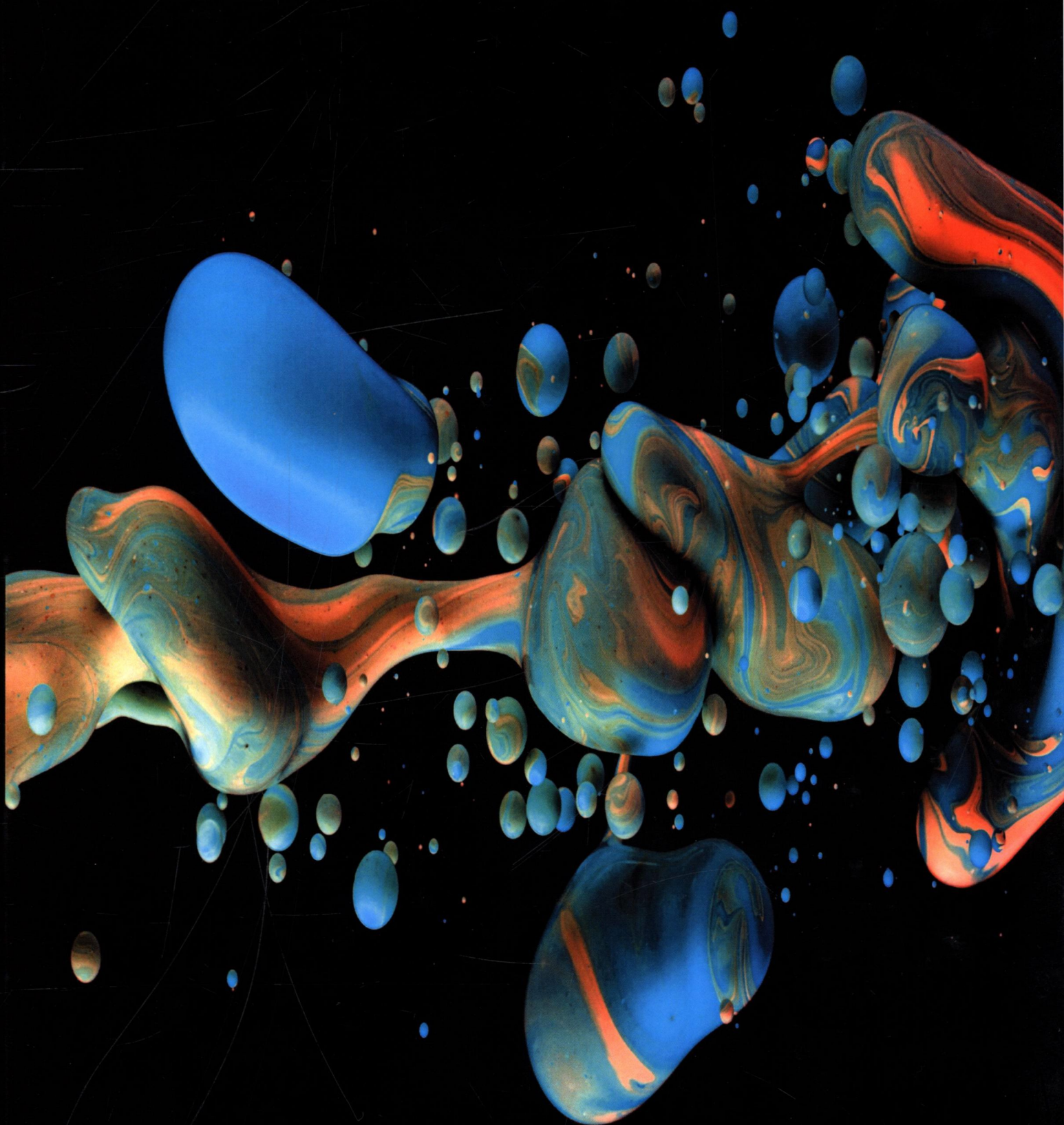


Theory And Problems Of Fluid Dynamics

Contributors: Marco Marcon , et al.



About the Book

Fluid dynamics is the study of the movement of fluids, including their interactions as two fluids come into contact with each other. The movement of liquids and gases is generally referred to as "flow," a concept that describes how fluids behave and how they interact with their surrounding environment — for example, water moving through a channel or pipe, or over a surface. Flow can be either steady or unsteady. Steady flows do not change over time. An example of steady flow would be water flowing through a pipe at a constant rate. On the other hand, a flood or water pouring from an old-fashioned hand pump are examples of unsteady flow. Flow can also be either laminar or turbulent. Laminar flows are smoother, while turbulent flows are more chaotic. One important factor in determining the state of a fluid's flow is its viscosity, or thickness, where higher viscosity increases the tendency of the flow to be laminar. In a moving fluid, the motion of a general fluid element can be thought of as being broken up into three parts: translation as a rigid body, rotation as a rigid body, and deformation. In fluid mechanics, there are generally three routes of work in the field, three ways to conduct experiments. The first category is theoretical, or analytical, fluid mechanics. Theoretical fluid mechanics includes theorizing, manipulating and solving equations with pen and paper. The Navier-Stokes equation governing incompressible fluid flow is an example of theoretical fluid mechanics. Secondly, many engineers and physicist work in the area of experimental fluid mechanics. Experimental fluid mechanics involves conducting actual physical experiments and studying the flow and the effect of various disturbances, shapes, and stimuli on the flow. Examples include waves generated by pools, air flow studies in actual wind tunnels, flow through physical pipes, etc. Lastly, a growing number of engineers, mathematicians, computer scientists, and physicists work in the area of computational fluid dynamics (CFD).

This book, *Theory and Problems of Fluid Dynamics*, is valuable tool for students and research workers in mathematics, physics, engineering and other sciences. Theory of fluid dynamics play an important role in many areas of science and technology. This work is a compilation of research articles by renowned contributors in their respective fields.



www.aurisreference.com



Marcon

Theory And Problems Of Fluid Dynamics



Theory and Problems of Fluid Dynamics

Contributors

Marco Marcon , et al.



www.aurisreference.com

Theory and Problems of Fluid Dynamics

Contributors: Marco Marcon , et al.

Published by Auris Reference Limited

www.aurisreference.com

United Kingdom

Copyright 2017

The information in this book has been obtained from highly regarded resources. The copyrights for individual articles remain with the authors, as indicated. All chapters are distributed under the terms of the Creative Commons Attribution License, which permit unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

Notice

Contributors, whose names have been given on the book cover, are not associated with the Publisher. The editors and the Publisher have attempted to trace the copyright holders of all material reproduced in this publication and apologise to copyright holders if permission has not been obtained. If any copyright holder has not been acknowledged, please write to us so we may rectify.

Reasonable efforts have been made to publish reliable data. The views articulated in the chapters are those of the individual contributors, and not necessarily those of the editors or the Publisher. Editors and/or the Publisher are not responsible for the accuracy of the information in the published chapters or consequences from their use. The Publisher accepts no responsibility for any damage or grievance to individual(s) or property arising out of the use of any material(s), instruction(s), methods or thoughts in the book.

Theory and Problems of Fluid Dynamics

ISBN: 978-1-78802-061-9

British Library Cataloguing in Publication Data

A CIP record for this book is available from the British Library

Printed in the United Kingdom

Theory and Problems of Fluid Dynamics

List of Contributors

Marco Marcon

Politecnico di Milano, Dipartimento di Elettronica e Informazione, via Ponzio 34/5, 20133 Milano Italy

Rui M. S. Pereira

University of Minho Portugal

Jitesh S. B. Gajjar

University of Manchester United Kingdom

Francesco Battista

CIRA SCpA Italian Aerospace Research Centre

Tommaso Misuri

ALTA SpA Italy

Mariano Andrenucci

ALTA SpA Italy

Balázs Pritz

Karlsruhe Institute of Technology, Department of Fluid Machinery Germany

Martin Gabi

Karlsruhe Institute of Technology, Department of Fluid Machinery Germany

Chaoqun Liu

University of Texas at Arlington, Arlington, TX, USA

Shuhyi Chen

University of Texas at Arlington, Arlington, TX, USA

Medzid Muhasilovic

IPSA-Institute, Sarajevo, Bosnia and Herzegovina

Kenan Imsirpasic

IPSA-Institute, Sarajevo, Bosnia and Herzegovina

Karel Ciahotny

VSCHT, Vysoka Skola Chemicko Technologicka, Praha, Czech Republic

Brano Sirok

Univerza v Ljubljani, Fakulteta za Strojnistvo, Ljubljana, Slovenija

Jorge Flores-Velázquez

Mexican Institute of Water Technology, Morelos, México

Abraham Rojano

Autonomous University of Chapingo, Texcoco, México, México

Adriana Rojas-Rishor

University of Costa Rica, San José, Costa Rica

Waldo Ojeda Bustamante

Mexican Institute of Water Technology, Morelos, México

Rodrigo Béttega

Federal University of Uberlândia, School of Chemical Engineering

Ronaldo Guimarães Corrêa

Federal University of São Carlos, Department of Chemical Engineering Brazil

José Teixeira Freire

Federal University of São Carlos, Department of Chemical Engineering Brazil

George Vahala

College of William & Mary

Bo Zhang

College of William & Mary

Jeffrey Yepez

Air Force Research Lab

Linda Vahala³

Old Dominion University

Min Soe⁴

Rogers State University USA

Preface

Fluid dynamics is the study of the movement of fluids, including their interactions as two fluids come into contact with each other. The movement of liquids and gases is generally referred to as “flow,” a concept that describes how fluids behave and how they interact with their surrounding environment — for example, water moving through a channel or pipe, or over a surface. Flow can be either steady or unsteady. Steady flows do not change over time. An example of steady flow would be water flowing through a pipe at a constant rate. On the other hand, a flood or water pouring from an old-fashioned hand pump are examples of unsteady flow. Flow can also be either laminar or turbulent. Laminar flows are smoother, while turbulent flows are more chaotic. One important factor in determining the state of a fluid’s flow is its viscosity, or thickness, where higher viscosity increases the tendency of the flow to be laminar. In a moving fluid, the motion of a general fluid element can be thought of as being broken up into three parts: translation as a rigid body, rotation as a rigid body, and deformation. In fluid mechanics, there are generally three routes of work in the field, three ways to conduct experiments. The first category is theoretical, or analytical, fluid mechanics. Theoretical fluid mechanics includes theorizing, manipulating and solving equations with pen and paper. The Navier-Stokes equation governing incompressible fluid flow is an example of theoretical fluid mechanics. Secondly, many engineers and physicist work in the area of experimental fluid mechanics. Experimental fluid mechanics involves conducting actual physical experiments and studying the flow and the effect of various disturbances, shapes, and stimuli on the flow. Examples include waves generated by pools, air flow studies in actual wind tunnels, flow through physical pipes, etc. Lastly, a growing number of engineers, mathematicians, computer scientists, and physicists work in the area of computational fluid dynamics (CFD).

This book, *Theory and Problems of Fluid Dynamics*, is valuable tool for students and research workers in mathematics, physics, engineering and other sciences. Theory of fluid dynamics play an important role in many areas of science and technology. This work is a compilation of research articles by renowned contributors in their respective fields.

Contents

	List of Contributors.....	vii
	Preface.....	ix
Chapter 1	Fluid Dynamics Without Fluids	1
	Marco Marcon	
Chapter 2	External Incompressible Flows.....	17
Chapter 3	Compressible Flow.....	27
Chapter 4	Non-Newtonian Fluids.....	45
Chapter 5	Solving Fluid Dynamics Problems With Matlab.....	57
	Rui M. S. Pereira and Jitesh S. B. Gajjar	
Chapter 6	A Magneto-Fluid-Dynamic Model and Computational Solving Methodologies for Aerospace Applications	71
	Francesco Battista, Tommaso Misuri and Mariano Andreucci	
Chapter 7	Stability Investigation of Combustion Chambers with LES	93
	Balázs Pritz and Martin Gabi	
Chapter 8	Self-Contradictions of Current Turbulence Theory and Liu’s New Turbulence Generation Theory	109
	Chaoqun Liu and Shuhyi Chen	
Chapter 9	Cfd-Based Investigation of Wind-Stroke Over Highway Bridge Section.....	123
	Medzid Muhasilovic, Kenan Imsirpasic, Karel Ciahotny and Brano Sirok	
Chapter 10	Computational Fluid Dynamics Achievements Applied to Optimal Crop Production in a Greenhouse	131
	Jorge Flores-Velázquez, Abraham Rojano, Adriana Rojas-Rishor and Waldo Ojeda Bustamante	
Chapter 11	Use of Fluid Dynamic Simulation to Improve the Design of Spouted Beds	149
	Rodrigo Béttega, Ronaldo Guimarães Corrêa and José Teixeira Freire	
Chapter 12	Unitary Qubit Lattice Gas Representation Of 2D And 3D Quantum Turbulence	167
	George Vahala, Bo Zhang, Jeffrey Yopez, Linda Vahala and Min Soe	
	Citation.....	193
	Index.....	195



1

FLUID DYNAMICS WITHOUT FLUIDS

Marco Marcon

Politecnico di Milano, Dipartimento di Elettronica e Informazione, via Ponzio 34/5, 20133
Milano Italy

INTRODUCTION

This chapter will discuss some interesting real applications where Fluid Dynamics equations found fruitful applications without dealing with “strictly speaking” fluids. In particular, thanks to the large set of analyses performed over different kinds of fluids in different operating and boundary conditions, a wide range of Computational Fluid Dynamics algorithms flourished tackling different aspects, from convergence rate, to stability according to the discretization, to multigrid and linearization problems. This robust and thorough background, both on theoretical and on practical aspects, made Computational Fluid Dynamics (CFD) appealing also to other sciences and applications where Fluid Dynamics equations, or similar equations very close to them, can be useful in describing complex phenomena not related to fluids. Some applications that will be discussed concern, e.g., Geometry of liquid snowflakes whose contour is growing steered by curvature, starting from a circle. Furthermore Image Restoration and Segmentation can also benefit from CFD since a set of evolutionary algorithms, based on level-set curvature flow equations, plays a fundamental role in steering active contours or snakes through the noise present in the image till the complete warping of the desired framed object. Also in this case advanced techniques like Ghost Fluids Method for two competing fluids dynamics can be used to separate different objects in images. Other interesting applications that will be described concern applicability of CFD to surface extraction from cloud of points. This is a common problem when complex clouds of points, representing 3D objects or scenes are obtained by laser scanners or multi-camera vision systems. These points represent unambiguous features from corners or sharp edges and the final 3D closed surface must fit on these points smoothly interpolating empty space between them. Also in this case CFD can provide useful tools to define the evolution of a 3D surface representing the border between two competing fluids, one representing the “inside” and the other the “outside” of the object itself. The two fluids evolution will stop when surface sticks on all the 3D points: the viscosity of the two fluids will control the smoothness of this surface that will wrap the cloud and turbulence is used to model injection into grooves or narrow holes. This chapter will also discuss another interesting application of CFD to robotic navigation in complex environments where we are looking for the best path, both in terms of length and distance from objects, through a set of obstacles, different terrains traversability or path slope. Also in this case an imaginary fluid with a predefined viscosity floods from the robot position through the whole environment, its front evolution speed, accordingly to CFD, will be slower in narrow passages and, once it reaches the target, it will define the easiest way.

SNOWFLAKES AND PHASE INTERFACES

In many physical problems outlines of a considered fluid have different speeds at different points, and in many

cases local speed is directly connected to the curvature. In the following we will show how a geometrical contour, represented by a 2D line or a 3D surface of the evolving shape changes from point to point accordingly to the local geometrical properties of the contour itself. The presented approach can be applied to a generic N-dimensional case and we can refer to the moving contour as a general hypersurface, anyway in the following, we will limit without loss of generalization our considerations to a 3D space where fluid boundary is represented by a closed surface.

In particular Sethian (1989) showed how algorithms based on direct parametrization of the surface evolution could present ambiguous and error-prone solutions due to the local error propagation while global approaches based on implicit representations can result in a much more robust solution. This kind of approaches usually relies on higher dimensional functions for which the considered surface is just represented by a level-set. The contour motion can then be described applying the proper Hamilton-Jacobi equation related to a hyperbolic conservation law to the implicit function and tracking the particular level-set representing the contour itself.

Implicit function formulation is much more robust to numerical techniques with respect to direct methods and can be implemented starting from an original closed and non-intersecting surface: two famous physical examples are the combustion model of a flame and the grow of a snowflake: Markstein (1951) assumed that thin flame fingers close to their ends are cooler than their inner part and move slower than hot nonconvex regions: the proposed evolution equation assumes that flame contour speed is inversely proportional to its curvature κ , (where κ is positive for convex regions and negative for nonconvex ones). This assumption come from the fact that in highly convex regions, flame particles collide with a high number of slower and cooler surrounding particles that slow down their motion. The symmetrical effect is present in spikes growing in snowflakes. In particular these two last examples, flames and snowflakes, represent two very good examples where simple equations can explain very well complex phenomena. For a comprehensive description of ice crystal growth we suggest Libbrecht (2005), while in the following we describe how solid-liquid interface dynamics are steered by curvature in liquid snowflakes (or Tyndall figures). We refer to some recent studies (Hennessy (2010), Hobbs (2010)) on cylindrical discs of liquid in superheated crystals of ice. In this case the snowflake is made of water inside ice and its geometrical shape evolution is determined by boundary condition between solid and liquid phase. In Fig. 1 there is a typical snapshot of a Liquid snowflake.

Liquid snowflakes surface evolution

The typical modelization of snowflakes surface growing is the Gibbs-Thomson equation where the interface temperature T_I between the solid and the liquid part is ruled by the equation:

$$T_I(\kappa) = T_M \left(1 - \kappa \frac{\gamma}{\rho L} \right) \quad (1)$$

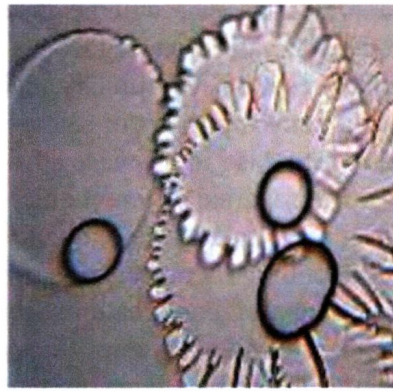


Figure 1: A typical Tyndall figure (Liquid snowflake)

Where T_M is the equilibrium temperature of a planar solid-liquid interface (273.15K for water), κ is the curvature, γ is the surface energy per unit area for the solid-liquid interface, ρ is the density of a specific phase but in this case we are assuming equal density both for ice and water and L is the latent heat of fusion. The steering interface force is then due to the pressure variation across the interface and depends from curvature:

$$p_s - p_l = \rho (s_l - s_s) (T_I - T_M) = \rho L \left(\frac{T_I(\kappa)}{T_M} - 1 \right) \quad (2)$$

Where the subscript l and s refer to liquid and solid state respectively and s indicates the entropy per unit mass, for further details refer to (Hills & Roberts (1993)).

A further improvement in the Gibbs-Thomson model considers surface energy anisotropy due to crystal ice structure (six fold symmetry), this will add a dependence of the surface energy γ from the relative orientation of the surface with respect to the underlying crystal. Accordingly to (Davis (2001)) a six fold symmetrical crystal will have a:

$$\gamma_6 = \gamma_0 \left[1 + \frac{\sigma_n}{n^2 - 1} \cos(n\theta) \right] \Big|_{n=6} \quad (3)$$

Where σ_n represents the anisotropy degree and the complete interface temperature will be, accordingly to (Pimpinelli & Villain (1999)) ;

$$T_l(\kappa) = T_M \left(1 - \kappa \frac{\gamma_6 + \frac{d^2\gamma_6}{d\theta^2}}{\rho L} \right) \quad (4)$$

In particular, accordingly to equations 3 and 4, whenever $|\sigma_n| < 1$ we are in a weak anisotropy condition, i.e. the surface energy γ is always positive, and, at equilibrium, the surface will resemble a smooth line, while, for anisotropy values greater than one, as for the snowflakes water example, the thermodynamic equilibrium implies a closed polygonal made of straight line segments (Dobrushin et al. (1992)).

Once the Temperature of the interface is determined the shape of the boundary between the liquid and solid part and its time evolution can be recovered from the energy conservation, in particular, in order for the ice-water interface to advance inside the solid part, two energy contributions must be provided: one for solid melting and one for interface stretching. The melting energy that must be provided per unit of time for a surface A that is moving inside the solid at a normal speed v_n is $l_{mel} = \rho L A v_n$ while the energy contribution per unit of time due to surface stretching is $l_{sur} = \gamma \frac{dA}{dt}$; which, accordingly to the differential geometry can be written as:

$$l_{sur} = \gamma \frac{dA}{dt} = A \gamma \kappa v_n \quad (5)$$

The surface advancement is steered by two opposite heat diffusive fluxes: one inside the liquid part and one inside the solid part; respectively: $-k_l \nabla T_l$ and $k_s \nabla T_s$ where k indicates the thermal conductivity of the considered phase.

The equilibrium of energy equations per area and time unit can be written as:

$$\frac{l_{mel}}{A} + \frac{l_{sur}}{A} = (-k_l \nabla T_l + k_s \nabla T_s) \cdot \mathbf{n} \quad (6)$$

$$(\rho L + \gamma \kappa) v_n = (-k_l \nabla T_l + k_s \nabla T_s) \cdot \mathbf{n} \quad (7)$$

Where $\mathbf{n}$ is surface normal oriented from the liquid part towards the solid one; this equation is called the Stefan condition. The numerical solution for the liquid-solid interface evolution, accordingly to the Stefan condition, requires some particular precaution due to accuracy that must be placed in surface tracking. Since the analyzed problem is far from thermodynamic equilibrium due to the superheated temperature of the solid, some typical approaches, e.g. the Enthalpy model (Shastri & Allen (1998)), are not accurate. In fact, Enthalpy approach, even if requires only the evaluation of a single parabolic PDE at each time step, assumes that the interface is within a mushy solid-liquid region and the evolving interface is just approximated. A possible extension of Enthalpy approach to account for superheated solid state could be phase-field models; anyway an accurate interface tracking requires a very small grid spacing resulting in a high computational cost (Biben et al. (2005)). One of the method that in recent years is getting growing attention is Level-set: in the following of this chapter we will show its versatility and accuracy in many CFD applications and in liquid snowflakes shape definition.

LEVEL-SET METHODS

For most of real CFD systems symbolic formulation that easily yield to symbolic mathematical solution is unfeasible, and the goal becomes to find the system modelization with the best trade-off between accuracy and computational complexity accounting for local, global and independent properties of the moving surface. In particular we will refer to “local” as properties associated to local surface features (e.g. curvature, local density or temperature), “global” as properties associated to overall surface shape or extension, while independent properties are not linked to the surface itself, like a flow below the surface. The Level-set method aims to track a thin propagating interface over time, it outperforms many other approaches handling topological complexities such as corners and cusps, and in handling complexities in the evolving interface such as entropy conditions and weak solutions. A detailed description of level-set can be found in (Sethian (1999)).

The general idea is to define a function $\phi(p, t)$ over the whole space-time domain, where p indicates a point in the space and t represents the time variable. The initial surface is the zero level-set of an implicit function for $t = 0$: i.e. the starting surface is the set of points p so that $\phi(p, t) = 0$ the value of the ϕ function for non-interface points is, the signed distance from the surface where 'signed' means that different signs indicate internal or external position with respect to the closed surface. The discrete formulation requires that the value of ϕ is initially evaluated over all the mesh points and the steering equations would require the update of each point at each time step. This computational complexity of order $O(n^3)$ at each time step could be unacceptable in many practical cases so many faster update solutions have been proposed; the most common are:

- Narrow band: the updated mesh points are only those in a small stripe surrounding the interface, this will reduce computation to $O(n^2)$ but as soon as the interface is approaching the stripe boundary all values must be recomputed (more details in (Adalsteinsson & Sethain (1995))
- Octree: relevant mesh points and high detailed surface regions can be divided in sub-grids, e.g. a cube can be split in eight sub-cubes (the method name come from here) and the procedure can be iterated for higher precision. More details can be found in (Losasso et al. (2004))
- Sparse block grid: The whole domain is subdivided in blocks representing clusters of the original grid points (they can be overlapped or not) then the update is evaluated only on blocks containing the surface narrow band. (Further details in (Bridson (2003))

Level-set evolution

Accordingly to the previous description the set of points belonging to the surface will satisfy the zero level-set condition: $\phi(\mathbf{p}(t), t) = 0$ where $\mathbf{p}(t)$ represents the path of a surface point. The first temporal derivative will become:

$$\frac{d\phi(\mathbf{p}(t), t)}{dt} = \frac{\partial \phi}{\partial t} + \nabla \phi \cdot \frac{d\mathbf{p}(t)}{dt} = 0 \tag{8}$$

Considering the normal unitary vector to a generic point of ϕ for a generic level-set (contour), $\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$ the speed function v along this direction is then:

$$v(t) = \frac{d\mathbf{p}(t)}{dt} \cdot \mathbf{n} \tag{9}$$

obtaining:

$$\frac{d\phi(\mathbf{p}(t), t)}{dt} = \frac{\partial \phi}{\partial t} + v(t) |\nabla \phi| = 0 \tag{10}$$

In the cases considered in the following and in many physical cases the interface evolution is ruled by a Hamilton-Jacobi equation and the local velocity can be assumed proportional to the local curvature of the surface itself:

$$v = v_0 - \epsilon \kappa \tag{11}$$

where ϵ is a constant and

$$\kappa = \frac{\phi_{xx}\phi_y^2 - 2\phi_y\phi_x\phi_{xy} + \phi_{yy}\phi_x^2}{(\phi_x^2 + \phi_y^2)^{3/2}} \tag{12}$$

We can rewrite ϕ as follows:

$$\frac{\partial \phi}{\partial t} = -v_0 |\nabla \phi| + \epsilon \kappa |\nabla \phi| \tag{13}$$

Liquid snowflakes surface evolution

Assuming that the ice is radiated with an external electromagnetic radiation normal to its surface (a focused beam with a gaussian shape) the heat will follow an exponential decay penetration, accordingly to the following heat propagation equation:

$$I(r, z) = I_0 e^{-(r/r_b)^2 - \alpha z} \tag{14}$$

where I_0 is the central beam intensity, r_b is its standard deviation while r is the in-plane coordinate for the ice surface. Furthermore z is the coordinate orthogonal to the ice surface and α is a penetration coefficient. The formation of the liquid snowflake is then