

Do not hesitate to send me links that may be of some interest or to signal me broken links -- thank you.
 Dear authors, I can remove, modify the reference or add your document - just send me an email.
 Contact: Julien.Reveillon@coria.fr

Version 2015-09-11

Title of the document	Name, firstName (& al)	Major Key	Second key	Year	Institution first author	Style
1 Modeling of compressible fluid problems with OpenFOAM using dynamic mesh technology	Aguerre, H. J. & al	dynamicMesh		2013	Research Center for Computational	paper
2 Liquid bridge simulations with OpenFOAM	Aguilar, G.G.	application	multiphase	2011	Universitat Politècnica de Catalunya	Master thesis
3 Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM	Beaudoin, M.	application	fluidStructure	2008	Hydro-Quebec	paper
4 OpenFOAM's basic solvers for linear systems of equations Solvers, preconditioners, smoothers	Behrens, T.	solver	misc	2009	Chalmers	report
5 OpenFOAM project: A modification of the movingConeTopoFVMesh library	Bjerklund, E.	dynamicMesh		2009	Chalmers	report
6 FoamedOver: A Dynamic Overset Grid Implementation in OpenFOAM	Boger, D. & al	dynamicMesh		2010	Pennsylvania State University	slides
7 Computational continuum mechanics for sediment transport in free-surface flow	Bohorquez, P.	application	multiphase	2008	Universidad de Malaga	slides
8 Dynamic-Mesh techniques for unsteady multiphase surface-ship hydrodynamics	Casadei, G. M.	dynamicMesh		2010	Pennsylvania State University	report
9 OpenFOAM for computational fluid dynamics	Chen, G. & al	OF basic		2014	Texas A&M University	paper
10 Skeletal Muscle Fascicle Arrangements Can Be Reconstructed Using a Laplacian Vector Field Simulation	Choi, H.F. & al	application	bioMechanics	2013	University of Virginia	paper
11 Towards Simulating the Atmospheric Boundary Layer and Wind Farm Flows	Churchfield, M.J. & al	application	windEnergy	2011	NREL	slides
12 An Extended Mixture Model for the Simultaneous Treatment of Short and Long Scale Interfaces	Damian, S.M.	PHD	multiphase	2013	Universidad Nacional del Litoral, A	PHD thesis
13 Description and utilization of InterFoam multiphase solver	Damian, S.M.	solver	multiphase	1900	CIMEC	report
14 Presentation_essai et validation du logiciel open-source OpenFOAM	Debain, B. & al	OF basic	firstSteps	2011	IPSA	report
15 CFD modelling of floating body response to regular waves	DeLaure, Y.	application	waves	2010	DCU - Dblin City University	slides
16 On Electromagnetically Induced Fluid Flows in Water	DeRues, M.	application		2013	Delft University of Technology	Master thesis
17 Contribution a l'étude de la formation des sprays [1]	Dos Santos, F.	PHD	multiphase	2012	Université de Bourgogne, France	PHD thesis
18 6-DOF VOF-solver without Damping in OpenFOAM	Ekedahl, E.	solver	floatingObject	2009	Chalmers	report
19 InterDyMFoam coupled with forces and mesh motion in 6-DoF	Ekedahl, E.	solver	floatingObject	2009	Chalmers	slides
20 Handling non-linear mesh deformation with OpenFOAM	Fabritius, B. & al	dynamicMesh		2008	TU Bergacademie Freiberg	slides
21 Simulation of turbulent channel flow over rippled bed with investigation of 4-way coupling for particles	Fonias E.	application	multiphase	2015	Chalmers	report
22 Evaluation of Turbulence Models for Prediction of Flow Separation at a Smooth Surface	Furbo, E. & al	modeling	turbulence	2009	Uppsala Universitet	report
23 Mesh motion alternatives	Gonzalez, A. O.	dynamicMesh		2009	Chalmers	report
24 Mesh motion alternatives	Gonzalez, A. O.	dynamicMesh		2009	Chalmers	slides
25 Description of porousSimpleFoam and adding the Brinkmann model to the porous models	Gooya, R.	solver	porous	2014	Chalmers	report
26 Sediment transport in open channel flows	Gruber, K.	application	hydro	2012	Johannes Kepler Universität Linz	master thesis
27 pyFoam -Productive use of OpenFOAM from the command line	Gschaidner, B. F.W.	tips		2013	ICE	slides
28 OpenFOAM hydrodynamics for yacht design	Gulinsky, O.	application	floatingObject	1900	Moscow Institute of Physics and Te	report
29 Porous Media in OpenFOAM	Haftsinsson H.E.	application	porous	2009	Chalmers	report
30 Simulating Wind Over Terrain: How to Build an OpenFOAM Case from GRASS GIS Digital Elevation Models	Hardin, E.	application	windEnergy	2013	unknown	paper
31 Free surface tutorial using InterFoam and rasInterFoam	Hemida, H.	tutorial	multiphase	2008	Chalmers	report
32 Two-Phase pipeline simulations with OpenFOAM	Herreras, N & al	application	multiphase	2013	Norwegian University of Science an	report
33 Two-Phase Flows & Waves	Higueira, P.	solver	multiphase	2013	University of Cantabria	slides
34 OF cheat sheet	Höpken, J.	OF basic		2015	sourceflux	poster
35 Implementation of Gasoline Properties in OpenFOAM Library	Huang C.	application		2010	Chalmers University of Technology	report
36 Forchheimer Porous-media Flow Models - Numerical Investigation and Comparison with Experimental Data	Jambhekar, V.A.	application	porousMedia	2011	University of Stuttgart, Germany	Master thesis
37 Block coupled calculations in OpenFOAM	Jareteg, Klas	numerics		2013	Chalmers	report
38 Error analysis and estimation for the finite volume method with applications to fluid flows	Jasak, H.	PHD	multiphase	1996	Imperial College, London	PHD thesis
39 Building Blocks and Library Organisation in OpenFOAM	Jasak, H.	bases		2012	University of Zagreb, Croatia	slides
40 Dynamic Mesh handling in OpenFOAM	Jasak, H.	dynamicMesh		2013	WIKKI	slides
41 Dynamic Mesh handling in OpenFOAM	Jasak, H.	dynamicMesh		2009	WIKKI	paper
42 Finite Volume Discretisation in OpenFOAM	Jasak, H.	numerics		2013	WIKKI	slides
43 OpenFOAM Tutorials: Basic Session	Jasak, H.	OF basic	firstSteps	2007	WIKKI	slides
44 Marine Hydrodynamics Solver in OpenFOAM	Jasak, H. & al	application	hydro	2008	WIKKI	slides
45 OpenFOAM Tutorials: Programming Session	Jasak, H. & al	OF basic	programming	2010	WIKKI	slides
46 Five Basic Classes in OpenFOAM	Jasak, H. & al	OF basic	firstSteps	2010	WIKKI	slides
47 Numerical simulation of three- dimensional flows in water storage tanks	Jaunatre, J.	application	hydro	2012	Lunds University	master thesis
48 Implementation of a turbulent inflow boundary condition for LES based on a vortex method	Jorgensen, N.G.	bases	boundaryConditio	2012	Chalmers	report
49 Numerical investigation of a BFR using OpenFOAM	Kær S.K.	application		2008	Aalborg University	report
50 Numerical Modelling of Diesel Spray Injection, Turbulence Interaction and Combustion	Kärholm, F. P.	PHD	multiphase	2008	Chalmers	PHD Thesis
51 Rhie-Chow interpolation in OpenFOAM	Kärholm, F. P.	OF basic	numerics	2006	Chalmers	report
52 Which strategy to move the mesh in the Computational Fluid Dynamic code OpenFOAM	Kassiotis, C.	dynamicMesh		2008	ENS Cachan	report
53 Partitioned solution to fluid-structure interaction problem in application to free-surface flows	Kassiotis, C. & al	solver	fluidStructure	2010	ENS Cachan	paper
54 Development and Verification of a Navier-Stokes Solver with Vorticity Confinement Using OpenFOAM	Kimbrell, A.B.	application		2012	University of Tennessee	Master thesis
55 A Coupled Volume-of-Fluid/ Level Set Method in OpenFOAM	Kisslin, K.	solver	multiphase	2010	University of Stuttgart, Germany	slides
56 A Coupled Pressure Based Solution Algorithm Based on the Volume-Of-Fluid Approach for Two or More Imm	Kissling, K. & al	programming	solver	2010	University of Stuttgart, Germany	paper
57 dieselDyMFoam: dynamic mesh refinement in dieselFoam	Koesters, A.	dynamicMesh	combustion	2010	Chalmers	slides
58 Dynamic mesh refinement in dieselFoam	Koesters, A.	dynamicMesh	combustion	2010	Chalmers	report
59 Salome to OpenFOAM mesh conversion tutorial	Kulakov, Yury	meshing		2010	University of Malta	report
60 CFD simulation of multicomponent gas flow through porous media	Kumar, C. M.	application	porousMedia	2010	Bergische Universität Wuppertal	Master thesis
61 Aerodynamic analysis of Sails with openfoam	Lderi, M.	application	fluidStructure	2007	Esteco, Trieste	slides
62 Application of OpenFOAM to prediction of hull resistance	Lee, S. B.	application		2013	Hyundai Heavy Ind.	slides
63 Introduction to OpenFOAM	Lindberg, O.	OF basic	firstSteps	2009	Technical University of Denmark	slides
64 Implementation of a myInterFoamDiabatic solver with OpenFOAM	Liu, Q.	solver	multiphase	2011	Chalmers	report
65 Numerical simulation of sailing boats: dynamics, FSI, and shape optimization	Lombardi, M.	application	floatingObject	2010	EPFL, Lausanne	report
66 Fluid-Structure Interaction (FSI) case study of a cantilever using OpenFOAM and DEAL_II with application to	Lorentzon J.	application	fluidStructure	2009	Lunds University	master thesis
67 Ship Resistance Simulations with OpenFOAM	Maki, K.	application	hydro	2011	Pennsylvania State University	slides
68 Modelling wind flow through canopy systems using OpenFOAM	Maldonado, J.M.	application	windEnergy	2012	University of Gävle	Master thesis
69 Geometrical Volume of Fluid (VoF) Method in OpenFOAM	Maric, T. & al	OF basic	volumeOffluid	2012	TU Darmstadt	slides
70 Wind resource in complex terrain with OpenFOAM	Martinez B.	application	windEnergy	2011	Technical University of Denmark	master thesis
71 Multi-dimensional simulation of intake and exhaust systems for internal combustion engines	Montorfano, A.	PHD	combustion	2013	Politecnico Di Milano	PHD thesis
72 A tutorial on how to use Dynamic Mesh solver IcoDyMFOAM	Moradnia, P.	dynamicMesh		2008	Chalmers	report
73 Application of the InterFoam VOF code to coastal wave/structure interaction	Morgan, G.C.J.	PHD	coastal	2013	University of Bath	PHD thesis
74 Application of OpenFOAM in Hydraulics Engineering	Nguyen, V.T.	application		2013	Seoul National University	slides
75 How to implement a new boundary condition	Nilsson, H.	bases	boundaryConditio	unknown	Chalmers	slides
76 Solving PDEs with OpenFOAM	Nilsson, H.	programming	pde	1900	Chalmers	slides
77 Turbomachinery	Nilsson, H.	application		2013	Chalmers	slides
78 Tutorial: Mesh motion class of the Vigor Wave Energy Converter	Olander, M.	dynamicMesh			Chalmers	report
79 Development of multiphase flow solvers based on OpenFOAM for volcanological research	Ongaro, T.O.	application	multiphase	2012	Istituto Nazionale di Geofisica e Vu	slides
80 Study of liquid metal MHD flows using OpenFOAM	Panchal A.	application	mhd	2011	IIT Bombay	report
81 Make my solver	Park, S.	code		2013	NEXTfoam	slides
82 The bubbleFoam solver	Passalacqua, A.	solver	multiphase	2013	Cocoon Project	report
83 Applications and case setup	Paterson, E.	OF basic	firstSteps	2009	Pennsylvania State University	slides
84 6DOF RANS Simulations of Floating and Submerged Bodies using OpenFOAM	Paterson, E. & al	application	floatingObject	2009	Pennsylvania State University	slides
85 Efficient computations of wave loads on offshore structures	Paulsen, B.T.	PHD	coastal	2013	DTU Mechanical Engineering	PHD thesis
86 Description and implementation of particle injection in OpenFOAM	Persson,A.	application	multiphase	2010	Chalmers	report
87 The ERCOFTAC centrifugal pump OpenFOAM case-study	Petit, Olivier & al	dynamicMesh		2009	Chalmers	paper
88 Development of Fully-Automatic Parallel Algorithms for Mesh Handling of ICE in OpenFOAM@-2.2.x	Piscaglia & al	dynamicMesh		2013	Politecnico di Milano	slides
89 Modelling multiphase flow through micro-CT images of the pore space	Raeini, A.Q.	PHD	multiphase	2013	Imperial College, London	PHD thesis
90 Coastal Structure Optimisation Using Advanced Numerical Methods	Richardson S. & al	application		2013	HR Wallingford	report
91 Comparative Study of the CFD codes Mistral and OpenFOAM	Rivola, V.	application	fluidStructure	2007	R-TEC	master thesis
92 Simple beginning 3D OpenFOAM Tutorial	Rodriguez, S.	application	hydrodynamics	2009	libremechanics	report
93 Rip current on a barred beach	Ruju, A. & al	application	coastal	2013	University of Cantabria	paper
94 Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions	Rusche, H.	PHD	multiphase	2002	Imperial College, London	PHD thesis
95 CFD Analysis of a Pelton Turbine in OpenFOAM	Rygg, J.R.	application		2013	NTNU-Trondheim	Master thesis
96 A wave motion class	Sam A.A.	programming	multiphase	2013	Chalmers	report
97 A wave motion class	Sam, Ali Al	dynamicMesh			Chalmers	report
98 CDF study of thick flatback airfoils using OpenFOAM	Sanz J.M.M.	application		2011	DTU Mechanical Engineering	Master thesis
99 Large Eddy Simulation of wind turbines: interaction with turbulent flow	Schito, P.	PHD	fluidStructure	2011	Politecnico Di Milano	PHD thesis
100 LES modelling of wind turbines	Schito, P.	application	fluidStructure	2012	Politecnico Di Milano	slides
101 Tutorial for Natural Convection Boundary Layer	Shiri A.	application		2008	Chalmers	report
102 Development and implementation of a polydispersed multiphase flow model in OpenFOAM	Silva, L.F.L.R.	modeling	multiphase	2011	Universidade Federal do Rio de Janeiro	report
103 Computation of viscous multiphase hydrodynamics and ship motions during wave-slap events and wave exci	Smith, K.R.	application	fluidStructure	2009	Pennsylvania State University	Master thesis
104 Modélisation des écoulements dans les gammages structurés : de l'échelle du pore à l'échelle de la colonne	Soulaïne, C.	PHD	porousMedia	2012	INP Toulouse	PHD thesis
105 Initiation à la CFD open-source avec le code OpenFOAM	Soulaïne, C.	OF basic	firstSteps	2013	IMFT	slides
106 Introduction to fluid mechanics simulation using the OpenFOAM@ technology Part I	Soulaïne, C.	application	porousMedia	2014	Stanford University	slides
107 Introduction to fluid mechanics simulation using the OpenFOAM@ technology Part II	Soulaïne, C.	application	porousMedia	2014	Stanford University	slides
108 Introduction to fluid mechanics simulation using the OpenFOAM@ technology Part III	Soulaïne, C.	application	porousMedia	2014	Stanford University	slides
109 Modélisation des écoulements dans les gammages structurés : de l'échelle du pore à l'échelle de la colonne	Soulaïne, C.	PHD	porousMedia	2012	IMFT	PHD thesis
110 A Coupled Pressure Based Solution Algorithm Based on the Volume-Of-Fluid Approach for Two or More Imm	Springer, J. & al	programming	solver	2010	University of Ulm, Germany	slides
111 Incorporation of the effects of accelerating -flow in the design of granular bed protections	Steenstra, R.S.	application		2014	Delft University of Technology	report
112 Numerical and Analytical Study of Steady State and Transient Heat Transfer in Liquid Filled Dead legs	Sundt, E.M.W.	application		2012	University of Stavanger	Master thesis
113 Implementation of solid body stress analysis in OpenFOAM	Tang, T.	application	fluidStructure	2012	Chalmers	report
114 Modelling of wind flow over complex terrain using OpenFoam	Tapia, X.P.	application	windEnergy	2009	University of Gauvle	master thesis
115 Modeling of Direct Contact Condensation With OpenFOAM	Thiele, R.	application	multiphase	2010	Royal Institute of Technology Stock	master thesis
116 Modeling liquid dominated two-phase flow in geothermal reservoirs in vicinity to, and inside wells	Thorvaldsson, L.	application	multiphase	2012	University of Iceland	report
117 A numerical analysis on flow in hydrothermal systems	Thorvaldsson, L.	application	multiphase	2013	University of Iceland	paper
118 Programming session: from the C++ basics to the compilation of user libraries	Trimarchi, D.	programming	misc	2011	University of Southampton	slides
119 OpenFOAM workshop	Trimarchi, D.	OF basic	firstSteps	2010	University of Southampton	slides
120 Fluid Structure Interactions in OpenFOAM	Trimarchi, D.	application	fluidStructure	2011	University of Southampton	slides
121 Fluid Structure Interactions of sail type structures	Trimarchi, D. & al	application	fluidStructure	2011	University of Southampton	slides
122 Numerical prediction of two fluid system with sharp interface	Ubbink, O.	PHD	multiphase	1997	Imperial College, London	PHD thesis
123 MovingWave class cuttingPlane class Velocity Profile postprocess	unknown	dynamicMesh			Chalmers	slides
124 CompressibleMixingPhaseChangeFoam_model equations	unknown	solver	multiphase	1900	os-cfd	slides
125 Coupling of VOF with LPT in OpenFOAM	Vallier, A.	programming	solver	2011	Chalmers	slides
126 Experimental and numerical study of wave induced porous flow in rubble mound breakwaters	Vanneste, D.	application	coastal	1900	University of Gent	PHD thesis
127 A tensorial approach to computational continuum mechanics using object-oriented techniques	Weller, H.G. & al	OF basic		1998	Imperial College, London	paper
128 Simulation and validation of compressible flow in nozzle geometries and validation of OpenFOAM for this app	Wutrich, B.	application		2007	Institute of Fluid Dynamics ETH Zurich	Master thesis
129 OpenFOAM Project: WhipLashMotion Library	Yang, J.	dynamicMesh		2009	Chalmers	report
pyFoam Happy foaming with Python	Gschaidner, B. F.W.	pospro		2009	ICE Stromungsforschung	slides