes out specified objects, *e.g.* fields, stored on the case database.

yPlus Calculates the turbulence $y+$, outputting the data as a **yPlus** field.

6.2.1.2 Flow rate calculation

flowRateFaceZone Calculates the flow rate through a specified face zone by summing the flux on patch faces. For solvers where the flux is volumetric, the flow rate is volumetric; where flux is mass flux, the flow rate is mass flow rate.

flowRatePatch Calculates the flow rate through a specified patch by summing the flux on patch faces. For solvers where the flux is volumetric, the flow rate is volumetric; where flux is mass flux, the flow rate is mass flow rate.

volFlowRateSurface Calculates volumetric flow rate through a specified triangulated surface by interpolating velocity onto the triangles and integrating over the surface area. Triangles need to be small (\leq cell size) for an accurate result.

6.2.1.3 Forces and force coefficients

forceCoeffsCompressible Calculates lift, drag and moment coefficients by summing forces on specified patches for a case where the solver is compressible (pressure is in units $M/(LT^2)$, *e.g.* Pa).

forceCoeffsIncompressible Calculates lift, drag and moment coefficients by summing forces on specified patches for a case where the solver is incompressible (pressure is kinematic, *e.g.* m^2/s^2).

forcesCompressible Calculates pressure and viscous forces over specified patches for a case where the solver is compressible (pressure is in units $M/(LT^2)$, *e.g.* Pa).

forcesIncompressible Calculates pressure and viscous forces over specified patches for a case where the solver is incompressible (pressure is kinematic, *e.g.* m^2/s^2).

6.2.1.4 Sampling for graph plotting

singleGraph Writes graph data for specified fields along a line, specified by start and end points.

6.2.1.5 Lagrangian data

dsmcFields Calculate intensive fields `UMean`, `translationalT`, `internalT`, `overallT` from averaged extensive fields from a DSMC calculation.

6.2.1.6 Monitoring minima and maxima

cellMax Writes out the maximum cell value for one or more fields.

cellMin Writes out the minimum cell value for one or more fields.

faceMax Writes out the maximum face value for one or more fields.

faceMin Writes out the minimum face value for one or more fields.

minMaxComponents Writes out the minimum and maximum values, by component for non-scalar fields, and the locations where they occur.

minMaxMagnitude Writes out the minimum and maximum values, by magnitude for non-scalar fields, and the locations where they occur.

6.2.1.7 Numerical data

residuals For specified fields, writes out the initial residuals for the first solution of each time step; for non-scalar fields (*e.g.* vectors), writes the largest of the residuals for each component (*e.g.* x, y, z).

6.2.1.8 Pressure tools

pressureDifferencePatch Calculates the difference between the average pressures on 2 specified patches.

pressureDifferenceSurface Interpolates pressures onto 2 specified triangulated surfaces and calculates the difference between the average pressures.

staticPressure Calculates the pressure field in units $M/(LT^2)$ (*e.g.* Pa) from kinematic pressure by scaling by a specified density.

totalPressureCompressible Calculates the total pressure field for a case where the solver is compressible (pressure is in units $M/(LT^2)$, *e.g.* Pa).

totalPressureIncompressible Calculates the total pressure field for a case where the solver is incompressible (pressure is kinematic, *e.g.* m^2/s^2).

6.2.1.9 Probes

boundaryCloud Writes out values of fields at a cloud of points, interpolated to specified boundary patches.

interfaceHeight Reports the height of the interface above a set of locations. For each location, it writes the vertical distance of the interface above both the location and the lowest boundary. It also writes the point on the interface from which these heights are computed.

`internalCloud` Writes out values of fields interpolated to a specified cloud of points.

`probes` Writes out values of fields from cells nearest to specified locations.

6.2.1.10 ‘Pluggable’ solvers

`scalarTransport` Solves a transport equation for a scalar field.

6.2.1.11 Visualisation tools

`streamlines` Writes out files of streamlines with interpolated field data in VTK format.

`surfaces` Writes out surface files with interpolated field data in VTK format, *e.g.* cutting planes, iso-surfaces and patch boundary surfaces.

6.2.2 Run-time data processing

When a user wishes to process data during a simulation, they need to configure the case accordingly. The configuration process is as follows, using an example of monitoring flow rate at an outlet patch named `outlet`.

Firstly, the user should include the `flowRatePatch` function in `functions` sub-dictionary in the case `controlDict` file, using the `#includeFunc` directive.

```
functions
{
    #includeFunc  flowRatePatch
    ... other function objects here ...
}
```

That will include the functionality in the `flowRatePatch` configuration file, located in the directory hierarchy beginning with `$FOAM_ETC/caseDicts/postProcessing`.

The configuration of `flowRatePatch` requires the name of the patch to be supplied. **Option 1** for doing this is that the user copies the `flowRatePatch` file into their case `system` directory by

```
cp $FOAM_ETC/caseDicts/postProcessing/flowRate/flowRatePatch system
```

and edits the patch name, in the copied file, as `outlet`. When the solver is run, it will pick up an included function in the local case `system` directory, in precedence over `$FOAM_ETC/caseDicts/postProcessing`. The flow rate through the patch will be calculated and written out into a file within a directory named `postProcessing`.

Option 2 for specifying the patch name is to provide the name as an argument to the `flowRatePatch` in the `#includeFunc` directive, using the syntax `keyword=entry`.

```
functions
{
    #includeFunc  flowRatePatch(name=outlet)
    ... other function objects here ...
}
```

In the case where the keyword is `field` or `fields`, only the entry is needed when specifying an argument to a function. For example, if the user wanted to calculate and write out the magnitude of velocity into time directories during a simulation they could simply add the following to the `functions` sub-dictionary in `controlDict`.

```
functions
{
    #includeFunc mag(U)
    ... other function objects here ...
}
```

This works because the function's argument `U` is represented by the keyword `field`, see `$FOAM_ETC/caseDicts/postProcessing/fields/mag`.

Some functions require the setting of many parameters, *e.g.* `forces`, `forceCoeffs`, `surfaces`, *etc.* For those functions, it is more reliable and convenient to copy and configure the function using option 1 (above) rather than through arguments.

6.2.3 The postProcess utility

The user can execute post-processing functions after the simulation is complete using the `postProcess` utility. Let us illustrate the use of `postProcess` using the `pitzDaily` case from the tutorials directory. The case can be copied, *e.g.* into the user's `run` directory; the user can then go into the case directory, generate the mesh with `blockMesh` and then run `simpleFoam`

```
run
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
cd pitzDaily
blockMesh
simpleFoam
```

Now the user can run execute post-processing functions with `postProcess`. The `-help` option provides a summary of its use.

```
postProcess -help
```

Simple functions like `mag` can be executed using the `-func` option; text on the command line generally needs to be quoted ("...") if it contains punctuation characters.

```
postProcess -func "mag(U)"
```

This operation calculates and writes the field of magnitude of velocity into a file named `mag(U)` in each time directory. Similarly, the `flowRatePatch` example can be executed using `postProcess`.

```
postProcess -func "flowRatePatch(name=outlet)"
```

Let us say the user now wants to calculate total pressure = $p + |U|^2/2$ for incompressible flow with kinematic pressure, p . The function is available, named `totalPressureIncompressible`, which the user could attempt first to run as follows.

```
postProcess -func totalPressureIncompressible
```

This returns the following error message.

```
--> FOAM Warning : functionObject pressure: Cannot find required field p
```

The error message is telling the user that the pressure field p is not loaded; the same is true of the velocity field U . For the function to work, both fields can be loaded as **comma separated** arguments.

```
postProcess -func "totalPressureIncompressible(p,U)"
```

Alternatively the user can load a **space separated** list of fields using the **-fields** option, which the function can access.

```
postProcess -fields "(p U)" -func totalPressureIncompressible
```

Both options work effectively because the pressure and velocity data is available directly from the files, p and U .

6.2.4 Solver post-processing

A more complex example is calculating wall shear stress using the **wallShearStress** function.

```
postProcess -fields "(p U)" -func wallShearStress
```

Even loading relevant fields, the post-processing fails with the following message.

```
--> FOAM FATAL ERROR:  
Unable to find turbulence model in the database
```

The message is telling us that the **postProcess** utility has not constructed the necessary models that the solver, **simpleFoam**, used when running the simulation, *i.e.* a turbulence model. This is a situation where we need to post-process (as opposed to run-time process) using the solver with the **-postProcess** option so that the modelling will be available that the post-processing function needs. Help for this operation can be printed with the following command.

```
simpleFoam -postProcess -help
```

It can be seen that the options for a solver with **-postProcess** are the same as when running **postProcess** utility. This means that the **-func** option can be used to execute the **wallShearStress** function effectively.

```
simpleFoam -postProcess -func wallShearStress
```

Note that no fields need to be supplied, either by function arguments " (p,U) " or using "**-fields** $(p\ U)$ ", because **simpleFoam** itself constructs and stores the required fields. Functions can also be selected by the **#includeFunc** directive in **functions** in the **controlDict** file, instead of the **-func** option.

6.3 Sampling and monitoring data

There are a set of general post-processing functions for sampling data across the domain for graphs and visualisation. Several functions also provide data in a single file, in the form of time versus values, that can be plotted onto graphs. This time-value data can be monitored during a simulation with the `foamMonitor` script.

6.3.1 Probing data

The functions for probing data are `boundaryCloud`, `internalCloud` and `probes` as listed in section 6.2.1.9. All functions work on the basis that the user provides some point locations and a list of fields, and the function writes out values of the fields at those locations. The differences between the cloud and probe functionality are as follows.

- `probes` identifies the nearest cells to the probe locations and writes out the cell values; data is written into a single file in time-value format, suitable for plotting a graph.
- `boundaryCloud` and `internalCloud` interpolate field data to the probe locations, with the locations being snapped onto boundaries for `boundaryCloud`; data sets are written to separate files at scheduled write times (like fields). data.

Generally `probes` is more suitable for monitoring values at smaller numbers of locations, whereas the cloud functions are typically for sampling at large numbers of locations.

As an example, the user could use the `pitzDaily` case set up in section 6.2.3. The `probes` function is best configured by copying the file to the local system directory as follows.

```
cp -r $FOAM_ETC/caseDicts/postProcessing/probes probes system
```

The user can modify the `probeLocations` in the `probes` file as follows.

```
12 #includeEtc "caseDicts/postProcessing/probes/probes.cfg"
13
14 fields (p U);
15 probeLocations
16 (
17     (0.01 0 0)
18 );
19 // ****
20 // *****
```

The configuration is completed by adding the `#includeFunc` directive to `functions` in the the `controlDict` file.

```
functions
{
    #includeFunc probes
    ... other function objects here ...
}
```

When `simpleFoam` is run, time-value data is written into `p` and `U` files in `postProcessing/probes/0`.

6.3.2 Sampling for graphs

The `singleGraph` function samples data for graph plotting. To use it, the `singleGraph` file should be copied into the `system` directory to be configured. We will configure it here using the `pitzDaily` case as before.

```
cp -r $FOAM_ETC/caseDicts/postProcessing/graphs/singleGraph system
```

The start and end points of the line, along which data is sampled, should be edited; the entries below provide a vertical line across the full height of the geometry 0.01 m beyond the back step.

```

13
14   start    (0.01  0.025 0);
15   end      (0.01 -0.025 0);
16   fields   (U p);
17
18   // Sampling and I/O settings
19   #includeEtc "caseDicts/postProcessing/graphs/sampleDict.cfg"
20
21   // Override settings here, e.g.
22   // setConfig { type midPoint; }
23
24   // Must be last entry
25   #includeEtc "caseDicts/postProcessing/graphs/graph.cfg"
26
27   // ****

```

The configuration is completed by adding the `#includeFunc` directive to functions in the the `controlDict` file.

```
functions
{
    #includeFunc  singleGraph
    ... other function objects here ...
}
```

`simpleFoam` can be then run; try running simply with the `-postProcess` option. Distance-value data is written into files in time directories within `postProcessing/singleGraph`. The user can quickly display the data for x -component of velocity, U_x in the last time 296, by running `gnuplot` and plotting values.

```
gnuplot
gnuplot> set style data linespoints
gnuplot> plot "postProcessing/singleGraph/296/line_U.xy" u 2:1
```

This produces the graph shown in Figure 6.5. The formatting of the graph is specified in configuration files in `$FOAM_ETC/caseDicts/postProcessing/graphs`. The `sampleDict.cfg` file in that directory contains a `setConfig` sub-dictionary as shown below.

```

8
9   interpolationScheme cellPoint;
10
11  setFormat    raw;
12
13  setConfig
14  {
15      type      uniform;    // midPoint, midPointAndFace
16      axis      distance;  // x, y, z, xyz
17      nPoints  100;
18  }
19
20  // ****

```

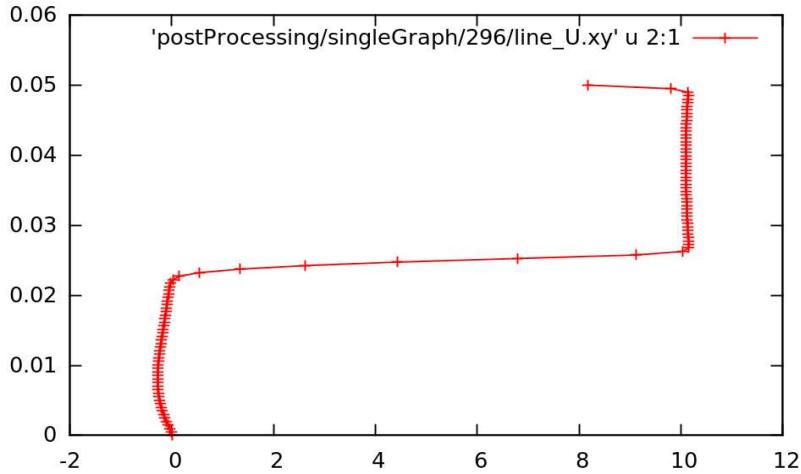


Figure 6.5: Graph of U_x at $x = 0.01$, uniform sampling

This shows that the sampling is at `uniform` distribution of 100 points and that the axis labelling is based on the `distance` from the start point. Instead the user may wish to sample at the mid-point of each cell, using the `y`-ordinate for the axis labelling. This can be done by override the settings in `setConfig` in their `singleGraph` file, as shown below.

```

13   start    (0.01 -0.025 0);
14   end      (0.01 0.025 0);
15   fields   (U p);
16
17
18 // Sampling and I/O settings
19 #includeEtc "caseDicts/postProcessing/graphs/sampleDict.cfg"
20
21 // Override settings here, e.g.
22 setConfig
23 {
24     type midPoint;
25     axis y;
26 }
27
28 // Must be last entry
29 #includeEtc "caseDicts/postProcessing/graphs/graph.cfg"
30
31 // ****

```

Running `simpleFoam` produces the graph in Figure 6.6.

6.3.3 Sampling for visualisation

The `surfaces` and `streamlines` functions can be used to generate files for visualisation. The use of `streamlines` is already configured in the `pitzDaily` case. The `surfaces` function is configured by copying the `surfaces` file to the `system` directory.

```
cp -r $FOAM_ETC/caseDicts/postProcessing/visualization/surfaces system
```

The file is configured with a set of example surfaces that includes cutting planes, isosurfaces and patch surfaces. We can edit the file to produce a cutting plane along the `pitzDaily` geometry, normal to the `z`-direction.

```

16
17 #includeEtc "caseDicts/postProcessing/visualization/surfaces.cfg"
18

```

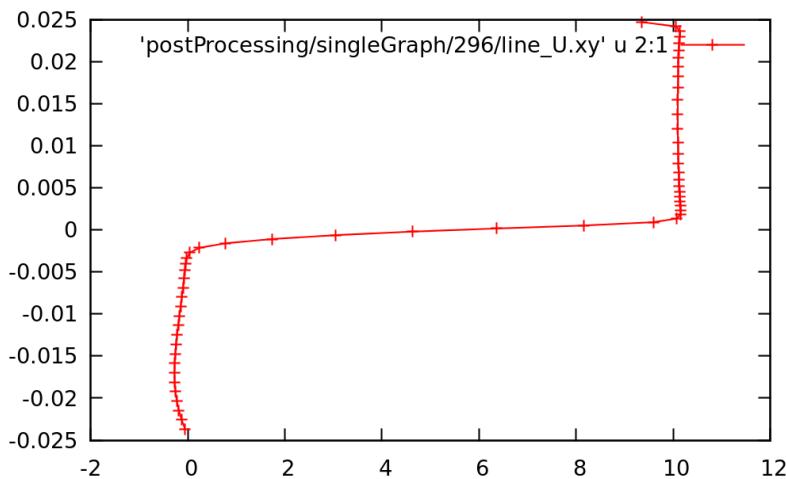


Figure 6.6: Graph of U_x at $x = 0.01$, mid-point sampling

```

19   fields      (p U);
20
21   surfaces
22   (
23     zNormal
24     {
25       $cuttingPlane;
26       pointAndNormalDict
27       {
28         normalVector $z;
29       }
30     }
31   );
32 // ****
33

```

The function can be included as normal from the by adding the `#includeFunc` directive to `functions` in the the *controlDict* file. Alternatively, the user could test running the function using the solver post-processing by the following command.

```
simpleFoam -postProcess -func surfaces
```

This produces VTK format files of the cutting plane with pressure and velocity data in time directories in the the *postProcessing/surfaces* directory. The user can display the cutting plane by opening ParaView (type `paraview`), then doing `File->Open` and selecting one of the files, e.g. *postProcessing/surfaces/296/U_zNormal.vtk* as shown in Figure 6.7.

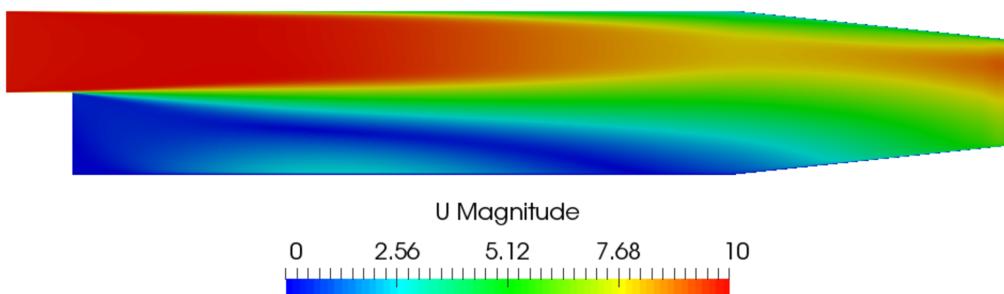


Figure 6.7: Cutting plane with velocity

6.3.4 Live monitoring of data

Functions like `probes` produce a single file of time-value data, suitable for graph plotting. When the function is executed during a simulation, the user may wish to monitor the data live on screen. The `foamMonitor` script enables this; to discover its functionality, the user run it with the `-help` option. The help option includes an example of monitoring residuals that we can demonstrate in this section.

Firstly, include the `residuals` function in the *controlDict* file.

```
functions
{
#includeFunc  residuals
... other function objects here ...
}
```

The default fields whose residuals are captured are p and U . Should the user wish to configure other fields, they should make copy the `residuals` file in their `system` and edit the `fields` entry accordingly. All functions files are within the `$FOAM_ETC/caseDicts` directory; the `residuals` file, for example, can be located by typing:

```
find $FOAM_ETC -name residuals
```

It is advisable to delete the `postProcessing` directory to avoid duplicate files for each function. The user can delete the directory, then run `simpleFoam` in the background.

```
rm -rf postProcessing
simpleFoam > log &
```

The user should then run `foamMonitor` using the `-l` option for a log scale y -axis on the `residuals` file as follows. If the command is executed before the simulation is complete, they can see the graph being updated live.

```
foamMonitor -l postProcessing/residuals/0/residuals.dat
```

It produces the graph of residuals for pressure and velocity in Figure 6.8.

6.4 Third-Party post-processing

OpenFOAM includes the following applications for converting data to formats for post-processing with several third-party tools. For EnSight, it additionally includes a reader module, described in the next section.

`foamDataToFluent` Translates OpenFOAM data to Fluent format.

`foamToEnsight` Translates OpenFOAM data to EnSight format.

`foamToEnsightParts` Translates OpenFOAM data to Ensight format. An Ensight part is created for each `cellZone` and patch.

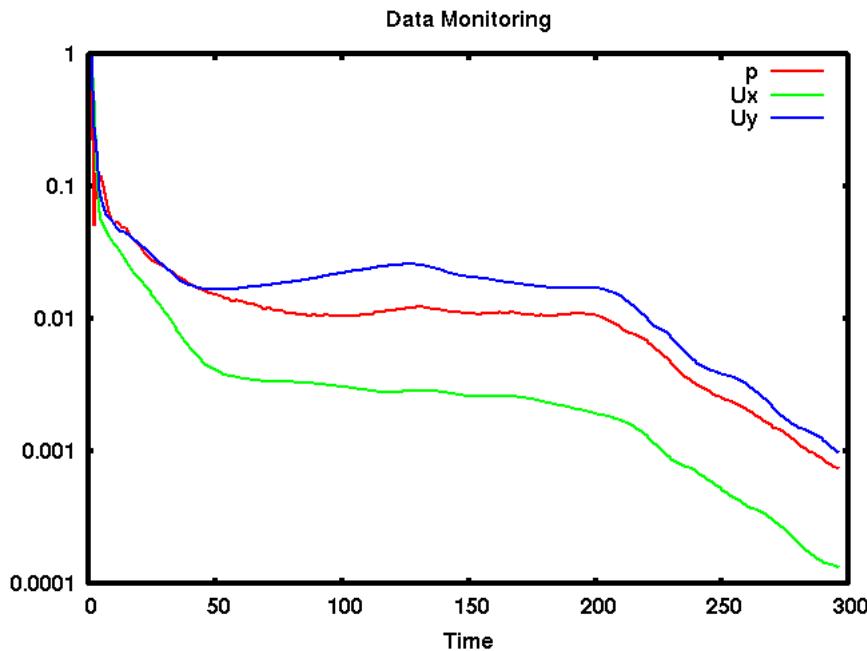


Figure 6.8: Live plot of residuals with `foamMonitor`

`foamToGMV` Translates foam output to GMV readable files.

`foamToTecplot360` Tecplot binary file format writer.

`foamToTetDualMesh` Converts `polyMesh` results to `tetDualMesh`.

`foamToVTK` Legacy VTK file format writer.

`smapToFoam` Translates a STAR-CD SMAP data file into OpenFOAM field format.

6.4.1 Post-processing with EnSight

OpenFOAM offers the capability for post-processing OpenFOAM cases with EnSight, with a choice of 2 options:

- converting the OpenFOAM data to EnSight format with the `foamToEnsight` utility;
- reading the OpenFOAM data directly into EnSight using the `ensight74FoamExec` module.

6.4.1.1 Converting data to EnSight format

The `foamToEnsight` utility converts data from OpenFOAM to EnSight file format. For a given case, `foamToEnsight` is executed like any normal application. `foamToEnsight` creates a directory named `Ensight` in the case directory, *deleting any existing Ensight directory in the process*. The converter reads the data in all time directories and writes into a case file and a set of data files. The case file is named `Ensight_Case` and contains details of the data file names. Each data file has a name of the form `Ensight_nn.ext`, where *nn* is an incremental counter starting from 1 for the first time directory, 2 for the second and so on and `ext` is a file extension of the name of the field that the data refers to, as described in the case file,

e.g.T for temperature, *mesh* for the mesh. Once converted, the data can be read into EnSight by the normal means:

1. from the EnSight GUI, the user should select Data (Reader) from the File menu;
2. the appropriate *EnSight_Case* file should be highlighted in the Files box;
3. the Format selector should be set to Case, the EnSight default setting;
4. the user should click (Set) Case and Okay.

6.4.1.2 The ensightFoamReader reader module

EnSight provides the capability of using a user-defined module to read data from a format other than the standard EnSight format. OpenFOAM includes its own reader module `ensightFoamReader` that is compiled into a library named `libuserd-foam`. It is this library that EnSight needs to use which means that it must be able to locate it on the filing system as described in the following section.

In order to run the EnSight reader, it is necessary to set some environment variables correctly. The settings are made in the `bashrc` (or `cshrc`) file in the `$WM_PROJECT_DIR/etc/apps/ensightFoam` directory. The environment variables associated with EnSight are prefixed by `$CEI_` or `$ENSIGHT7_` and listed in Table 6.1. With a standard user setup, only `$CEI_HOME` may need to be set manually, to the path of the EnSight installation.

Environment variable	Description and options
<code>\$CEI_HOME</code>	Path where EnSight is installed, eg <code>/usr/local/ensight</code> , added to the system path by default
<code>\$CEI_ARCH</code>	Machine architecture, from a choice of names corresponding to the machine directory names in <code>\$CEI_HOME/ensight74/machines</code> ; default settings include <code>linux_2.4</code> and <code>sgi_6.5_n32</code>
<code>\$ENSIGHT7_READER</code>	Path that EnSight searches for the user defined libuserd-foam reader library, set by default to <code>\$FOAM_LIBBIN</code>
<code>\$ENSIGHT7_INPUT</code>	Set by default to <code>dummy</code>

Table 6.1: Environment variable settings for EnSight.

The principal difficulty in using the EnSight reader lies in the fact that EnSight expects that a case to be defined by the contents of a particular file, rather than a directory as it is in OpenFOAM. Therefore in following the instructions for the using the reader below, the user should pay particular attention to the details of case selection, since EnSight does not permit selection of a directory name.

1. from the EnSight GUI, the user should select Data (Reader) from the File menu;
2. The user should now be able to select the OpenFOAM from the Format menu; if not, there is a problem with the configuration described above.
3. The user should find their case directory from the File Selection window, highlight one of top 2 entries in the Directories box ending in `/.` or `/..` and click (Set) Geometry.

4. The path field should now contain an entry for the case. The (Set) Geometry text box should contain a '/'.
5. The user may now click **Okay** and EnSight will begin reading the data.
6. When the data is read, a new **Data Part Loader** window will appear, asking which part(s) are to be read. The user should select **Load all**.
7. When the mesh is displayed in the EnSight window the user should close the **Data Part Loader** window, since some features of EnSight will not work with this window open.

Chapter 7

Models and physical properties

OpenFOAM includes a large range of solvers each designed for a specific class of problem. The equations and algorithms differ from one solver to another so that the selection of a solver involves the user making some initial choices on the modelling for their particular case. The choice of solver typically involves scanning through their descriptions in section 3.5 to find the one suitable for the case. It ultimately determines many of the parameters and physical properties required to define the case but leaves the user with some modelling options that can be specified at runtime through the entries in dictionary files in the *constant* directory of a case. This chapter deals with many of the more common models and associated properties that must be specified at runtime.

7.1 Thermophysical models

Thermophysical models are concerned with energy, heat and physical properties. The *thermophysicalProperties* dictionary is read by any solver that uses the thermophysical model library. A thermophysical model is constructed in OpenFOAM as a pressure-temperature $p - T$ system from which other properties are computed. There is one compulsory dictionary entry called **thermoType** which specifies the package of thermophysical modelling that is used in the simulation. OpenFOAM includes a large set of pre-compiled combinations of modelling, built within the code using C++ templates. This coding approach assembles thermophysical modelling packages beginning with the equation of state and then adding more layers of thermophysical modelling that derive properties from the previous layer(s). The keyword entries in **thermoType** reflects the multiple layers of modelling and the underlying framework in which they combined. Below is an example entry for **thermoType**:

```
thermoType
{
    type          hePsiThermo;
    mixture       pureMixture;
    transport     const;
    thermo        hConst;
    equationOfState perfectGas;
    specie        specie;
    energy         sensibleEnthalpy;
}
```

The keyword entries specify the choice of thermophysical models, *e.g.* constant `transport` (constant viscosity, thermal diffusion), Perfect Gas `equationOfState`, *etc.* In addition there is a keyword entry named `energy` that allows the user to specify the form of energy to be used in the solution and thermodynamics. The following sections explains the entries and options in the `thermoType` package.

7.1.1 Thermophysical and mixture models

Each solver that uses thermophysical modelling constructs an object of a specific thermophysical model class. The model classes are listed below.

`psiThermo` Thermophysical model for fixed composition, based on compressibility $\psi = (RT)^{-1}$, where R is Gas Constant and T is temperature. The solvers that construct `psiThermo` include the *compressible* family of solvers (`sonicFoam`, `simpleFoam`, *etc.*, excluding `rhoPorousSimpleFoam`) and `uncoupledKinematicParcelFoam` and `coldEngineFoam`.

`rhoThermo` Thermophysical model for fixed composition, based on density ρ . The solvers that construct `rhoThermo` include the *heatTransfer* family of solvers (`buoyantSimpleFoam`, `CHT` solvers, *etc.*, excluding Boussinesq solvers) and `rhoPorousSimpleFoam`, `twoPhaseEulerFoam` and `thermoFoam`.

`psiReactionThermo` Thermophysical model for reacting mixture, based on ψ . The solvers that construct `psiReactionThermo` include many of the *combustion* solvers, *e.g.* `sprayFoam`, `chemFoam`, `fireFoam` and `reactingFoam`, and some *lagrangian* solvers, *e.g.* `coalChemistryFoam` and `reactingParcelFilmFoam`.

`psiReactionThermo` Thermophysical model for combustion, based on compressibility of unburnt gas ψ_u . The solvers that construct `psiReactionThermo` include *combustion* solvers that model combustion based on laminar flame speed and regressing variable, *e.g.* `XiFoam`, `PDRFoam` and `engineFoam`.

`rhoReactionThermo` Thermophysical model for reacting mixture, based on ρ . The solvers that construct `rhoReactionThermo` include some *combustion* solvers, *e.g.* `rhoReactingFoam`, `rhoReactingBuoyantFoam`, and some *lagrangian* solvers, *e.g.* `reactingParcelFoam` and `simpleReactingParcelFoam`.

`multiphaseMixtureThermo` Thermophysical models for multiple phases. The solvers that construct `multiphaseMixtureThermo` include compressible *multiphase* interface-capturing solvers, *e.g.* `compressibleInterFoam`, and `compressibleMultiphaseInterFoam`.

Note: a new generalised `fluidThermo` model currently exists that allows the user to specify the thermophysical model through the `type` entry (described below) at run-time. Currently only the `rhoSimpleFoam` and `rhoPimpleFoam` solvers construct the generalised `fluidThermo`.

The `type` keyword specifies the underlying thermophysical model. Options are listed below.

- `hePsiThermo`: for solvers that construct `psiThermo` and `psiReactionThermo`.

- `heRhoThermo`: for solvers that construct `rhoThermo`, `rhoReactionThermo` and `multiPhaseMixtureThermo`.
- `heheuPsiThermo`: for solvers that construct `psiReactionThermo`.

The `mixture` specifies the mixture composition. The option typically used for thermophysical models without reactions is `pureMixture`, which represents a mixture with fixed composition. When `pureMixture` is specified, the thermophysical models coefficients are specified within a sub-dictionary called `mixture`.

For mixtures with variable composition, required by thermophysical models with reactions, the `reactingMixture` option is used. Species and reactions are listed in a chemistry file, specified by the `foamChemistryFile` keyword. The `reactingMixture` model then requires the thermophysical models coefficients to be specified for each specie within sub-dictionaries named after each specie, *e.g.* `O2`, `N2`.

For combustion based on laminar flame speed and regress variables, constituents are a set of mixtures, such as `fuel`, `oxidant` and `burntProducts`. The available mixture models for this combustion modelling are `homogeneousMixture`, `inhomogeneousMixture` and `veryInhomogeneousMixture`.

Other models for variable composition are `egrMixture`, `multiComponentMixture` and `singleStepReactingMixture`.

7.1.2 Transport model

The transport modelling concerns evaluating dynamic viscosity μ , thermal conductivity κ and thermal diffusivity α (for internal energy and enthalpy equations). The current transport models are as follows:

`const` assumes a constant μ and Prandtl number $Pr = c_p\mu/\kappa$ which is simply specified by a two keywords, `mu` and `Pr`, respectively.

`sutherland` calculates μ as a function of temperature T from a Sutherland coefficient A_s and Sutherland temperature T_s , specified by keywords `As` and `Ts`; μ is calculated according to:

$$\mu = \frac{A_s \sqrt{T}}{1 + T_s/T}. \quad (7.1)$$

`polynomial` calculates μ and κ as a function of temperature T from a polynomial of any order N , *e.g.*:

$$\mu = \sum_{i=0}^{N-1} a_i T^i. \quad (7.2)$$

`logPolynomial` calculates $\ln(\mu)$ and $\ln(\kappa)$ as a function of $\ln(T)$ from a polynomial of any order N ; from which μ , κ are calculated by taking the exponential, *e.g.*:

$$\ln(\mu) = \sum_{i=0}^{N-1} a_i [\ln(T)]^i. \quad (7.3)$$

7.1.3 Thermodynamic models

The thermodynamic models are concerned with evaluating the specific heat c_p from which other properties are derived. The current `thermo` models are as follows:

`hConst` assumes a constant c_p and a heat of fusion H_f which is simply specified by a two values c_p H_f , given by keywords `Cp` and `Hf`.

`eConst` assumes a constant c_v and a heat of fusion H_f which is simply specified by a two values c_v H_f , given by keywords `Cv` and `Hf`.

`janaf` calculates c_p as a function of temperature T from a set of coefficients taken from JANAF tables of thermodynamics. The ordered list of coefficients is given in Table 7.1. The function is valid between a lower and upper limit in temperature T_l and T_h respectively. Two sets of coefficients are specified, the first set for temperatures above a common temperature T_c (and below T_h), the second for temperatures below T_c (and above T_l). The function relating c_p to temperature is:

$$c_p = R(((a_4T + a_3)T + a_2)T + a_1)T + a_0. \quad (7.4)$$

In addition, there are constants of integration, a_5 and a_6 , both at high and low temperature, used to evaluating h and s respectively.

`hPolynomial` calculates c_p as a function of temperature by a polynomial of any order N :

$$c_p = \sum_{i=0}^{N-1} a_i T^i. \quad (7.5)$$

The following case provides an example of its use: `$FOAM_TUTORIALS/lagrangian/-porousExplicitSourceReactingParcelFoam/filter`

Description	Entry	Keyword
Lower temperature limit	T_l (K)	<code>Tlow</code>
Upper temperature limit	T_h (K)	<code>Thigh</code>
Common temperature	T_c (K)	<code>Tcommon</code>
High temperature coefficients	$a_0 \dots a_4$	<code>highCpCoeffs</code> (a_0 a_1 a_2 a_3 $a_4 \dots$)
High temperature enthalpy offset	a_5	<code>a5\dots</code>
High temperature entropy offset	a_6	<code>a6</code>
Low temperature coefficients	$a_0 \dots a_4$	<code>lowCpCoeffs</code> (a_0 a_1 a_2 a_3 $a_4 \dots$)
Low temperature enthalpy offset	a_5	<code>a5\dots</code>
Low temperature entropy offset	a_6	<code>a6</code>

Table 7.1: JANAF thermodynamics coefficients.

7.1.4 Composition of each constituent

There is currently only one option for the `specie` model which specifies the composition of each constituent. That model is itself named `specie`, which is specified by the following entries.

- **nMoles**: number of moles of component. This entry is only used for combustion modelling based on regress variable with a homogeneous mixture of reactants; otherwise it is set to 1.
- **molWeight** in grams per mole of specie.

7.1.5 Equation of state

The following equations of state are available in the thermophysical modelling library.

rhoConst Constant density:

$$\rho = \text{constant}. \quad (7.6)$$

perfectGas Perfect gas:

$$\rho = \frac{1}{RT} p. \quad (7.7)$$

incompressiblePerfectGas Perfect gas for an incompressible fluid:

$$\rho = \frac{1}{RT} p_{\text{ref}}, \quad (7.8)$$

where p_{ref} is a reference pressure.

perfectFluid Perfect fluid:

$$\rho = \frac{1}{RT} p + \rho_0, \quad (7.9)$$

where ρ_0 is the density at $T = 0$.

linear Linear equation of state:

$$\rho = \psi p + \rho_0, \quad (7.10)$$

where ψ is compressibility (not necessarily $(RT)^{-1}$).

adiabaticPerfectFluid Adiabatic perfect fluid:

$$\rho = \rho_0 \left(\frac{p + B}{p_0 + B} \right)^{1/\gamma}, \quad (7.11)$$

where ρ_0, p_0 are reference density and pressure respectively, and B is a model constant.

Boussinesq Boussinesq approximation

$$\rho = \rho_0 [1 - \beta (T - T_0)] \quad (7.12)$$

where β is the coefficient of volumetric expansion and ρ_0 is the reference density at reference temperature T_0 .

`PengRobinsonGas` Peng Robinson equation of state:

$$\rho = \frac{1}{zRT} p, \quad (7.13)$$

where the complex function $z = z(p, T)$ can be referenced in the source code in `Peng-RobinsonGas.H`, in the `$FOAM_SRC/thermophysicalModels/specie/equationOfState/` directory.

`icoPolynomial` Incompressible, polynomial equation of state:

$$\rho = \sum_{i=0}^{N-1} a_i T^i, \quad (7.14)$$

where a_i are polynomial coefficients of any order N .

7.1.6 Selection of energy variable

The user must specify the form of energy to be used in the solution, either internal energy e and enthalpy h , and in forms that include the heat of formation Δh_f or not. This choice is specified through the `energy` keyword.

We refer to *absolute* energy where heat of formation is included, and *sensible* energy where it is not. For example absolute enthalpy h is related to sensible enthalpy h_s by

$$h = h_s + \sum_i c_i \Delta h_f^i \quad (7.15)$$

where c_i and h_f^i are the molar fraction and heat of formation, respectively, of specie i . In most cases, we use the sensible form of energy, for which it is easier to account for energy change due to reactions. Keyword entries for `energy` therefore include *e.g.* `sensibleEnthalpy`, `sensibleInternalEnergy` and `absoluteEnthalpy`.

7.1.7 Thermophysical property data

The basic thermophysical properties are specified for each species from input data. Data entries must contain the name of the specie as the keyword, *e.g.* `O2`, `H2O`, `mixture`, followed by sub-dictionaries of coefficients, including:

`specie` containing *i.e.* number of moles, `nMoles`, of the specie, and molecular weight, `molWeight` in units of g/mol;

`thermodynamics` containing coefficients for the chosen thermodynamic model (see below);

`transport` containing coefficients for the chosen transpsort model (see below).

The following is an example entry for a specie named `fuel` modelled using `sutherland` transport and `janaf` thermodynamics:

```
fuel
{
    specie
```

```

{
    nMoles      1;
    molWeight   16.0428;
}
thermodynamics
{
    Tlow        200;
    Thigh       6000;
    Tcommon     1000;
    highCpCoeffs (1.63543 0.0100844 -3.36924e-06 5.34973e-10
                   -3.15528e-14 -10005.6 9.9937);
    lowCpCoeffs (5.14988 -0.013671 4.91801e-05 -4.84744e-08
                   1.66694e-11 -10246.6 -4.64132);
}
transport
{
    As          1.67212e-06;
    Ts          170.672;
}
}

```

The following is an example entry for a specie named `air` modelled using `const` transport and `hConst` thermodynamics:

```

air
{
    specie
    {
        nMoles      1;
        molWeight   28.96;
    }
    thermodynamics
    {
        Cp          1004.5;
        Hf          2.544e+06;
    }
    transport
    {
        mu         1.8e-05;
        Pr         0.7;
    }
}

```

7.2 Turbulence models

The `turbulenceProperties` dictionary is read by any solver that includes turbulence modelling. Within that file is the `simulationType` keyword that controls the type of turbulence

modelling to be used, either:

- `laminar` uses no turbulence models;
- `RAS` uses Reynolds-averaged simulation (RAS) modelling;
- `LES` uses large-eddy simulation (LES) modelling.

7.2.1 Reynolds-averaged simulation (RAS) modelling

If RAS is selected, the choice of RAS modelling is specified in a RAS sub-dictionary which requires the following entries.

- `RASModel`: name of RAS turbulence model.
- `turbulence`: switch to turn the solving of turbulence modelling on/off.
- `printCoeffs`: switch to print model coeffs to terminal at simulation start up.
- `<RASModel>Coeffs`: optional dictionary of coefficients for the respective `RASModel`, to override the default coefficients.

Turbulence models can be listed by running a solver with the `-listTurbulenceModels` option, *e.g.*

```
simpleFoam -listTurbulenceModels
```

With `simpleFoam`, the incompressible models are listed. The compressible models are listed for a compressible solver, *e.g.* `rhoSimpleFoam`.

The RAS models used in the tutorials can be listed using `foamSearch` with the following command. The lists of available models are given in the following sections.

```
foamSearch $FOAM_TUTORIALS RAS.RASModel turbulenceProperties
```

Users can locate tutorials using a particular model, *e.g.* `buoyantKEpsilon`, using `find/grep`.

```
find $FOAM_TUTORIALS -name turbulenceProperties | \
xargs grep -l buoyantKEpsilon
```

7.2.1.1 Incompressible RAS turbulence models

For incompressible flows, the `RASModel` can be chosen from the list below.

`LRR` Launder, Reece and Rodi Reynolds-stress turbulence model for incompressible and compressible flows.

`LamBremhorstKE` Lam and Bremhorst low-Reynolds number k-epsilon turbulence model for incompressible flows

`LaunderSharmaKE` Lauder and Sharma low-Reynolds k-epsilon turbulence model for incompressible and compressible and combusting flows including rapid distortion theory (RDT) based compression term.

LienCubicKE Lien cubic non-linear low-Reynolds k-epsilon turbulence models for incompressible flows.

LienLeschziner Lien and Leschziner low-Reynolds number k-epsilon turbulence model for incompressible flows.

RNGkEpsilon Renormalization group k-epsilon turbulence model for incompressible and compressible flows.

SSG Speziale, Sarkar and Gatski Reynolds-stress turbulence model for incompressible and compressible flows.

ShihQuadraticKE Shih's quadratic algebraic Reynolds stress k-epsilon turbulence model for incompressible flows

SpalartAllmaras Spalart-Allmaras one-eqn mixing-length model for incompressible and compressible external flows.

kEpsilon Standard k-epsilon turbulence model for incompressible and compressible flows including rapid distortion theory (RDT) based compression term.

kOmega Standard high Reynolds-number k-omega turbulence model for incompressible and compressible flows.

kOmegaSSTLM Langtry-Menter 4-equation transitional SST model based on the k-omega-SST RAS model.

kOmegaSSTSAS Scale-adaptive URAS model based on the k-omega-SST RAS model.

kkLOmega Low Reynolds-number k-kl-omega turbulence model for incompressible flows.

qZeta Gibson and Dafa'Alla's q-zeta two-equation low-Re turbulence model for incompressible flows

realizableKE Realizable k-epsilon turbulence model for incompressible and compressible flows.

v2f Lien and Kalitzin's v2-f turbulence model for incompressible and compressible flows, with a limit imposed on the turbulent viscosity given by Davidson et al.

7.2.1.2 Compressible RAS turbulence models

For compressible flows, the **RASModel** can be chosen from the list below.

LRR Launder, Reece and Rodi Reynolds-stress turbulence model for incompressible and compressible flows.

LaunderSharmaKE Launder and Sharma low-Reynolds k-epsilon turbulence model for incompressible and compressible and combusting flows including rapid distortion theory (RDT) based compression term.

RNGkEpsilon Renormalization group k-epsilon turbulence model for incompressible and compressible flows.

SSG Speziale SSG Speziale, Sarkar and Gatski Reynolds-stress turbulence model for incompressible and compressible flows.

SpalartAllmaras Spalart-Allmaras one-eqn mixing-length model for incompressible and compressible external flows.

buoyantKEpsilon Additional buoyancy generation/dissipation term applied to the k and epsilon equations of the standard k-epsilon model.

kEpsilon Standard k-epsilon turbulence model for incompressible and compressible flows including rapid distortion theory (RDT) based compression term.

kOmega Standard high Reynolds-number k-omega turbulence model for incompressible and compressible flows.

kOmegaSSTLM Langtry-Menter 4-equation transitional SST model based on the k-omega-SST RAS model.

kOmegaSSTSAS Scale-adaptive URAS model based on the k-omega-SST RAS model.

realizableKE Realizable k-epsilon turbulence model for incompressible and compressible flows.

v2f Lien and Kalitzin's v2-f turbulence model for incompressible and compressible flows, with a limit imposed on the turbulent viscosity given by Davidson et al.

7.2.2 Large eddy simulation (LES) modelling

If LES is selected, the choice of LES modelling is specified in a LES sub-dictionary which requires the following entries.

- **LESModel:** name of LES turbulence model.
- **delta:** name of delta δ model.
- **<LESModel>Coeffs:** dictionary of coefficients for the respective **LESModel**, to override the default coefficients.
- **<delta>Coeffs:** dictionary of coefficients for the **delta** model.

The LES models used in the tutorials can be listed using **foamSearch** with the following command. The lists of available models are given in the following sections.

```
foamSearch $FOAM_TUTORIALS turbulenceProperties LES.LESModel
```

7.2.2.1 Incompressible LES turbulence models

For incompressible flows, the **LESModel** can be chosen from the list below.

DeardorffDiffStress Differential SGS Stress Equation Model for incompressible and compressible flows

Smagorinsky The Smagorinsky SGS model.

SpalartAllmarasDDES SpalartAllmaras DDES turbulence model for incompressible and compressible flows

SpalartAllmarasDES SpalartAllmarasDES DES turbulence model for incompressible and compressible flows

SpalartAllmarasIDDES SpalartAllmaras IDDES turbulence model for incompressible and compressible flows

WALE The Wall-adapting local eddy-viscosity (WALE) SGS model.

dynamicKEqn Dynamic one equation eddy-viscosity model

dynamicLagrangian Dynamic SGS model with Lagrangian averaging

kEqn One equation eddy-viscosity model

kOmegaSSTDES Implementation of the k-omega-SST-DES turbulence model for incompressible and compressible flows.

7.2.2.2 Compressible LES turbulence models

For compressible flows, the **LESModel** can be chosen from the list below.

DeardorffDiffStress Differential SGS Stress Equation Model for incompressible and compressible flows

Smagorinsky The Smagorinsky SGS model.

SpalartAllmarasDDES SpalartAllmaras DDES turbulence model for incompressible and compressible flows

SpalartAllmarasDES SpalartAllmarasDES DES turbulence model for incompressible and compressible flows

SpalartAllmarasIDDES SpalartAllmaras IDDES turbulence model for incompressible and compressible flows

WALE The Wall-adapting local eddy-viscosity (WALE) SGS model.

dynamicKEqn Dynamic one equation eddy-viscosity model

dynamicLagrangian Dynamic SGS model with Lagrangian averaging

kEqn One equation eddy-viscosity model

kOmegaSSTDES Implementation of the k-omega-SST-DES turbulence model for incompressible and compressible flows.

7.2.3 Model coefficients

The coefficients for the RAS turbulence models are given default values in their respective source code. If the user wishes to override these default values, then they can do so by adding a sub-dictionary entry to the `RAS` sub-dictionary file, whose keyword name is that of the model with `Coeffs` appended, *e.g.* `kEpsilonCoeffs` for the `kEpsilon` model. If the `printCoeffs` switch is on in the `RAS` sub-dictionary, an example of the relevant `...Coeffs` dictionary is printed to standard output when the model is created at the beginning of a run. The user can simply copy this into the `RAS` sub-dictionary file and edit the entries as required.

7.2.4 Wall functions

A range of wall function models is available in OpenFOAM that are applied as boundary conditions on individual patches. This enables different wall function models to be applied to different wall regions. The choice of wall function model is specified through the turbulent viscosity field ν_t in the `0/nut` file. **Note for OpenFOAM versions prior to v3.0.0:** wall functions for compressible RAS are specified through the μ_t field in the `0/mut` file, through ν_{sgs} in the `0/nuSgs` file for incompressible LES and μ_{sgs} in the `0/muSgs` file for compressible LES. For example, a `0/nut` file:

```

17 dimensions      [0 2 -1 0 0 0];
18 internalField   uniform 0;
20 boundaryField
21 {
22     movingWall
23     {
24         type          nutkWallFunction;
25         value         uniform 0;
26     }
27     fixedWalls
28     {
29         type          nutkWallFunction;
30         value         uniform 0;
31     }
32     frontAndBack
33     {
34         type          empty;
35     }
36 }
37
38 // ****
39

```

There are a number of wall function models available in the release, *e.g.* `nutWallFunction`, `nutRoughWallFunction`, `nutUSpaldingWallFunction`, `nutkWallFunction` and `nutkAtmWallFunction`. The user can consult the relevant directories for a full list of wall function models:

```
find $FOAM_SRC/TurbulenceModels -name wallFunctions
```

Within each wall function boundary condition the user can over-ride default settings for E , κ and C_μ through optional `E`, `kappa` and `Cmu` keyword entries.

Having selected the particular wall functions on various patches in the `nut/mut` file, the user should select `epsilonWallFunction` on corresponding patches in the `epsilon` field and `kqRwallFunction` on corresponding patches in the turbulent fields k , q and R .

7.3 Transport/rheology models

In OpenFOAM, solvers that do not include energy/heat, include a library of models for viscosity ν . The models typically relate viscosity to strain rate $\dot{\gamma}$ and are specified by the user in the *transportProperties* dictionary. The available models are listed in the following sections.

7.3.1 Newtonian model

The Newtonian model assumes ν is constant. Viscosity is specified by a `dimensionedScalar nu` in *transportProperties*, e.g.

```
transportModel Newtonian;
nu [ 0 2 -1 0 0 0 ] 1.5e-05;
```

Note the units for kinematic viscosity are L^2/T .

7.3.2 Bird-Carreau model

The Bird-Carreau model is:

$$\nu = \nu_\infty + (\nu_0 - \nu_\infty) [1 + (k\dot{\gamma})^a]^{(n-1)/a} \quad (7.16)$$

where the coefficient a has a default value of 2. An example specification of the model in *transportProperties* is:

```
transportModel BirdCarreau;
BirdCarreauCoeffs
{
    nu0 [ 0 2 -1 0 0 0 ] 1e-03;
    nuInf [ 0 2 -1 0 0 0 ] 1e-05;
    k [ 0 0 1 0 0 0 ] 1;
    n [ 0 0 0 0 0 0 ] 0.5;
}
```

7.3.3 Cross Power Law model

The Cross Power Law model is:

$$\nu = \nu_\infty + \frac{\nu_0 - \nu_\infty}{1 + (m\dot{\gamma})^n} \quad (7.17)$$

An example specification of the model in *transportProperties* is:

```
transportModel CrossPowerLaw;
CrossPowerLawCoeffs
{
    nu0 [ 0 2 -1 0 0 0 ] 1e-03;
```

```

nuInf      [ 0 2 -1 0 0 0 0 ] 1e-05;
m          [ 0 0 1 0 0 0 0 ] 1;
n          [ 0 0 0 0 0 0 0 ] 0.5;
}

```

7.3.4 Power Law model

The Power Law model provides a function for viscosity, limited by minimum and maximum values, ν_{\min} and ν_{\max} respectively. The function is:

$$\nu = k\dot{\gamma}^{n-1} \quad \nu_{\min} \leq \nu \leq \nu_{\max} \quad (7.18)$$

An example specification of the model in *transportProperties* is:

```

transportModel powerLaw;
powerLawCoeffs
{
    nuMax      [ 0 2 -1 0 0 0 0 ] 1e-03;
    nuMin      [ 0 2 -1 0 0 0 0 ] 1e-05;
    k          [ 0 2 -1 0 0 0 0 ] 1e-05;
    n          [ 0 0 0 0 0 0 0 ] 1;
}

```

7.3.5 Herschel-Bulkley model

The Herschel-Bulkley model combines the effects of Bingham plastic and power-law behavior in a fluid. For low strain rates, the material is modelled as a very viscous fluid with viscosity ν_0 . Beyond a threshold in strain-rate corresponding to threshold stress τ_0 , the viscosity is described by a power law. The model is:

$$\nu = \min (\nu_0, \tau_0/\dot{\gamma} + k\dot{\gamma}^{n-1}) \quad (7.19)$$

An example specification of the model in *transportProperties* is:

```

transportModel HerschelBulkley;
HerschelBulkleyCoeffs
{
    nu0       [ 0 2 -1 0 0 0 0 ] 1e-03;
    tau0      [ 0 2 -2 0 0 0 0 ] 1;
    k         [ 0 2 -1 0 0 0 0 ] 1e-05;
    n         [ 0 0 0 0 0 0 0 ] 1;
}

```

7.3.6 Casson model

The Casson model is a basic model used in blood rheology that specifies minimum and maximum viscosities, ν_{\min} and ν_{\max} respectively. Beyond a threshold in strain-rate corresponding to threshold stress τ_0 , the viscosity is described by a “square-root” relationship.

The model is:

$$\nu = \left(\sqrt{\tau_0/\dot{\gamma}} + \sqrt{m} \right)^2 \quad \nu_{\min} \leq \nu \leq \nu_{\max} \quad (7.20)$$

An example specification of model parameters for blood is:

```
transportModel Casson;
CassonCoeffs
{
    m          [ 0 2 -1 0 0 0 0 ] 3.934986e-6;
    tau0       [ 0 2 -2 0 0 0 0 ] 2.9032e-6;
    nuMax     [ 0 2 -1 0 0 0 0 ] 13.3333e-6;
    nuMin     [ 0 2 -1 0 0 0 0 ] 3.9047e-6;
}
```

7.3.7 General strain-rate function

A `strainRateFunction` model exists that allows a user to specify viscosity as a function of strain rate at run-time. It uses the same `Function1` functionality to specify the function of strain-rate, used by time varying properties in boundary conditions described in section 5.2.3.4. An example specification of the model in `transportProperties` is shown below using the `polynomial` function:

```
transportModel strainRateFunction;
strainRateFunctionCoeffs
{
    function polynomial ((0 0.1) (1 1.3));
}
```


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

-listTurbulenceModels keyword, U-206
/*...*/
 C++ syntax, U-82
//
 C++ syntax, U-82
 OpenFOAM file syntax, U-106
include
 C++ syntax, U-76, U-82
bounded keyword, U-122
<LESModel>Coeffs keyword, U-208
<RASModel>Coeffs keyword, U-206
<delta>Coeffs keyword, U-208
0.00000e+00 directory, U-106
1-dimensional mesh, U-138
1D mesh, U-138
2-dimensional mesh, U-138
2D mesh, U-138
O directory, U-106

add post-processing, U-185
addLayers keyword, U-159
addLayersControls keyword, U-159
adiabaticFlameT utility, U-104
adiabaticPerfectFluid model, U-203
adjointShapeOptimizationFoam solver, U-92
adjustableRunTime
 keyword entry, U-66, U-116
adjustTimeStep keyword, U-66, U-117
agglomerator keyword, U-129
Animations window panel, U-182
Annotation window panel, U-26
ansysToFoam utility, U-98
applications, U-73
Apply button, U-178, U-182
applyBoundaryLayer utility, U-97
arc
 keyword entry, U-150
arc keyword, U-149

As keyword, U-201
ascii
 keyword entry, U-117
attachMesh utility, U-99
Auto Accept button, U-182
autoPatch utility, U-99
autoRefineMesh utility, U-101
axes
 right-handed, U-148
 right-handed rectangular Cartesian, U-21
axi-symmetric cases, U-141, U-157
axi-symmetric mesh, U-138

background
 process, U-26, U-86
backward
 keyword entry, U-120
binary
 keyword entry, U-117
block
 expansion ratio, U-151
block keyword, U-148, U-149
blockMesh utility, U-98
blocking
 keyword entry, U-84
blockMesh utility, U-41, U-147
blockMesh executable
 vertex numbering, U-151
blockMeshDict
 dictionary, U-21, U-22, U-40, U-53, U-147,
 U-158
blocks keyword, U-22, U-35, U-150
boundaries, U-138
boundary, U-138
boundary
 dictionary, U-137, U-148
boundary keyword, U-148, U-153
boundary condition
 calculated, U-142

constantAlphaContactAngle, U-63
cyclic, U-141, U-154
cyclicAMI, U-141
directionMixed, U-143
empty, U-21, U-138, U-141
fixedGradient, U-142
fixedValue, U-142, U-146
inletOutlet, U-143
mixed, U-143
noSlip, U-23
patch, U-141
pressureInletOutletVelocity, U-144
processor, U-141
setup, U-23
symmetry, U-141
symmetryPlane, U-140, U-141
totalPressure, U-144
uniformFixedValue, U-146
wall, U-44
wall, U-63, U-140, U-141
wedge, U-138, U-141, U-157
zeroGradient, U-142
boundaryCloud post-processing, U-187
boundaryFoam solver, U-92
boundaryField keyword, U-23, U-111
bounded keyword, U-122
Boussinesq model, U-203
boxTurb utility, U-97
boxToCell keyword, U-64
breaking of a dam, U-60
BSpline
 keyword entry, U-150
buoyantBoussinesqPimpleFoam solver, U-95
buoyantBoussinesqSimpleFoam solver, U-95
buoyantPimpleFoam solver, U-95
buoyantSimpleFoam solver, U-95
buoyantKEpsilon model, U-208
burntProducts keyword, U-201
button
 Apply, U-178, U-182
 Auto Accept, U-182
 Camera Parallel Projection, U-182
 Choose Preset, U-181
 Delete, U-178
 Edit Color Legend Properties, U-31
 Edit Color Map, U-180
 Enable Line Series, U-39
 Lights, U-182
 Orientation Axes, U-26
 Refresh Times, U-28, U-179
Rescale, U-28
Reset, U-178
Set Ambient Color, U-181
Update GUI, U-179
Use Parallel Projection, U-26
C++ syntax
 `/*...*/`, U-82
 `//`, U-82
 `# include`, U-76, U-82
cacheAgglomeration keyword, U-129
calculated
 boundary condition, U-142
cAlpha keyword, U-67
Camera Parallel Projection button, U-182
case
 management, U-132
cases, U-105
castellatedMesh keyword, U-159
castellatedMeshControls
 dictionary, U-160–U-163
castellatedMeshControls keyword, U-159
cavitatingDyMFoam solver, U-93
cavitatingFoam solver, U-93
cavity flow, U-20
ccm26ToFoam utility, U-99
CEI_ARCH
 environment variable, U-197
CEI_HOME
 environment variable, U-197
cell
 expansion ratio, U-151
cellMax post-processing, U-187
cellMin post-processing, U-187
cellLimited
 keyword entry, U-121
cells
 dictionary, U-148
cfx4ToFoam utility, U-98
cfx4ToFoam utility, U-168
changeDictionary utility, U-97
Charts window panel, U-182
checkMesh utility, U-99
checkMesh utility, U-169
chemFoam solver, U-95
chemkinToFoam utility, U-104
Choose Preset button, U-181
chtMultiRegionFoam solver, U-95
chtMultiRegionSimpleFoam solver, U-95
class
 polyMesh, U-135, U-137

vector, U-109
class keyword, U-107
clockTime
 keyword entry, U-117
coalChemistryFoam solver, U-96
coldEngineFoam solver, U-95
collapseEdges utility, U-101
Color By menu, U-181
Color Legend window, U-33
Color Legend window panel, U-181
Color Scale window panel, U-181
Colors window panel, U-182
combinePatchFaces utility, U-101
comments, U-82
Common menu, U-31
commsType keyword, U-84
components post-processing, U-185
compressibleInterDyMFoam solver, U-93
compressibleInterFoam solver, U-93
compressibleMultiphaseInterFoam solver, U-93
constant directory, U-105, U-199
constant keyword, U-146
constantAlphaContactAngle
 boundary condition, U-63
Contour
 menu entry, U-31
control
 of time, U-115
controlDict
 dictionary, U-24, U-35, U-46, U-55, U-66,
 U-105, U-174
convergence, U-43
convertToMeters keyword, U-149
convertToMeters keyword, U-148, U-149
coordinate system, U-21
corrected
 keyword entry, U-124
Courant number, U-24
CourantNo post-processing, U-185
Cp keyword, U-202
cpuTime
 keyword entry, U-117
CrankNicolson
 keyword entry, U-120
createBaffles utility, U-99
createExternalCoupledPatchGeometry utility, U-97
createPatch utility, U-99
CrossPowerLaw
 keyword entry, U-64
csvFile keyword, U-146
Current Time Controls menu, U-28, U-179
Cv keyword, U-202
cyclic
 boundary condition, U-141, U-154
cyclicAMI
 boundary condition, U-141
dam
 breaking of a, U-60
datToFoam utility, U-98
DeardorffDiffStress model, U-208, U-209
decomposePar utility, U-103
decomposePar utility, U-86, U-87
decomposeParDict
 dictionary, U-86
decomposition
 of field, U-86
 of mesh, U-86
defaultFieldValues keyword, U-64
deformedGeom utility, U-100
Delete button, U-178
delta keyword, U-88, U-208
deltaT keyword, U-116
dependencies, U-76
dependency lists, U-76
diagonal
 keyword entry, U-126, U-128
DIC
 keyword entry, U-128
DICGaussSeidel
 keyword entry, U-128
dictionary
 PIMPLE, U-131
 PISO, U-25, U-131
 SIMPLE, U-131
 blockMeshDict, U-21, U-22, U-40, U-53,
 U-147, U-158
 boundary, U-137, U-148
 castellatedMeshControls, U-160–U-163
 cells, U-148
 controlDict, U-24, U-35, U-46, U-55, U-66,
 U-105, U-174
 decomposeParDict, U-86
 faces, U-137, U-148
 fvSchemes, U-67, U-105, U-118
 fvSolution, U-105, U-125
 fvSchemes, U-67
 mechanicalProperties, U-55
 neighbour, U-137
 owner, U-137
 points, U-137, U-148

thermalProperties, U-55
thermophysicalProperties, U-199
transportProperties, U-24, U-42, U-46, U-211
turbulenceProperties, U-45, U-65, U-205
DILU
 keyword entry, U-128
dimension
 checking in OpenFOAM, U-109
dimensional units, U-109
dimensions keyword, U-23, U-111
directionMixed
 boundary condition, U-143
directory
 $0.000000e+00$, U-106
 0, U-106
 Make, U-77
 constant, U-105, U-199
 polyMesh, U-105, U-137
 processorN, U-87
 run, U-19, U-105
 system, U-105
 tutorials, U-19
Display window panel, U-26, U-28, U-178, U-180
distance
 keyword entry, U-163
distributed keyword, U-88, U-90
div post-processing, U-185
div(phi,e) keyword, U-121
div(phi,k) keyword, U-121
div(phi,U) keyword, U-121
divSchemes keyword, U-118
dnsFoam solver, U-94
DPMFoam solver, U-96
DPMDFoam solver, U-96
driftFluxFoam solver, U-93
dsmcFoam solver, U-97
dsmcInitialise utility, U-98
dsmcFields post-processing, U-187
dynamicLagrangian model, U-209
dynamicKEqn model, U-209

edgeGrading keyword, U-151
edges keyword, U-148, U-149
Edit menu, U-182
 Edit Color Legend Properties button, U-31
 Edit Color Map button, U-180
egrMixture keyword, U-201
electrostaticFoam solver, U-97
empty
 boundary condition, U-21, U-138, U-141
Enable Line Series button, U-39

endTime keyword, U-24, U-116
energy keyword, U-200, U-204
engineCompRatio utility, U-101
engineFoam solver, U-95
engineSwirl utility, U-98
ENSIGHT7_INPUT
 environment variable, U-197
ENSIGHT7_READER
 environment variable, U-197
ensightFoamReader utility, U-197
enstrophy post-processing, U-185
environment variable
 CEI_ARCH, U-197
 CEI_HOME, U-197
 ENSIGHT7_INPUT, U-197
 ENSIGHT7_READER, U-197
 FOAM_FILEHANDLER, U-89
 FOAM_RUN, U-105
 WM_ARCH_OPTION, U-80
 WM_ARCH, U-80
 WM_COMPILER_TYPE, U-80
 WM_COMPILER, U-80
 WM_COMPILE_OPTION, U-80
 WM_DIR, U-80
 WM_LABEL_OPTION, U-80
 WM_LABEL_SIZE, U-80
 WM_LINK_LANGUAGE, U-80
 WM_MPLIB, U-80
 WM_OPTIONS, U-80
 WM_PRECISION_OPTION, U-80
 WM_PROJECT_DIR, U-80
 WM_PROJECT_INST_DIR, U-80
 WM_PROJECT_USER_DIR, U-80
 WM_PROJECT_VERSION, U-80
 WM_PROJECT, U-80
 WM_THIRD_PARTY_DIR, U-80
 wmake, U-79
equationOfState keyword, U-200
equilibriumFlameT utility, U-104
equilibriumCO utility, U-104
errorReduction keyword, U-167
Euler
 keyword entry, U-120
expansionRatio keyword, U-166
extrudeMesh utility, U-98
extrudeToRegionMesh utility, U-98

faceMax post-processing, U-187
faceMin post-processing, U-187
faceAreaPair
 keyword entry, U-129

faces
 dictionary, U-137, U-148

FDIC
 keyword entry, U-128

featureAngle keyword, U-166

features keyword, U-161

field
 U, U-25
 p, U-25
 decomposition, U-86

field keyword, U-189

fields
 mapping, U-174

fields keyword, U-189

fieldValues keyword, U-64

file
 Make/files, U-79
 files, U-77
 g, U-65
 options, U-77
 snappyHexMeshDict, U-159
 transportProperties, U-64
 handler, U-89
 parallel I/O, U-87

file format, U-106

fileModificationChecking keyword, U-84

fileModificationSkew keyword, U-84

files file, U-77

Filters menu, U-31

finalLayerThickness keyword, U-166

financialFoam solver, U-97

fireFoam solver, U-95

firstLayerThickness keyword, U-166

firstTime keyword, U-116

fixed
 keyword entry, U-117

fixedGradient
 boundary condition, U-142

fixedValue
 boundary condition, U-142, U-146

flattenMesh utility, U-100

floatTransfer keyword, U-84

flow
 free surface, U-60
 laminar, U-20
 turbulent, U-20

flowRateFaceZone post-processing, U-186

flowRatePatch post-processing, U-186

flowType post-processing, U-185

fluent3DMeshToFoam utility, U-99

fluentMeshToFoam utility, U-99

fluentMeshToFoam utility, U-168

fluidThermo model, U-200

OpenFOAM
 cases, U-105

FOAM_FILEHANDLER
 environment variable, U-89

FOAM_RUN
 environment variable, U-105

foamDataToFluent utility, U-102, U-195

foamDictionary utility, U-104

foamFormatConvert utility, U-104

foamHelp utility, U-104

foamListTimes utility, U-104

foamMeshToFluent utility, U-99

foamToEnsight utility, U-102, U-195

foamToEnsightParts utility, U-102, U-195

foamToGMV utility, U-102, U-196

foamToStarMesh utility, U-99

foamToSurface utility, U-99

foamToTecplot360 utility, U-102, U-196

foamToTetDualMesh utility, U-102, U-196

foamToVTK utility, U-102, U-196

foamUpgradeCyclics utility, U-98

foamChemistryFile keyword, U-201

foamCloneCase script, U-42, U-132

foamCorrectVrt script, U-172

foamDictionary utility, U-132

FoamFile keyword, U-107

foamFormatConvert utility, U-89

foamListTimes utility, U-132

foamSearch script, U-119

foamyHexMesh utility, U-98

foamyHexMeshBackgroundMesh utility, U-98

foamyHexMeshSurfaceSimplify utility, U-98

foamyQuadMesh utility, U-98

forceCoeffsCompressible post-processing, U-186

forceCoeffsIncompressible post-processing, U-186

forcesCompressible post-processing, U-186

forcesIncompressible post-processing, U-186

foreground
 process, U-26

format keyword, U-107

fuel keyword, U-201

functions keyword, U-118

fvSchemes
 dictionary, U-67

fvSchemes
 dictionary, U-67, U-105, U-118

fvSchemes

menu entry, U-56
fvSolution
 dictionary, U-105, U-125

g file, U-65
gambitToFoam utility, U-99
gambitToFoam utility, U-168
GAMG
 keyword entry, U-57, U-126, U-128
Gauss cubic
 keyword entry, U-121
GaussSeidel
 keyword entry, U-128
General window panel, U-182
general
 keyword entry, U-117
geometric-algebraic multi-grid, U-129
geometry keyword, U-156, U-159
gmshToFoam utility, U-99
gnuplot
 keyword entry, U-117
grad post-processing, U-185
gradient
 Gauss's theorem, U-56
 least square fit, U-56
 least squares method, U-56
gradSchemes keyword, U-118
graphFormat keyword, U-117

halfCosineRamp keyword, U-146
heheuPsiThermo
 keyword entry, U-201
Help menu, U-181
hePsiThermo
 keyword entry, U-200
heRhoThermo
 keyword entry, U-201
Hf keyword, U-202
hierarchical
 keyword entry, U-87, U-88
highCpCoeffs keyword, U-202
homogeneousMixture keyword, U-201

icoFoam solver, U-92
icoUncoupledKinematicParcelDyMFoam solver, U-96
icoUncoupledKinematicParcelFoam solver, U-96
icoFoam solver, U-20, U-24, U-28
icoPolynomial model, U-204
ideasUnvToFoam utility, U-99
ideasToFoam utility, U-168

incompressiblePerfectGas model, U-203
Information window panel, U-178
inhomogeneousMixture keyword, U-201
inletOutlet
 boundary condition, U-143
inletValue keyword, U-143
inotify
 keyword entry, U-84
inotifyMaster
 keyword entry, U-84
inside
 keyword entry, U-163
insideCells utility, U-100
interDyMFoam solver, U-93
interFoam solver, U-93
interMixingFoam solver, U-93
interPhaseChangeDyMFoam solver, U-94
interPhaseChangeFoam solver, U-94
interfaceHeight post-processing, U-187
internalCloud post-processing, U-188
internalField keyword, U-23, U-111
interpolationSchemes keyword, U-118
iterations
 maximum, U-127

jplot
 keyword entry, U-117

kEpsilon model, U-207, U-208
kEqn model, U-209
kOmega model, U-207, U-208
kOmegaSSTDES model, U-209
kOmegaSSTLM model, U-207, U-208
kOmegaSSTSAS model, U-207, U-208
keyword
 -listTurbulenceModels, U-206
As, U-201
Cp, U-202
Cv, U-202
FoamFile, U-107
Hf, U-202
LESModel, U-208, U-209
MULESCorr, U-66
N2, U-201
O2, U-201
Pr, U-201
RASModel, U-206, U-207
Tcommon, U-202
Thigh, U-202
Tlow, U-202
Ts, U-201

bounded, U-122
 addLayersControls, U-159
 addLayers, U-159
 adjustTimeStep, U-66, U-117
 agglomerator, U-129
 arc, U-149
 blocks, U-22, U-35, U-150
 block, U-148, U-149
 boundaryField, U-23, U-111
 boundary, U-148, U-153
 bounded, U-122
 boxToCell, U-64
 burntProducts, U-201
 cAlpha, U-67
 cacheAgglomeration, U-129
 castellatedMeshControls, U-159
 castellatedMesh, U-159
 class, U-107
 commsType, U-84
 constant, U-146
 convertToMeters, U-148, U-149
 convertToMeters, U-149
 csvFile, U-146
 defaultFieldValues, U-64
 deltaT, U-116
 delta, U-88, U-208
 dimensions, U-23, U-111
 distributed, U-88, U-90
 div(phi,U), U-121
 div(phi,e), U-121
 div(phi,k), U-121
 divSchemes, U-118
 edgeGrading, U-151
 edges, U-148, U-149
 egrMixture, U-201
 endTime, U-24, U-116
 energy, U-200, U-204
 equationOfState, U-200
 errorReduction, U-167
 expansionRatio, U-166
 featureAngle, U-166
 features, U-161
 fieldValues, U-64
 fields, U-189
 field, U-189
 fileModificationChecking, U-84
 fileModificationSkew, U-84
 finalLayerThickness, U-166
 firstLayerThickness, U-166
 firstTime, U-116
 floatTransfer, U-84
 foamChemistryFile, U-201
 format, U-107
 fuel, U-201
 functions, U-118
 geometry, U-156, U-159
 gradSchemes, U-118
 graphFormat, U-117
 halfCosineRamp, U-146
 highCpCoeffs, U-202
 homogeneousMixture, U-201
 inhomogeneousMixture, U-201
 inletValue, U-143
 internalField, U-23, U-111
 interpolationSchemes, U-118
 laplacianSchemes, U-118
 latestTime, U-42
 layers, U-164
 leastSquares, U-56
 levels, U-163
 libs, U-85, U-118
 linearRamp, U-146
 locationInMesh, U-161, U-162
 location, U-107
 lowCpCoeffs, U-202
 manualCoeffs, U-88
 maxAlphaCo, U-66
 maxBoundarySkewness, U-167
 maxConcave, U-167
 maxCo, U-66, U-117
 maxDeltaT, U-66
 maxFaceThicknessRatio, U-166
 maxGlobalCells, U-161
 maxInternalSkewness, U-167
 maxIter, U-127
 maxLocalCells, U-161
 maxNonOrtho, U-167
 maxPostSweeps, U-129
 maxPreSweeps, U-129
 maxThicknessToMedialRatio, U-166
 maxThreadFileBufferSize, U-89
 mergeLevels, U-129
 mergePatchPairs, U-148, U-149
 mergeTolerance, U-159
 meshQualityControls, U-159
 method, U-88
 minArea, U-167
 minDeterminant, U-167
 minFaceWeight, U-167
 minFlatness, U-167

minMedianAxisAngle, U-166
 minRefinementCells, U-161
 minTetQuality, U-167
 minThickness, U-166
 minTriangleTwist, U-167
 minTwist, U-167
 minVolRatio, U-167
 minVol, U-167
 mixture, U-201
 mode, U-163
 molWeight, U-204
 momentumPredictor, U-131
 multiComponentMixture, U-201
 mu, U-201
 nAlphaSubCycles, U-67
 nBufferCellsNoExtrude, U-166
 nCellsBetweenLevels, U-161
 nCorrectors, U-131
 nFaces, U-137
 nFinestSweeps, U-129
 nGrow, U-166
 nLayerIter, U-166
 nMoles, U-204
 nNonOrthogonalCorrectors, U-131
 nPostSweeps, U-129
 nPreSweeps, U-129
 nRelaxIter, U-164, U-166
 nRelaxedIter, U-166
 nSmoothNormals, U-166
 nSmoothPatch, U-164
 nSmoothScale, U-167
 nSmoothSurfaceNormals, U-166
 nSmoothThickness, U-166
 nSolveIter, U-164
 name, U-157
 neighbourPatch, U-154
 numberofSubdomains, U-88
 nu, U-211
 n, U-88
 object, U-107
 one, U-146
 order, U-88
 oxidant, U-201
 pRefCell, U-25, U-26, U-131
 pRefValue, U-25, U-26, U-131
 patchMap, U-174
 patches, U-149
 phases, U-64
 polynomial, U-146
 postSweepsLevelMultiplier, U-129
 preSweepsLevelMultiplier, U-129
 preconditioner, U-126, U-128
 pressure, U-55
 printCoeffs, U-45, U-206
 processorWeights, U-87
 probeLocations, U-191
 processorWeights, U-88
 project, U-156
 purgeWrite, U-117
 quadraticRamp, U-146
 quarterSineRamp, U-146
 quarterCosineRamp, U-146
 refinementRegions, U-161, U-163
 refinementSurfaces, U-161, U-162
 refinementRegions, U-163
 regions, U-64
 relTol, U-57, U-126, U-127
 relativeSizes, U-166
 relaxed, U-167
 resolveFeatureAngle, U-161, U-162
 roots, U-88, U-90
 runTimeModifiable, U-117
 scale, U-146
 scotchCoeffs, U-88
 sigma, U-64
 simpleGrading, U-151
 simulationType, U-45, U-65, U-205
 sine, U-146
 singleStepReactingMixture, U-201
 smoother, U-129
 snGradSchemes, U-118
 snapControls, U-159
 snap, U-159
 solvers, U-126
 solver, U-57, U-126
 specie, U-204
 spline, U-149
 square, U-146
 startFace, U-137
 startFrom, U-24, U-116
 startTime, U-24, U-116
 stopAt, U-116
 strategy, U-87, U-88
 tableFile, U-146
 table, U-146
 thermoType, U-199
 thermodynamics, U-204
 thickness, U-166
 timeFormat, U-117
 timePrecision, U-117

timeScheme, U-118
tolerance, U-57, U-126, U-127, U-164
topoSetSource, U-64
traction, U-55
transport, U-200, U-204
turbulence, U-206
type, U-200
uniformValue, U-146
valueFraction, U-143
value, U-23, U-142
version, U-107
vertices, U-22, U-148, U-149
veryInhomogeneousMixture, U-201
wallDist, U-118
writeCompression, U-117
writeControl, U-25, U-66, U-116
writeFormat, U-117
writeInterval, U-25, U-36, U-117
writePrecision, U-117
zero, U-146
<LESModel>Coeffs, U-208
<RASModel>Coeffs, U-206
<delta>Coeffs, U-208
keyword entry
 BSpline, U-150
 CrankNicolson, U-120
 CrossPowerLaw, U-64
 DICGaussSeidel, U-128
 DIC, U-128
 DILU, U-128
 Euler, U-120
 FDIC, U-128
 GAMG, U-57, U-126, U-128
 Gauss cubic, U-121
 GaussSeidel, U-128
 LES, U-45, U-206
 LUST, U-122
 MGridGen, U-129
 Newtonian, U-64
 PBiCGStab, U-126
 PBiCG, U-126
 PCG, U-126
 RAS, U-45, U-206
 adjustableRunTime, U-66, U-116
 arc, U-150
 ascii, U-117
 backward, U-120
 binary, U-117
 blocking, U-84
 cellLimited, U-121
 clockTime, U-117
 corrected, U-124
 cpuTime, U-117
 diagonal, U-126, U-128
 distance, U-163
 faceAreaPair, U-129
 fixed, U-117
 general, U-117
 gnuplot, U-117
 hePsiThermo, U-200
 heRhoThermo, U-201
 heheuPsiThermo, U-201
 hierarchical, U-87, U-88
 inotifyMaster, U-84
 inotify, U-84
 inside, U-163
 jplot, U-117
 laminar, U-45, U-206
 latestTime, U-116
 leastSquares, U-121
 limitedLinear, U-122
 limited, U-124
 linearUpwind, U-122
 linear, U-121
 line, U-150
 localEuler, U-120
 manual, U-87, U-88
 masterUncollated, U-87
 multivariateSelection, U-123
 nextWrite, U-116
 noWriteNow, U-116
 nonBlocking, U-84
 none, U-119, U-128
 orthogonal, U-124
 outside, U-163
 polyLine, U-150
 pureMixture, U-201
 raw, U-117
 reactingMixture, U-201
 runTime, U-36, U-116
 scheduled, U-84
 scientific, U-117
 scotch, U-87, U-88
 simple, U-87, U-88
 smoothSolver, U-126
 spline, U-150
 startTime, U-24, U-116
 steadyState, U-120
 symGaussSeidel, U-128
 timeStampMaster, U-84

timeStamp, U-84
timeStep, U-25, U-36, U-116
uncollated, U-87
uncorrected, U-124
upwind, U-122
writeNow, U-116
xmgr, U-117
kivaToFoam utility, U-99
kkLOmega model, U-207

LamBremhorstKE model, U-206
Lambda2 post-processing, U-185
laminar
 keyword entry, U-45, U-206
laplacianFoam solver, U-92
laplacianSchemes keyword, U-118
latestTime
 keyword entry, U-116
latestTime keyword, U-42
LaunderSharmaKE model, U-206, U-207
layers keyword, U-164
leastSquares
 keyword entry, U-121
leastSquares keyword, U-56
LES
 keyword entry, U-45, U-206
LESModel keyword, U-208, U-209
levels keyword, U-163
libraries, U-73
library
 PVFoamReader, U-177
 thermophysical, U-199
 vtkPVFoam, U-177
libs keyword, U-85, U-118
lid-driven cavity flow, U-20
LienCubicKE model, U-207
LienLeschziner model, U-207
Lights button, U-182
limited
 keyword entry, U-124
limitedLinear
 keyword entry, U-122
line
 keyword entry, U-150
Line Style menu, U-39
linear model, U-203
linear
 keyword entry, U-121
linearRamp keyword, U-146
linearUpwind
 keyword entry, U-122

localEuler
 keyword entry, U-120
location keyword, U-107
locationInMesh keyword, U-161, U-162
lowCpCoeffs keyword, U-202
LRR model, U-206, U-207
LUST
 keyword entry, U-122

MachNo post-processing, U-185
mag post-processing, U-185
magSqr post-processing, U-185
magneticFoam solver, U-97
Make directory, U-77
make script, U-75
Make/files file, U-79
manual
 keyword entry, U-87, U-88
manualCoeffs keyword, U-88
mapFields utility, U-98
mapFieldsPar utility, U-98
mapFields utility, U-35, U-42, U-46, U-59, U-174
mapping
 fields, U-174
Marker Style menu, U-39
masterUncollated
 keyword entry, U-87
maxAlphaCo keyword, U-66
maxBoundarySkewness keyword, U-167
maxCo keyword, U-66, U-117
maxConcave keyword, U-167
maxDeltaT keyword, U-66
maxFaceThicknessRatio keyword, U-166
maxGlobalCells keyword, U-161
maximum iterations, U-127
maxInternalSkewness keyword, U-167
maxIter keyword, U-127
maxLocalCells keyword, U-161
maxNonOrtho keyword, U-167
maxPostSweeps keyword, U-129
maxPreSweeps keyword, U-129
maxThicknessToMedialRatio keyword, U-166
maxThreadBufferSize keyword, U-89
mdEquilibrationFoam solver, U-97
mdFoam solver, U-97
mdlInitialise utility, U-98
mechanicalProperties
 dictionary, U-55
menu
 Color By, U-181
 Common, U-31

Current Time Controls, U-28, U-179
Edit, U-182
Filters, U-31
Help, U-181
Line Style, U-39
Marker Style, U-39
Recent, U-31
VCR Controls, U-28, U-179
View, U-178, U-181
menu entry
 Contour, U-31
 Plot Over Line, U-38
 Save Animation, U-183
 Save Screenshot, U-183
 Settings, U-182
 Slice, U-31
 Solid Color, U-181
 Toolbars, U-181
 View Settings, U-26, U-181
 Wireframe, U-181
fvSchemes, U-56
mergeMeshes utility, U-100
mergeOrSplitBaffles utility, U-100
mergeLevels keyword, U-129
mergePatchPairs keyword, U-148, U-149
mergeTolerance keyword, U-159
mesh
 1-dimensional, U-138
 1D, U-138
 2-dimensional, U-138
 2D, U-138
 axi-symmetric, U-138
 block structured, U-147
 decomposition, U-86
 description, U-135
 generation, U-147, U-158
 grading, U-147, U-151
 resolution, U-33
 specification, U-135
 split-hex, U-158
 Stereolithography (STL), U-158
 surface, U-158
 validity constraints, U-135
Mesh Parts window panel, U-26
meshQualityControls keyword, U-159
message passing interface
 openMPI, U-90
method keyword, U-88
MGridGen
 keyword entry, U-129
mhdFoam solver, U-97
minMaxComponents post-processing, U-187
minMaxMagnitude post-processing, U-187
minArea keyword, U-167
minDeterminant keyword, U-167
minFaceWeight keyword, U-167
minFlatness keyword, U-167
minMedianAxisAngle keyword, U-166
minRefinementCells keyword, U-161
minTetQuality keyword, U-167
minThickness keyword, U-166
minTriangleTwist keyword, U-167
minTwist keyword, U-167
minVol keyword, U-167
minVolRatio keyword, U-167
mirrorMesh utility, U-100
mixed
 boundary condition, U-143
mixture keyword, U-201
mixtureAdiabaticFlameT utility, U-104
mode keyword, U-163
model
 Boussinesq, U-203
 DeardorffDiffStress, U-208, U-209
 LRR, U-206, U-207
 LamBremhorstKE, U-206
 LaunderSharmaKE, U-206, U-207
 LienCubicKE, U-207
 LienLeschziner, U-207
 PengRobinsonGas, U-204
 RNGkEpsilon, U-207
 SSG, U-207, U-208
 ShihQuadraticKE, U-207
 Smagorinsky, U-208, U-209
 SpalartAllmarasDDES, U-209
 SpalartAllmarasDES, U-209
 SpalartAllmarasIDDES, U-209
 SpalartAllmaras, U-207, U-208
 WALE, U-209
 adiabaticPerfectFluid, U-203
 buoyantKEpsilon, U-208
 dynamicKEqn, U-209
 dynamicLagrangian, U-209
 fluidThermo, U-200
 icoPolynomial, U-204
 incompressiblePerfectGas, U-203
 kEpsilon, U-207, U-208
 kEqn, U-209
 kOmegaSSTDES, U-209
 kOmegaSSTLM, U-207, U-208

kOmegaSSTSAS, U-207, U-208
 kOmega, U-207, U-208
 kkLOmega, U-207
 linear, U-203
 multiphaseMixtureThermo, U-200
 perfectFluid, U-203
 perfectGas, U-203
 psiReactionThermo, U-200
 psiThermo, U-200
 psiuReactionThermo, U-200
 qZeta, U-207
 realizableKE, U-207, U-208
 rhoConst, U-203
 rhoReactionThermo, U-200
 rhoThermo, U-200
 v2f, U-207, U-208
 modifyMesh utility, U-101
 molWeight keyword, U-204
 momentumPredictor keyword, U-131
 moveDynamicMesh utility, U-100
 moveEngineMesh utility, U-100
 moveMesh utility, U-100
 MPI
 openMPI, U-90
 MPPICDyMFoam solver, U-96
 MPPICFoam solver, U-96
 mshToFoam utility, U-99
 mu keyword, U-201
 MULESCorr keyword, U-66
 multiComponentMixture keyword, U-201
 multigrid
 geometric-algebraic, U-129
 multiphaseEulerFoam solver, U-94
 multiphaseInterDyMFoam solver, U-94
 multiphaseInterFoam solver, U-94
 multiphaseMixtureThermo model, U-200
 multivariateSelection
 keyword entry, U-123

 n keyword, U-88
 N2 keyword, U-201
 nAlphaSubCycles keyword, U-67
 name keyword, U-157
 nBufferCellsNoExtrude keyword, U-166
 nCellsBetweenLevels keyword, U-161
 nCorrectors keyword, U-131
 neighbour
 dictionary, U-137
 neighbourPatch keyword, U-154
 netgenNeutralToFoam utility, U-99
 Newtonian
 keyword entry, U-64
 nextWrite
 keyword entry, U-116
 nFaces keyword, U-137
 nFinestSweeps keyword, U-129
 nGrow keyword, U-166
 nLayerIter keyword, U-166
 nMoles keyword, U-204
 nNonOrthogonalCorrectors keyword, U-131
 noise utility, U-101
 nonNewtonianIcoFoam solver, U-92
 nonBlocking
 keyword entry, U-84
 none
 keyword entry, U-119, U-128
 noSlip
 boundary condition, U-23
 noWriteNow
 keyword entry, U-116
 nPostSweeps keyword, U-129
 nPreSweeps keyword, U-129
 nRelaxedIter keyword, U-166
 nRelaxIter keyword, U-164, U-166
 nSmoothNormals keyword, U-166
 nSmoothPatch keyword, U-164
 nSmoothScale keyword, U-167
 nSmoothSurfaceNormals keyword, U-166
 nSmoothThickness keyword, U-166
 nSolveIter keyword, U-164
 nu keyword, U-211
 numberOfWorkstations keyword, U-88

 O2 keyword, U-201
 objToVTK utility, U-100
 object keyword, U-107
 one keyword, U-146
 Opacity text box, U-181
 OpenFOAM
 applications, U-73
 file format, U-106
 libraries, U-73
 OpenFOAM file syntax
 //, U-106
 openMPI
 message passing interface, U-90
 MPI, U-90
 Options window, U-182
 options file, U-77
 order keyword, U-88
 orientFaceZone utility, U-100
 Orientation Axes button, U-26

orthogonal dictionary, U-137, U-148
keyword entry, U-124
outside keyword entry, U-163
owner dictionary, U-137
oxidant keyword, U-201

p field, U-25
paraFoam, U-177
paraFoam, U-26
parallel running, U-86
parallel I/O, U-87 file handler, U-89
threading support, U-89
Paramters window panel, U-179
ParaView, U-26
particleTracks utility, U-101
patch boundary condition, U-141
patchSummary utility, U-104
patches keyword, U-149
patchMap keyword, U-174
PBiCG keyword entry, U-126
PBiCGStab keyword entry, U-126
PCG keyword entry, U-126
pdfPlot utility, U-101
PDRFoam solver, U-95
PDRMesh utility, U-101
PecletNo post-processing, U-185
PengRobinsonGas model, U-204
perfectFluid model, U-203
perfectGas model, U-203
phases keyword, U-64
PIMPLE dictionary, U-131
pimpleDyMFoam solver, U-92
pimpleFoam solver, U-92
Pipeline Browser window, U-26, U-178
PISO dictionary, U-25, U-131
pisoFoam solver, U-92
pisoFoam solver, U-20
Plot Over Line menu entry, U-38
plot3dToFoam utility, U-99
points dictionary, U-137, U-148
polyDualMesh utility, U-100
polyLine keyword entry, U-150
polyMesh directory, U-105, U-137
polyMesh class, U-135, U-137
polynomial keyword, U-146
porousSimpleFoam solver, U-92
post-processing, U-177
CourantNo, U-185
Lambda2, U-185
MachNo, U-185
PecletNo, U-185
Q, U-185
R, U-185
XiReactionRate, U-185
add, U-185
boundaryCloud, U-187
cellMax, U-187
cellMin, U-187
components, U-185
div, U-185
dsmcFields, U-187
enstrophy, U-185
faceMax, U-187
faceMin, U-187
flowRateFaceZone, U-186
flowRatePatch, U-186
flowType, U-185
forceCoeffsCompressible, U-186
forceCoeffsIncompressible, U-186
forcesCompressible, U-186
forcesIncompressible, U-186
grad, U-185
interfaceHeight, U-187
internalCloud, U-188
magSqr, U-185
mag, U-185
minMaxComponents, U-187
minMaxMagnitude, U-187
pressureDifferencePatch, U-187
pressureDifferenceSurface, U-187
probes, U-188, U-191
randomise, U-185
residuals, U-187, U-195
scalarTransport, U-188
singleGraph, U-186, U-192
staticPressure, U-187
streamFunction, U-185
streamlines, U-188, U-193

subtract, U-185
 surfaces, U-188, U-193
 totalPressureCompressible, U-187
 totalPressureIncompressible, U-187
 turbulenceFields, U-185
 volFlowRateSurface, U-186
 vorticity, U-185
 wallHeatFlux, U-186
 wallShearStress, U-186
 writeCellCentres, U-186
 writeCellVolumes, U-186
 writeObjects, U-186
 yPlus, U-186
 post-processing
 paraFoam, U-177
 postChannel utility, U-101
 postProcess utility, U-101
 postProcess utility, U-37, U-184
 postSweepsLevelMultiplier keyword, U-129
 potentialFoam solver, U-92
 potentialFreeSurfaceDyMFoam solver, U-94
 potentialFreeSurfaceFoam solver, U-94
 Pr keyword, U-201
 preconditioner keyword, U-126, U-128
 pRefCell keyword, U-25, U-26, U-131
 pRefValue keyword, U-25, U-26, U-131
 pressure keyword, U-55
 pressureDifferencePatch post-processing, U-187
 pressureDifferenceSurface post-processing, U-187
 pressureInletOutletVelocity
 boundary condition, U-144
 preSweepsLevelMultiplier keyword, U-129
 printCoeffs keyword, U-45, U-206
 processorWeights keyword, U-87
 probeLocations keyword, U-191
 probes post-processing, U-188, U-191
 process
 background, U-26, U-86
 foreground, U-26
 processor
 boundary condition, U-141
 processorN directory, U-87
 processorWeights keyword, U-88
 project keyword, U-156
 Properties window, U-179, U-180
 Properties window panel, U-28, U-178
 psiReactionThermo model, U-200
 psiThermo model, U-200
 psiuReactionThermo model, U-200
 pureMixture
 keyword entry, U-201
 purgeWrite keyword, U-117
 PVFoamReader
 library, U-177
 Q post-processing, U-185
 qZeta model, U-207
 quadraticRamp keyword, U-146
 quarterSineRamp keyword, U-146
 quarterCosineRamp keyword, U-146
 R post-processing, U-185
 randomise post-processing, U-185
 RAS
 keyword entry, U-45, U-206
 RASModel keyword, U-206, U-207
 raw
 keyword entry, U-117
 reactingFoam solver, U-95
 reactingMultiphaseEulerFoam solver, U-94
 reactingParcelFilmFoam solver, U-96
 reactingParcelFoam solver, U-96
 reactingTwoPhaseEulerFoam solver, U-94
 reactingMixture
 keyword entry, U-201
 realizableKE model, U-207, U-208
 Recent menu, U-31
 reconstructPar utility, U-103
 reconstructParMesh utility, U-103
 reconstructPar utility, U-91
 redistributePar utility, U-103
 refineHexMesh utility, U-101
 refineMesh utility, U-100
 refineWallLayer utility, U-101
 refinementLevel utility, U-101
 refinementRegions keyword, U-163
 refinementRegions keyword, U-161, U-163
 refinementSurfaces keyword, U-161, U-162
 Refresh Times button, U-28, U-179
 regions keyword, U-64
 relative tolerance, U-127
 relativeSizes keyword, U-166
 relaxed keyword, U-167
 relTol keyword, U-57, U-126, U-127
 removeFaces utility, U-101
 Render View window, U-182
 Render View window panel, U-181, U-182
 renumberMesh utility, U-100
 Rescale button, U-28
 Reset button, U-178
 residuals

monitoring, U-195
residuals post-processing, U-187, U-195
resolveFeatureAngle keyword, U-161, U-162
restart, U-42
Reynolds number, U-20, U-24
rhoCentralDyMFoam solver, U-92
rhoCentralFoam solver, U-92
rhoPimpleDyMFoam solver, U-93
rhoPimpleFoam solver, U-93
rhoPorousSimpleFoam solver, U-93
rhoReactingBuoyantFoam solver, U-95
rhoReactingFoam solver, U-95
rhoSimpleFoam solver, U-93
rhoConst model, U-203
rhoReactionThermo model, U-200
rhoThermo model, U-200
RNGkEpsilon model, U-207
roots keyword, U-88, U-90
rotateMesh utility, U-100
run
 parallel, U-86
run directory, U-19, U-105
runTime
 keyword entry, U-36, U-116
runTimeModifiable keyword, U-117

sammToFoam utility, U-99
Save Animation
 menu entry, U-183
Save Screenshot
 menu entry, U-183
scalarTransportFoam solver, U-92
scalarTransport post-processing, U-188
scale keyword, U-146
scalePoints utility, U-171
scheduled
 keyword entry, U-84
scientific
 keyword entry, U-117
scotch
 keyword entry, U-87, U-88
scotchCoeffs keyword, U-88
script
 foamCloneCase, U-42, U-132
 foamCorrectVrt, U-172
 foamSearch, U-119
 make, U-75
 wclean, U-79
 wmake, U-75
Seed window, U-183
selectCells utility, U-101

Set Ambient Color button, U-181
setFields utility, U-98
setSet utility, U-100
setFields utility, U-63, U-64
setsToZones utility, U-100
Settings
 menu entry, U-182
shallowWaterFoam solver, U-92
shape, U-151
ShihQuadraticKE model, U-207
SI units, U-110
sigma keyword, U-64
SIMPLE
 dictionary, U-131
simple
 keyword entry, U-87, U-88
simpleFoam solver, U-92
simpleReactingParcelFoam solver, U-96
simpleGrading keyword, U-151
simulationType keyword, U-45, U-65, U-205
sine keyword, U-146
singleCellMesh utility, U-100
singleGraph post-processing, U-186, U-192
singleStepReactingMixture keyword, U-201
Slice
 menu entry, U-31
Smagorinsky model, U-208, U-209
smapToFoam utility, U-102, U-196
smoother keyword, U-129
smoothSolver
 keyword entry, U-126
snap keyword, U-159
snapControls keyword, U-159
snappyHexMesh utility, U-98
snappyHexMesh utility
 background mesh, U-160
 cell removal, U-162
 cell splitting, U-160
 mesh layers, U-164
 meshing process, U-159
 snapping to surfaces, U-163
snappyHexMesh utility, U-158
snappyHexMeshDict file, U-159
snGradSchemes keyword, U-118
Solid Color
 menu entry, U-181
solidDisplacementFoam solver, U-97
solidEquilibriumDisplacementFoam solver, U-97
solidDisplacementFoam solver, U-55
solver

DPMDyMFoam, U-96
 DPMFoam, U-96
 MPPICDyMFoam, U-96
 MPPICFoam, U-96
 PDRFoam, U-95
 SRFPimpleFoam, U-92
 SRFSimpleFoam, U-92
 XiFoam, U-95
 adjointShapeOptimizationFoam, U-92
 boundaryFoam, U-92
 buoyantBoussinesqPimpleFoam, U-95
 buoyantBoussinesqSimpleFoam, U-95
 buoyantPimpleFoam, U-95
 buoyantSimpleFoam, U-95
 cavitatingDyMFoam, U-93
 cavitatingFoam, U-93
 chemFoam, U-95
 chtMultiRegionFoam, U-95
 chtMultiRegionSimpleFoam, U-95
 coalChemistryFoam, U-96
 coldEngineFoam, U-95
 compressibleInterDyMFoam, U-93
 compressibleInterFoam, U-93
 compressibleMultiphaselInterFoam, U-93
 dnsFoam, U-94
 driftFluxFoam, U-93
 dsmcFoam, U-97
 electrostaticFoam, U-97
 engineFoam, U-95
 financialFoam, U-97
 fireFoam, U-95
 icoFoam, U-20, U-24, U-28
 icoFoam, U-92
 icoUncoupledKinematicParcelDyMFoam, U-96
 icoUncoupledKinematicParcelFoam, U-96
 interDyMFoam, U-93
 interFoam, U-93
 interMixingFoam, U-93
 interPhaseChangeDyMFoam, U-94
 interPhaseChangeFoam, U-94
 laplacianFoam, U-92
 magneticFoam, U-97
 mdEquilibrationFoam, U-97
 mdFoam, U-97
 mhdFoam, U-97
 multiphaseEulerFoam, U-94
 multiphaselInterDyMFoam, U-94
 multiphaselInterFoam, U-94
 nonNewtonianIcoFoam, U-92
 pimpleDyMFoam, U-92
 pimpleFoam, U-92
 pisoFoam, U-20
 pisoFoam, U-92
 porousSimpleFoam, U-92
 potentialFoam, U-92
 potentialFreeSurfaceDyMFoam, U-94
 potentialFreeSurfaceFoam, U-94
 reactingFoam, U-95
 reactingMultiphaseEulerFoam, U-94
 reactingParcelFilmFoam, U-96
 reactingParcelFoam, U-96
 reactingTwoPhaseEulerFoam, U-94
 rhoCentralDyMFoam, U-92
 rhoCentralFoam, U-92
 rhoPimpleDyMFoam, U-93
 rhoPimpleFoam, U-93
 rhoPorousSimpleFoam, U-93
 rhoReactingBuoyantFoam, U-95
 rhoReactingFoam, U-95
 rhoSimpleFoam, U-93
 scalarTransportFoam, U-92
 shallowWaterFoam, U-92
 simpleFoam, U-92
 simpleReactingParcelFoam, U-96
 solidDisplacementFoam, U-55
 solidDisplacementFoam, U-97
 solidEquilibriumDisplacementFoam, U-97
 sonicDyMFoam, U-93
 sonicFoam, U-93
 sonicLiquidFoam, U-93
 sprayDyMFoam, U-96
 sprayEngineFoam, U-96
 sprayFoam, U-96
 thermoFoam, U-95
 twoLiquidMixingFoam, U-94
 twoPhaseEulerFoam, U-94
 uncoupledKinematicParcelDyMFoam, U-96
 uncoupledKinematicParcelFoam, U-96
 solver keyword, U-57, U-126
 solver relative tolerance, U-127
 solver tolerance, U-127
 solvers keyword, U-126
 sonicDyMFoam solver, U-93
 sonicFoam solver, U-93
 sonicLiquidFoam solver, U-93
 SpalartAllmaras model, U-207, U-208
 SpalartAllmarasDDES model, U-209
 SpalartAllmarasDES model, U-209
 SpalartAllmarasIDDES model, U-209
 specie keyword, U-204

spline
 keyword entry, U-150
spline keyword, U-149
splitCells utility, U-101
splitMesh utility, U-100
splitMeshRegions utility, U-100
sprayDyMFoam solver, U-96
sprayEngineFoam solver, U-96
sprayFoam solver, U-96
square keyword, U-146
SRFPimpleFoam solver, U-92
SRFSimpleFoam solver, U-92
SSG model, U-207, U-208
star3ToFoam utility, U-99
star4ToFoam utility, U-99
startFace keyword, U-137
startFrom keyword, U-24, U-116
starToFoam utility, U-168
startTime
 keyword entry, U-24, U-116
startTime keyword, U-24, U-116
staticPressure post-processing, U-187
steadyParticleTracks utility, U-101
steadyState
 keyword entry, U-120
Stereolithography (STL), U-158
stitchMesh utility, U-100
stopAt keyword, U-116
strategy keyword, U-87, U-88
streamFunction post-processing, U-185
streamlines post-processing, U-188, U-193
stress analysis of plate with hole, U-50
Style window panel, U-181
subsetMesh utility, U-100
subtract post-processing, U-185
surface mesh, U-158
surfaceAdd utility, U-102
surfaceAutoPatch utility, U-102
surfaceCheck utility, U-102
surfaceClean utility, U-102
surfaceCoarsen utility, U-102
surfaceConvert utility, U-102
surfaceFeatureConvert utility, U-102
surfaceFeatureExtract utility, U-102
surfaceFind utility, U-102
surfaceHookUp utility, U-102
surfaceInertia utility, U-102
surfaceLambdaMuSmooth utility, U-102
surfaceMeshConvert utility, U-102
surfaceMeshConvertTesting utility, U-102
surfaceMeshExport utility, U-103
surfaceMeshImport utility, U-103
surfaceMeshInfo utility, U-103
surfaceMeshTriangulate utility, U-103
surfaceOrient utility, U-103
surfacePointMerge utility, U-103
surfaceRedistributePar utility, U-103
surfaceRefineRedGreen utility, U-103
surfaceSplitByPatch utility, U-103
surfaceSplitNonManifolds utility, U-103
surfaceSubset utility, U-103
surfaceToPatch utility, U-103
surfaceTransformPoints utility, U-103
surfaceFeatureExtract utility, U-161
surfaces post-processing, U-188, U-193
symGaussSeidel
 keyword entry, U-128
symmetry
 boundary condition, U-141
symmetryPlane
 boundary condition, U-140, U-141
system directory, U-105
table keyword, U-146
tableFile keyword, U-146
Tcommon keyword, U-202
temporallInterpolate utility, U-101
tetgenToFoam utility, U-99
text box
 Opacity, U-181
thermalProperties
 dictionary, U-55
thermoFoam solver, U-95
thermodynamics keyword, U-204
thermophysical
 library, U-199
thermophysicalProperties
 dictionary, U-199
thermoType keyword, U-199
thickness keyword, U-166
Thigh keyword, U-202
time
 control, U-115
time step, U-24
timeFormat keyword, U-117
timePrecision keyword, U-117
timeScheme keyword, U-118
timeStamp
 keyword entry, U-84
timeStampMaster
 keyword entry, U-84

timeStep
 keyword entry, U-25, U-36, U-116

Tlow keyword, U-202

tolerance
 solver, U-127
 solver relative, U-127

tolerance keyword, U-57, U-126, U-127, U-164

Toolbars
 menu entry, U-181

topoSet utility, U-100

topoSetSource keyword, U-64

totalPressure
 boundary condition, U-144

totalPressureCompressible post-processing, U-187

totalPressureIncompressible post-processing, U-187

traction keyword, U-55

transformPoints utility, U-100

transport keyword, U-200, U-204

transportProperties
 dictionary, U-24, U-42, U-46, U-211

transportProperties file, U-64

Ts keyword, U-201

turbulence
 dissipation, U-44
 kinetic energy, U-44
 length scale, U-45

turbulence keyword, U-206

turbulence model
 RAS, U-44

turbulenceFields post-processing, U-185

turbulenceProperties
 dictionary, U-45, U-65, U-205

tutorials
 breaking of a dam, U-60
 lid-driven cavity flow, U-20
 stress analysis of plate with hole, U-50

tutorials directory, U-19

twoLiquidMixingFoam solver, U-94

twoPhaseEulerFoam solver, U-94

type keyword, U-200

U field, U-25

uncollated
 keyword entry, U-87

uncorrected
 keyword entry, U-124

uncoupledKinematicParcelDyMFoam solver, U-96

uncoupledKinematicParcelFoam solver, U-96

uniformFixedValue
 boundary condition, U-146

uniformValue keyword, U-146

units
 base, U-110
 of measurement, U-109
 SI, U-110
 Système International, U-110
 United States Customary System, U-110
 USCS, U-110

Update GUI button, U-179

upwind
 keyword entry, U-122

upwind differencing, U-67

USCS units, U-110

Use Parallel Projection button, U-26

utility
 PDRMesh, U-101
 adiabaticFlameT, U-104
 ansysToFoam, U-98
 applyBoundaryLayer, U-97
 attachMesh, U-99
 autoPatch, U-99
 autoRefineMesh, U-101
 blockMesh, U-41, U-147
 blockMesh, U-98
 boxTurb, U-97
 ccm26ToFoam, U-99
 cfx4ToFoam, U-168
 cfx4ToFoam, U-98
 changeDictionary, U-97
 checkMesh, U-169
 checkMesh, U-99
 chemkinToFoam, U-104
 collapseEdges, U-101
 combinePatchFaces, U-101
 createBaffles, U-99
 createExternalCoupledPatchGeometry, U-97
 createPatch, U-99
 datToFoam, U-98
 decomposePar, U-86, U-87
 decomposePar, U-103
 deformedGeom, U-100
 dsmcInitialise, U-98
 engineCompRatio, U-101
 engineSwirl, U-98
 ensightFoamReader, U-197
 equilibriumCO, U-104
 equilibriumFlameT, U-104
 extrudeMesh, U-98
 extrudeToRegionMesh, U-98
 flattenMesh, U-100

fluent3DMeshToFoam, U-99
fluentMeshToFoam, U-168
fluentMeshToFoam, U-99
foamDictionary, U-132
foamFormatConvert, U-89
foamListTimes, U-132
foamDataToFluent, U-102, U-195
foamDictionary, U-104
foamFormatConvert, U-104
foamHelp, U-104
foamListTimes, U-104
foamMeshToFluent, U-99
foamToEnsightParts, U-102, U-195
foamToEnsight, U-102, U-195
foamToGMV, U-102, U-196
foamToStarMesh, U-99
foamToSurface, U-99
foamToTecplot360, U-102, U-196
foamToTetDualMesh, U-102, U-196
foamToVTK, U-102, U-196
foamUpgradeCyclics, U-98
foamyHexMeshBackgroundMesh, U-98
foamyHexMeshSurfaceSimplify, U-98
foamyHexMesh, U-98
foamyQuadMesh, U-98
gambitToFoam, U-168
gambitToFoam, U-99
gmshToFoam, U-99
ideasToFoam, U-168
ideasUnvToFoam, U-99
insideCells, U-100
kivaToFoam, U-99
mapFields, U-35, U-42, U-46, U-59, U-174
mapFieldsPar, U-98
mapFields, U-98
mdlInitialise, U-98
mergeMeshes, U-100
mergeOrSplitBaffles, U-100
mirrorMesh, U-100
mixtureAdiabaticFlameT, U-104
modifyMesh, U-101
moveDynamicMesh, U-100
moveEngineMesh, U-100
moveMesh, U-100
mshToFoam, U-99
netgenNeutralToFoam, U-99
noise, U-101
objToVTK, U-100
orientFaceZone, U-100
particleTracks, U-101
patchSummary, U-104
pdfPlot, U-101
plot3dToFoam, U-99
polyDualMesh, U-100
postProcess, U-37, U-184
postChannel, U-101
postProcess, U-101
reconstructPar, U-91
reconstructParMesh, U-103
reconstructPar, U-103
redistributePar, U-103
refineHexMesh, U-101
refineMesh, U-100
refineWallLayer, U-101
refinementLevel, U-101
removeFaces, U-101
renumberMesh, U-100
rotateMesh, U-100
sammToFoam, U-99
scalePoints, U-171
selectCells, U-101
setFields, U-63, U-64
setFields, U-98
setSet, U-100
setsToZones, U-100
singleCellMesh, U-100
smashToFoam, U-102, U-196
snappyHexMesh, U-158
snappyHexMesh, U-98
splitCells, U-101
splitMeshRegions, U-100
splitMesh, U-100
star3ToFoam, U-99
star4ToFoam, U-99
starToFoam, U-168
steadyParticleTracks, U-101
stitchMesh, U-100
subsetMesh, U-100
surfaceFeatureExtract, U-161
surfaceAdd, U-102
surfaceAutoPatch, U-102
surfaceCheck, U-102
surfaceClean, U-102
surfaceCoarsen, U-102
surfaceConvert, U-102
surfaceFeatureConvert, U-102
surfaceFeatureExtract, U-102
surfaceFind, U-102
surfaceHookUp, U-102
surfaceInertia, U-102

surfaceLambdaMuSmooth, U-102
 surfaceMeshConvertTesting, U-102
 surfaceMeshConvert, U-102
 surfaceMeshExport, U-103
 surfaceMeshImport, U-103
 surfaceMeshInfo, U-103
 surfaceMeshTriangulate, U-103
 surfaceOrient, U-103
 surfacePointMerge, U-103
 surfaceRedistributePar, U-103
 surfaceRefineRedGreen, U-103
 surfaceSplitByPatch, U-103
 surfaceSplitNonManifolds, U-103
 surfaceSubset, U-103
 surfaceToPatch, U-103
 surfaceTransformPoints, U-103
 temporallInterpolate, U-101
 tetgenToFoam, U-99
 topoSet, U-100
 transformPoints, U-100
 viewFactorsGen, U-98
 vtkUnstructuredToFoam, U-99
 wallFunctionTable, U-98
 writeMeshObj, U-99
 zipUpMesh, U-100

v2f model, U-207, U-208
 value keyword, U-23, U-142
 valueFraction keyword, U-143
 VCR Controls menu, U-28, U-179
 vector class, U-109
 version keyword, U-107
 vertices keyword, U-22, U-148, U-149
 veryInhomogeneousMixture keyword, U-201
 View menu, U-178, U-181
 View Render window panel, U-26
 View Settings
 menu entry, U-26, U-181
 viewFactorsGen utility, U-98
 viscosity
 kinematic, U-24, U-46
 volFlowRateSurface post-processing, U-186
 vorticity post-processing, U-185
 vtkUnstructuredToFoam utility, U-99
 vtkPVFoam
 library, U-177

WALE model, U-209
 wall
 boundary condition, U-63, U-140, U-141
 wallFunctionTable utility, U-98

wallHeatFlux post-processing, U-186
 wallShearStress post-processing, U-186
 wallDist keyword, U-118
 wclean script, U-79
 wedge
 boundary condition, U-138, U-141, U-157
 window
 Color Legend, U-33
 Options, U-182
 Pipeline Browser, U-26, U-178
 Properties, U-179, U-180
 Render View, U-182
 Seed, U-183
 window panel
 Animations, U-182
 Annotation, U-26
 Charts, U-182
 Color Legend, U-181
 Color Scale, U-181
 Colors, U-182
 Display, U-26, U-28, U-178, U-180
 General, U-182
 Information, U-178
 Mesh Parts, U-26
 Paramters, U-179
 Properties, U-28, U-178
 Render View, U-181, U-182
 Style, U-181
 View Render, U-26

Wireframe
 menu entry, U-181

WM_ARCH
 environment variable, U-80

WM_ARCH_OPTION
 environment variable, U-80

WM_COMPILE_OPTION
 environment variable, U-80

WM_COMPILER
 environment variable, U-80

WM_COMPILER_TYPE
 environment variable, U-80

WM_DIR
 environment variable, U-80

WM_LABEL_OPTION
 environment variable, U-80

WM_LABEL_SIZE
 environment variable, U-80

WM_LINK_LANGUAGE
 environment variable, U-80

WM_MPLIB

environment variable, U-80
WM_OPTIONS
 environment variable, U-80
WM_PRECISION_OPTION
 environment variable, U-80
WM_PROJECT
 environment variable, U-80
WM_PROJECT_DIR
 environment variable, U-80
WM_PROJECT_INST_DIR
 environment variable, U-80
WM_PROJECT_USER_DIR
 environment variable, U-80
WM_PROJECT_VERSION
 environment variable, U-80
WM_THIRD_PARTY_DIR
 environment variable, U-80
wmake script, U-75
writeCellCentres post-processing, U-186
writeCellVolumes post-processing, U-186
writeMeshObj utility, U-99
writeObjects post-processing, U-186
writeCompression keyword, U-117
writeControl keyword, U-25, U-66, U-116
writeFormat keyword, U-117
writeInterval keyword, U-25, U-36, U-117
writeNow
 keyword entry, U-116
writePrecision keyword, U-117
XiFoam solver, U-95
XiReactionRate post-processing, U-185
xmgr
 keyword entry, U-117
yPlus post-processing, U-186
zero keyword, U-146
zeroGradient
 boundary condition, U-142
zipUpMesh utility, U-100