ANNUAL REPORT &
RESEARCH PROGRAMME

2020

J.M.Burgerscentrum
Research School for Fluid Mechanics

Research School for Fluid Mechanics
TUD, TUE, UT, RUG, WUR, UU

Annual Report & Research Programme 2020

JM Burgerscentrum

Research School for Fluid Mechanics
TUD, TU/e, UT, RUG, WUR, UU

Correspondence for reproduction

JM Burgerscentrum, Mekelweg 2, 2628 CD Delft, The Netherlands

jmburgerscentrum@tudelft.nl

www.jmburgerscentrum.nl

Burgers Program for Fluid Dynamics (University of Maryland, USA) http://hydro.umd.edu

This report includes contributions by the Participating Groups, Industrial Board and Contact Groups.

Preface

It is our pleasure to present the year 2020 Annual Report of the J.M. Burgerscentrum for Fluid Mechanics. 2020 was a special year in the sense that we were confronted with the Covid-19 pandemic, which had quite some impact on how the participants of the centre had to cope with their day-to-day work life activities. But most were able to find new ways of working, while keeping a focus on the high-quality research results. All JMBC participants really deserve recognition for this. Despite the more challenging working conditions, both the scientific staff, the PhD students and the postdocs were able to continue executing their research plans. Collaborations were continued, albeit mainly “online” instead of “in-person”. Some took the opportunity to focus on preparing journal publications based on earlier obtained results, instead of generating new simulation data and experimental data. Obviously, such publications will only become visible in the next annual report.

The present annual report of the J.M. Burgerscentrum provides an overview of the activities of our research school during the year 2020. The core of the report consists of the description of the research projects, carried out by the JMBC groups. In each project report the relevant information (title, theme, staff involved, Project aim, achievements, publications, funding source, application, etc.) is given. As usual, a number of research highlights are presented. The Annual Report also provides general information about the research school, such as its goals, its organization, and its relation with industries and technological institutes.

The number of PhD projects carried out by the JMBC groups has remained large (approximately 350). In addition, about 60 Postdocs are registered at the school. Although the sponsoring of such projects directly via the universities has become almost non-existing, the financing of projects via NWO and via industries and technological institutes remains at a high level. Also, the sponsoring of projects via the European Research Council is quite substantial. Finding funds for our PhD projects remains an important task for all JMBC groups.

In the year 2020 the following JMBC courses were organised: ‘CFD 1’, ‘Turbulence’, ‘Computational Multiphase flows’, and ‘PIV’. The latter two courses were given fully online. The number of courses that were realized was less than originally planned, as some were postponed to 2021 due to the pandemic.

All these courses are organised in order to give the JMBC PhD students the opportunity to deepen their knowledge in various aspects of fluid dynamics, but also to widen their perspective and give them an overview of the wide field of fluid dynamics. It is therefore highly recommended that our PhD students take the opportunity to participate in these courses.

The activities of the JMBC continue to ensure that the Dutch fluid mechanics field is a lively and well-connected community, with numerous mutual appointments at collaborating groups and collaborative research projects in which multiple JMBC groups participate. The JMBC contact groups continue to be an important instrument for cohesion. These contact groups stimulate interaction and collaboration between researchers, developers, and users. This is done through organizing regular meetings aimed at getting to know each other’s activities and learn about developments and applications.

Another important instrument to maintain the coherence throughout the research school is the annual JMBC meeting: the two-day Burgers Symposium. Due to the pandemic, the Burgers Symposium 2020 in Lunteren has been cancelled. This is very unfortunate, but given the serious circumstances there was no other choice.

Occasionally we are approached by university groups that are potentially interested in becoming a member of our research school. For admission of a new group, we follow a standard procedure. The group leader of the group that wishes to participate needs to submit a written motivation, a research plan and CVs of the staff members involved. Based on this application the scientific director consults a number of professors in the JMBC about the candidate group. The application and the advice are then sent to the board of the JMBC, which takes the final decision.

We are very pleased that in 2020 a new group joined the JMBC: the group Mesoscale Chemical Systems (MCS) of the Faculty of Science and Technology at the University of Twente, consisting of Prof. Han Gardeniers, Dr Niels Tas, Dr David Fernandes Rivas, Dr Arturo Susarrey Arce and PhD students + postdocs. The MCS group investigates the influence of mesoscale phenomena on the macroscopic outcome of chemical processes and focuses on the design, modeling, fabrication and testing of 3-dimensional micro and nano structures and their integration in chemical reaction and separation or miniaturized physical and chemical analysis units.

Certainly to our regret, two groups have decided to leave the JMBC: the group of Radiation Science and Technology and the group of Geoscience and Remote Sensing (both from Delft University of Technology). Their research has developed in a direction that is now maybe slightly more remote from the fluid mechanics centre of gravity. The two groups are greatly acknowledged for their participation in the JMBC over the past years.

In 2020, some prestigious awards were obtained by a number of members of the JMBC. A few are mentioned below.

- Anouk Bomers was the recipient of the KIVI Hoogendoorn Fluid Mechanics Award for the best PhD thesis defended in the academic year 2019-2020. The title of her PhD thesis is: 'Hydraulic modelling approaches to decrease uncertainty in flood frequency relations'.
- Philip de Goey (Professor of Combustion Technology, Dept Mechanical Engineering, TU/e) has been awarded an Advanced Grant from the European Research Council (ERC) to carry out research on combustion of metal fuels. Although this type of combustion is not a new phenomenon, there is a fundamental lack of understanding of this process. In his ERC research programme, he will focus on gaining insight into the fundamental aspects of combustion of metal fuels, by a combined experimental – numerical approach. This type of combustion has important applications in, as it is essentially CO2-free.
- Jerry Westerweel (Professor of Fluid Dynamics, Faculty 3mE, TUD) has been awarded an Advanced Grant from the European Research Council (ERC) to carry out research on impulsive flows. In this research, he will focus on fluid flows around objects undergoing strong and sudden motion change, aiming to gain insight into the fundamental aspects of such impulsive flows. The results of this research programme are relevant for applications in aerodynamics, wind energy, maritime and offshore technology, and other fields.
- Jeroen van Oijen (associate professor at Mechanical Engineering, TU/e) has been awarded a Vici Grant. This NWO grant will enable him to develop a revolutionary combustion engine that converts renewable fuels into clean power. It allows him and his team to develop the advanced computer models and laser diagnostics that are needed to optimize the sustainable combustion process.

The JMBC has opened a discussion forum on a specific medical care problem, related to ventilation therapy of corona infected patients in intensive care units. The purpose of this forum is to start a discussion among fluid flow experts on various ventilation methods in order to gain insight in the potential spread of the virus around the patient. This is expected to lead to a research programme in the next years.

Jointly with the research school Engineering Mechanics, the JMBC forms the Centre of Excellence 'Fluid and Solid Mechanics'. This centre also forms the basis of the 4TU Research Centre Fluid & Solid Mechanics (FSM). The J.M. Burgerscentrum forms an excellent example for successful collaboration between the 4 Dutch technical universities, with strengthening from the participating teams from the universities of Groningen and Utrecht. Collaboration between JMBC groups and teams in the industry and in the technological, as represented in the JMBC Advisory Board, is also of great importance to demonstrate the societal impact of the fundamental and applied research efforts.

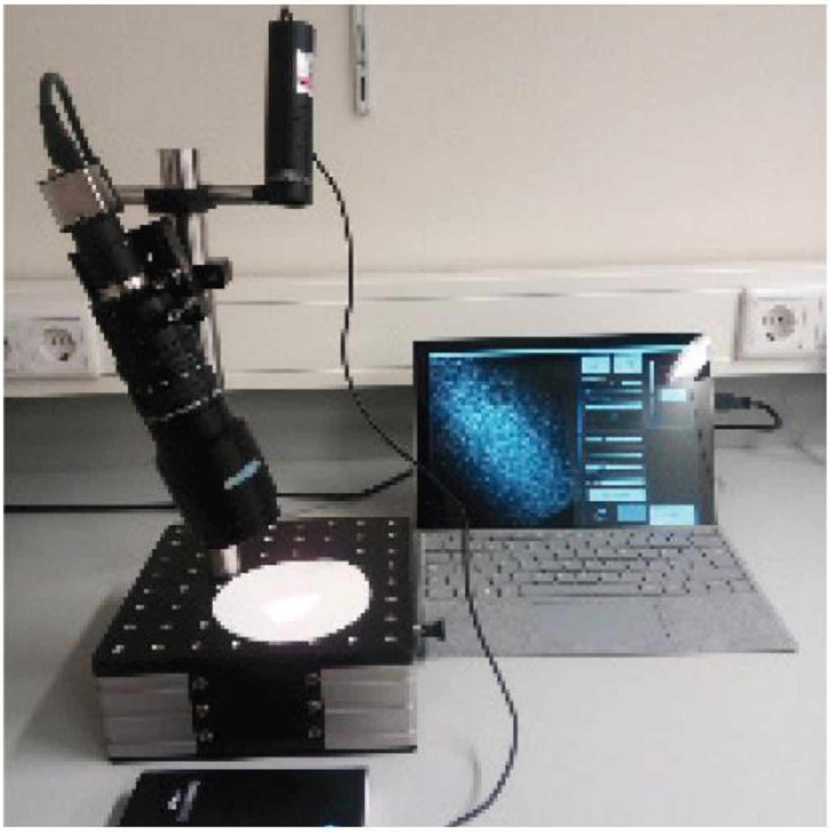
Due to the enthusiasm and the combined knowledge, skills and facilities of the participating research groups, the JMBC continues to be a very stimulating, multidisciplinary environment for advanced research in fluid mechanics and for the education of talented graduate and postgraduate students. The board and the management team of the JMBC highly value the large effort of the staff of the JMBC in reaching the goals of the research school.

Prof.dr.ir. CJ van Duijn
Chairman of the JMBC Board

Prof.dr.ir. RAWM Henkes
Scientific Director JMBC

Note: in 2020 Prof.dr.ir. G.J. van Heijst still served as the Scientific Director of the JMBC

CONTENTS



Contents (page 7)

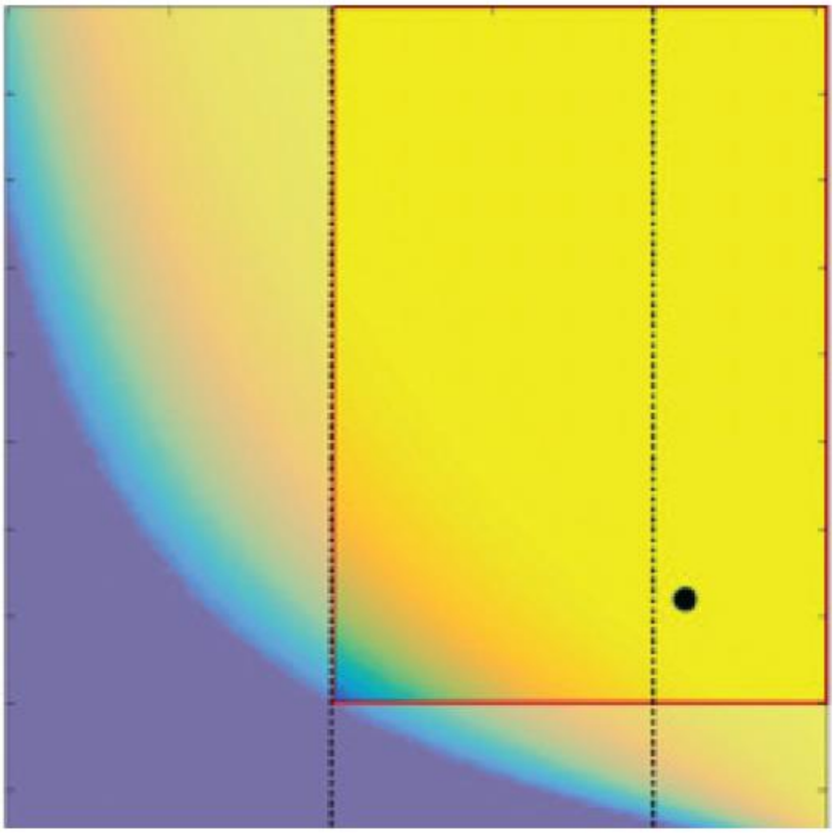
Organisation (page 13)

Highlights (page 21)

Research from participating groups (page 49)

Delft University of Technology (TUD) (page 53)

ORGANISATION



Participation

The JM Burgerscentrum (JMBC) is the national research school for fluid mechanics in The Netherlands. Its main goals are:

- Stimulation of collaboration of the participating groups with respect to their research efforts. The JMBC aims at being one of the leading institutes for fluid mechanics in the world.
- Organization of advanced courses for PhD-students. These courses are also attended by postdocs and by researchers from industries and technological institutes.
- Co-operation with industries and technological institutes. The aim is to promote the use of up-to-date knowledge on fluid mechanics for solving practical problems.
- Strengthen the contacts between fluid mechanics research groups at Dutch universities and the international fluid mechanics community.

About 60 professors with their groups, in total about 200 senior scientific staff members, participate in the JMBC. These groups are located at the universities in Delft, Eindhoven, Twente, Groningen, Wageningen, and Utrecht. The various fluid-dynamics groups are based in different departments, and in different disciplines: in Civil Engineering; Mechanical Engineering, Maritime Technology, (Applied) Physics, Aerospace Engineering, Applied Mathematics, Chemical Technology, Biology, and in Physical Oceanography. The professors with their senior staff form the Council of Project leaders, which meets regularly. At the end of 2020, approximately 350 PhD-students and 60 postdocs were registered as participants of the JMBC.

The scientific director of the JMBC is assisted by the Management Team, consisting of the local directors in Delft, Eindhoven and Twente (who are also representing the groups in Groningen, Wageningen and Utrecht). The running of the JMBC takes place under final responsibility of the Board of the JMBC.

The research projects carried out by the JMBC groups can be grouped into three main research themes:

- Complex dynamics of fluids
- Complex structures of fluids
- Mathematical and computational methods for fluid flow analysis.

A number of contact groups in different topical areas are active, in the sense that they strengthen the network between researchers at different groups, promoting the exchange of expertise and experience between the participating groups. The various JMBC groups have intensive, active contacts and close links with industries and technological institutes in The Netherlands. This connection is formally facilitated by the Industrial Advisory Board, with members representing a large number of companies and technological institutes.

The JMBC research groups have various scientific contacts with research groups in other countries, often in the form of individual collaboration projects, but also in the form of organised networks. This international setting implies joint publications with other researchers from all over the world, and also exchange of staff: external visitors to the JMBC groups and JMBC staff visiting foreign fluid-mechanics groups.

As common practice in the scientific community, the research groups present their work at international conferences and in the form of journal publications. The number of publications from JMBC staff in well-known scientific journals is considerable.

Together with Engineering Mechanics (the Dutch research school on solid mechanics) the JMBC forms the 4TU Research Centre for Fluid & Solid Mechanics (FSM). This Research Centre has been recognized as a "centre of excellence" in The Netherlands and has in the past received significant funding by the Dutch Government for stimulating new research areas in fluid and solid mechanics.

An important activity of the JMBC is the organisation of the Burgers Symposium, which is the annual two-day meeting of the research school. This annual meeting is usually attended by more than 250 participants (both staff, PhD students, and postdocs). In addition to the plenary Burgers Lecture and the Evening Lecture the Symposium programme includes oral and poster presentations by the JMBC PhD students. In the year 2020 the Burgers Symposium had to be canceled due to the Covid-19 pandemic.

Each academic year the JMBC organises a number of special courses, meant primarily as advanced fluid-dynamical education of the PhD students and postdocs of the JMBC. The topics of these courses varies from one year to another, although some courses are given every other year. These courses are also open to participants from other research schools and from industry.

Overview of the university participating in the JMBC

The fte figures for Scientific staff are *effective fte*, based on the following weight factors: Professor 0.3 fte | Part-time professor 0.1 fte | Associate professor 0.4 fte | Assistant professor 0.4 fte. The figures for PhD students and Postdoctoral fellows represent *numbers*.

	TUD	TU/e	UT	RUG	WUR	UU	Total
Scientific staff (effective fte)	24.7	19.5	16.8	1.5	2.5	1	66
PhD students	125	117	100	3	8	2	355
Postdocs	5	19	32	3	1	0	60

Industrial Board

It is privilege, as the chair of the Industrial Advisory Board, to contribute to the 2020 Annual Report of the J.M. Burgerscentrum (JMBC), the research school for fluid dynamics in The Netherlands. The Industrial Advisory Board has 16 members, representing Industries and Technology Institutes in the Netherlands. As an example of the industrial involvement in the JMBC, mentioned can be that Industries, Technology Institutes and Academia are undertaking a collaboration project (with NWO funding) in the area of bubbly flow in electrolysers and industrial granular flow. The focus is on developing new models hand-in-hand with experimental validation. Another initiative is in the area of electro-chemical reactors, aiming at creating new functionalities via a combination of traditional and additive manufacturing methods. This is a close collaboration in the areas of membrane materials, catalysis and manufacturing methods. As can be seen from these examples, we are seeking interaction between industries, universities, and institutes, and push for projects in both existing and new business areas and on top of that interactions between disciplines like process engineering, materials, mechanical engineering, etc. Like in recent years, also in 2020 the fast development of computational and data science and digitalisation was on the agenda. Sensors become cheaper and faster, while models can be operated 'real time'.

These developments have significant effects on workflows and interactions between fundamental sciences and process engineering tools. Open source and open access are also now recognized at EU level as important vehicles to validate, disseminate and maintain knowledge developments. An important aspect in creating easier access to models is the availability of the models and enough and affordable hardware to run the models on. In 2020 this has been explored further, in combination with earlier mentioned open source and open access developments.

Besides the interactions between technical disciplines, the fluid mechanics community can also play a pivotal role in the interactions with environmental sciences. Examples can be found in dikes, channels, wind, and other similar developments, which are important for the future of a country like The Netherlands. Increased attention should be given to this in the next few years.

Ir. P Veenstra

Shell Projects & Technology
Chairman of the Industrial Advisory Board

Contact Groups

Contact group "Multiphase Flow"
The objective of the contact group Multiphase Flow is to stimulate interaction and collaboration between researchers, developers, and users in the area of multiphase flow from universities, institutes and industries. This is done through organizing regular meetings (once or twice per year) aimed at getting to know each other's activities and to learn about developments and applications of multiphase flow technology. This will provide a good forum to identify the needs of the users and to bring to the attention new possibilities for applying multiphase flow research results. Industry, a research institute or a university in turn act as host of the meetings. The program consists of a series of lectures on a specific theme and a visit of some of the local multiphase flow facilities. Examples of themes covered are: dynamic multiphase flows, multiphase flows with surface-active agents, and innovation with multiphase flow. Due to the Covid-19 pandemic no dedicated meeting was held in 2020. The JMBC PhD Course "Computational Multiphase Flow" was given on-line on November 4-6.

Contact: Dr. Y Tang (TUE), Dr.ir. SG Huisman (UT), Prof.dr.ir. C Poelma (TUD)

Contact group "Computational Fluid Dynamics (CFD)"
Computational Fluid Dynamics (CFD) has established itself as an indispensable scientific discipline at the intersection of physics, engineering, mathematics and computer science. Society relies more and more on numerical simulations, while at the same time the field becomes more and more specialized. In a series of JMBC courses the state of the art in computational fluid dynamics is presented from an introductory level to state-of-the-art methods. The aim of the contact group CFD is to bring together developers, scientific staff, PhD students and users to share new developments and experiences. To that end the CFD contact group organizes an annual meeting, which provides a platform for discussing the latest development. This event also aims to promote the interaction between CFD users from academia, industry and research institutes. Young researchers are encouraged to present their latest work during this one-day event. Due to the pandemic, no annual CFD contact group meeting was held in 2020.

Contact: Dr.ir. MI Gerritsma (TUD), Prof.dr.ir. RWCP Verstappen (RUG)

Contact group "Combustion"
The contact group Combustion is an informal network between the groups active in combustion research in Delft, Eindhoven, and Twente, bringing together the researchers in the Netherlands in an international context. The combustion topic is interpreted in a broad sense and covers solid, liquid and gaseous fuels, with a focus on renewable fuels.

About once every three years the contact group Combustion organizes the JMBC course on Combustion, bringing PhD students and other academic and industrial researchers to the forefront of experimental, theoretical and numerical research on fundamental and applied combustion. The latest edition took place in January 2019, and the next one will be organized in January 2022 in association with ERCOFTAC. Since many years the JMBC groups also play an important role in the organization of the annual COMBURA symposium. This symposium is the major annual event in the Netherlands for exchange of information on combustion research and its applications. Its goal is to enhance the mutual collaboration between the different academic and industrial researchers, and to interest more industrial parties for the fundamental research on combustion. In 2020, the COMBURA conference was not held because of the COVID-19 pandemic. The next COMBURA is to be held on 10-11 November 2021 with keynote lectures by Prof. Christof Schulz (University of Duisburg-Essen), Prof. Alessandro Parente (ULB Brussels), and Dr. Martien Visser (Hanze University of Applied Sciences, Groningen) embedded in an interesting program of oral and poster presentations on the research by the participants.

Contact: Dr.ir. RJM Bastiaans (TUE), Dr.ir. JBW Kok (UT), Prof.dr.ir. JA van Oijen (TUE), Prof.dr. DJEM Roekaerts (TUD)

Contact group "Lattice-Boltzmann techniques"
The Lattice-Boltzmann schemes can be seen both as flexible and efficient solvers for macroscopic fluid equations or as particle-based simulation techniques which make close contact with the kinetic theory of gases. It is this last feature that allowed, in recent years, the partial disclosing of the huge potential of the method. The Lattice Boltzmann method has demonstrated great accuracy and performance in dealing with multiphase and multicomponent flows, from laminar to turbulent, in presence of simple or complex boundary conditions. Additionally, the method allows the study of colloidal systems, of complex fluids and of thermal flow problems.

The contact group promotes the organization of educational and research events, also in synergy with other national and international organizations.

Contact: Prof.dr. F Toschi (TUE)

Contact group "Turbulence"
Turbulent flows are omnipresent in industrial applications and the environment. Owing to the non-linear character of the governing Navier-Stokes equations, the structure and dynamics of turbulence is complex. It is for these reasons that turbulence has been studied for already more than a century, in particular after the pioneering work of Osborne Reynolds on transition and turbulence in pipe flow. While early research focused on understanding of turbulence in single-phase flow, research on turbulence nowadays addresses a much broader class of turbulent flows such as e.g. turbulent multiphase flows, turbulent reacting flows, turbulence in supercritical fluids, etc.

The contact group "Turbulence" organizes annual meetings between researchers of the JM Burgerscentrum active in the field of turbulence with the aim to strengthen contact between them and to exchange results and experience. PhD students and other researchers are given the opportunity to present their results in an informal setting that promotes discussion. The meetings take about a day with a program consisting of typically 7-8 talks from different researchers/ groups, usually followed by a tour through the laboratory of the hosting institute and a "drinks session" at the end of the day. The program of a meeting typically covers both fundamental and applied research topics as well as the development of experimental techniques and numerical simulation methods for turbulent flows. Due to the pandemic, the annual contact day was skipped in 2020.

Contact: Dr. R Pecnik (TUD), Dr.ir. WP Breugem (TUD), Dr.ir. RJAM Steven (UT), Dr. M Duran Matute (TUE)

Contact group "Experimental Techniques"
The contact group Experimental Techniques forms a platform where experiments and experimental techniques can be shared and discussed. The main function of the contact group is to organize meetings with experimental fluid dynamics research groups, including research on turbulence, multiphase flows, granular flows and microfluidics. An important contribution of the contact group is the organization of the JMBC course on Experimental Techniques in Fluid Mechanics. The course is very popular among the JMBC members as it gives a broad overview of advanced experimental techniques commonly used in fluid mechanics laboratories. The last course was organised in the University of Twente in June 2019 and counted 40 participants and 13 lecturers from all over the country. The next course will be held in November 2021, again at the University of Twente.

Contact: Prof.dr.ir. M Versluis (UT), Prof.dr.ir. C Poelma (TUD), Dr.ir. RPJ Kunnen (TUE), Dr. A Marin (UT)

Contact group "Biological Fluid Mechanics"
More and more research is conducted at the interface between biology and fluid mechanics. This happens within many disciplines, ranging from physiology (for example, the interaction between blood flow and vessel walls) to aerodynamics (for example, the biofluidmechanics of flying and swimming animals). All these topics deal with the interaction between fluids and a complex deforming geometry. Therefore, they require similar experimental, numerical and analytical approaches.

One of the main challenges in this highly interdisciplinary research field is to bridge the gap between physics (fluid mechanics) and medical/biological sciences. To stimulate this, we aim to bring together researchers that work on Bio-Fluid Mechanics, by organizing seminars, workshops and courses on this topic. Although the contact group is affiliated with the JM Burgerscentrum, the participation from researchers from outside the JM Burgerscentrum is highly encouraged.

As part of this effort, we organized in 2020 an online seminar with the title "Biofluidmechanics of flying insects and bioinspired drones", and which was followed by the JMBC/WUR PhD thesis defence of Dr Wouter van Veen. Because both the seminar and defence were online, it was attended by a large range of researchers from all around the world.

Contact: Prof.dr. S Kenjeres, Dipl.-Ing. (TUD), Dr.ir. F Muijres (WUR), Dr.ir. B Borsje (UT)

Contact group “Microfluidics”

The contact group “Microfluidics” was established in 2005. The purpose of the contact group is to bring together students and postdocs interested in fluid dynamic aspects of microfluidics and give them a forum for presenting their results and exchanging ideas. Also, the contact group serves as a platform to exchange information about relevant conferences, workshops, courses, and research grant opportunities. Topics of interest include wetting and capillarity-driven flows, two-phase flow, micro-mixing, drop generation and control, emulsification, contact line dynamics, flow visualization, and measurement techniques. Attention is also given to also related applications such as microfluidic devices for medical diagnostics, water quality monitoring, and advanced cell culture systems.

Members of the contact group organize the JMBC courses “Capillarity-driven flows in microfluidics” and “Micro- and Nanofluidics”. In the latter course, the participant will learn about micro- and nanofluidic principles, technology, and applications, but also get extensive hands-on experience with designing, making, and testing microfluidic devices.

Students and researchers, who are interested in the activities of the group and want to attend our symposia, are invited to contact the organizers of the contact group and have their name added to the mailing list.

Contact : Prof.dr. F Mugele (UT), Prof.dr.ir. JMJ den Toonder (TUE)

Burgers Program for Fluid Dynamics at the University of Maryland (USA)

Inspired by the intellectual heritage of Johannes M. Burgers, who had a second career (1955 - 1981) at the University of Maryland after his retirement at the Technical University of Delft , the mission of the Burgers Program for Fluid Dynamics is to enhance the quality and international visibility of the research and educational programs in fluid dynamics and related areas at the University of Maryland, in partnership with the J.M. Burgerscentrum (JMBC). Fluid dynamics in this context is viewed to include a broad range of dynamics, from nanoscales to geophysical scales, in simple and complex fluids.

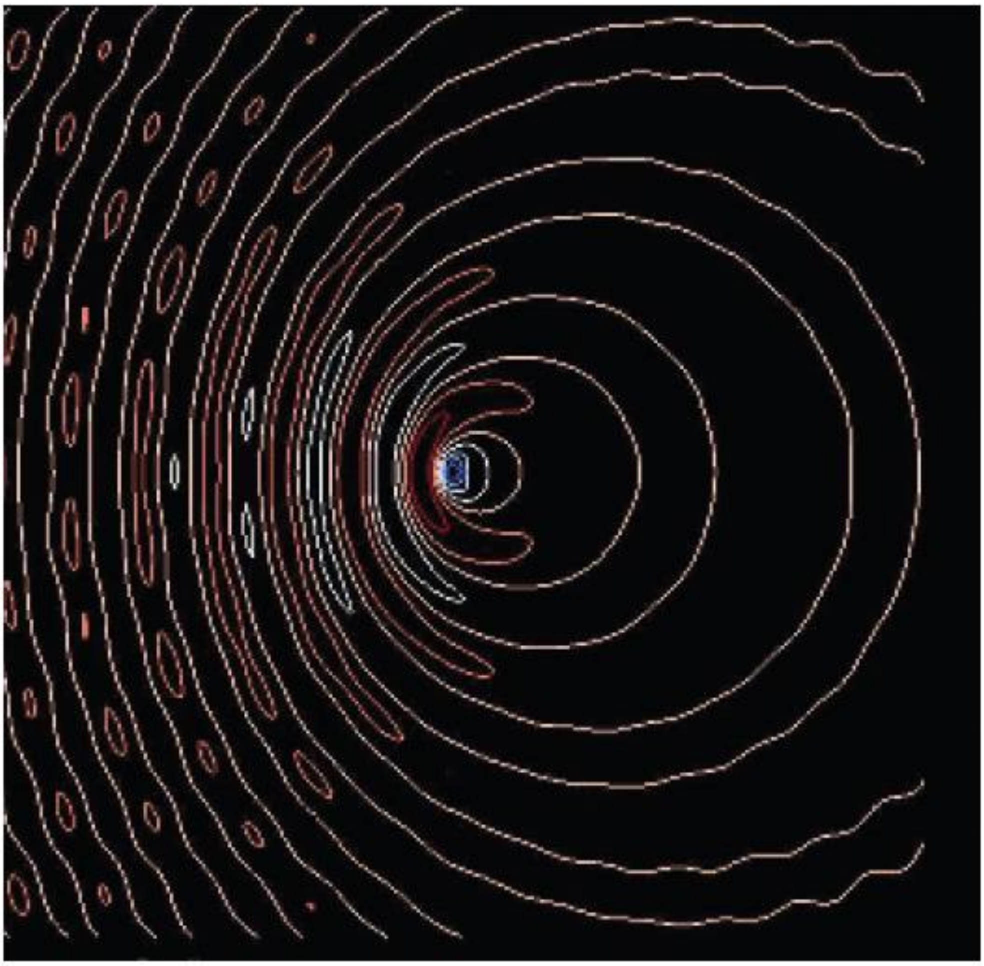
The establishment of the Burgers Program was celebrated with an inaugural symposium at the University of Maryland in November 2004. Gijs Ooms, then Scientific Director of the JMBC, gave a lecture on the life and legacy of Burgers on that occasion. The interdisciplinary Burgers Program encompasses almost 80 faculty members spread over 22 different units in the College of Computer, Mathematical and Natural Sciences and the A. James Clark School of Engineering. For detailed information, go to http://www.burgers.umd.edu/.

There have been numerous faculty and student exchanges between groups of the JMBC and the Burgers Program. Visitors to Maryland have come from Delft University of Technology and Eindhoven University of Technology, Leiden University, Twente University , and Utrecht University. Over thirty journal articles have resulted from these exchanges. At the annual Burgers Symposium at Maryland in November of each year, the Burgers Lecture has been given by JMBC faculty: Frans Nieuwstadt, Bruno Eckhardt, Gijs Ooms, Detlef Lohse, Wim van Saarloos, Kees Vuik, Wim Briels and Henk Dijkstra as well as by several others from France, Germany and the United States. In the fall of 2020, Jim Kok of the University of Twente gave the Burgers Lecture at the 17th Annual Burgers Symposium. His presentation (online due to the pandemic travel restrictions) was entitled: “Combustion Dynamics in Turbulent Flames and the Path to Instability”.

Each spring semester the Burgers Program holds a PhD student/Post-doctoral Fellow showcase Symposium, together with fluid dynamics groups from Johns Hopkins University and George Washington University. Five or six seminars are offered each academic semester in the Fluid Dynamics Reviews series. Beginning in 2010, the Burgers Program also began offering advanced level, week-long Research Summer Schools. The subjects have been Topics in Turbulence (twice), Granular Flows - from Simulations to Astrophysical Applications and Data Assimilation in the Geosciences. Students from the JMBC have participated in each of these. The one on ‘Fire Safety Science’ was planned for 2020, but had to be postponed until 2021 (again due to the coronavirus pandemic).

Prof. James Duncan
Keystone Professor at the Department of Mechanical Engineering
University of Maryland

HIGHLIGHTS



Hydraulic modelling approaches to decrease uncertainty in flood frequency relations

Anouk Bomers and Suzanne Hulscher[1]

[1] University of Twente, Faculty of Engineering Technology, Water Engineering & Management

River floods are a major global hazard causing extensive damage and loss of lives. To protect the hinterland from severe inundations, flood defences are commonly designed according to appropriate safety levels that are determined based on a statistical return period. To estimate discharges associated with different return periods, flood frequency analyses are used that fit a distribution to the data set of annual maximum discharges.

The data sets of measured annual maximum discharges are generally in the order of 100 years. Consequently, the predicted design discharges with a return period of e.g. 100,000 years are based on extrapolation and therefore highly uncertain (Figure 1). To decrease the uncertainty of flood frequency relations, historical flood information can be added to the data set of measured discharges.

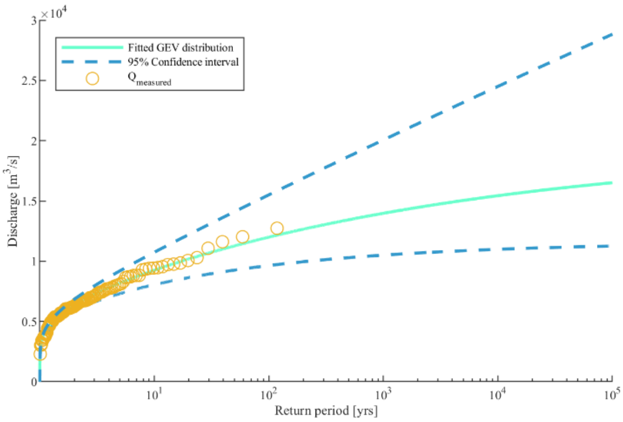


Figure 1: Fitted GEV flood frequency curve based on ~120 years of annual maximum discharge observations of the Rhine river at Lobith (German-Dutch border) and corresponding 95% confidence interval.

During this research (which was part of a PhD project by the first author), we aimed to study the effect of extending the data set of measured discharges on the reduction of the 95% uncertainty interval of flood frequency relations. The data set was extended with reconstructed historic flood events using hydraulic modelling approaches. The Rhine delta was used as a case study (Figure 2), but the proposed methodologies can also be applied to other river basins and coastal areas provided that sufficient historical data are available.

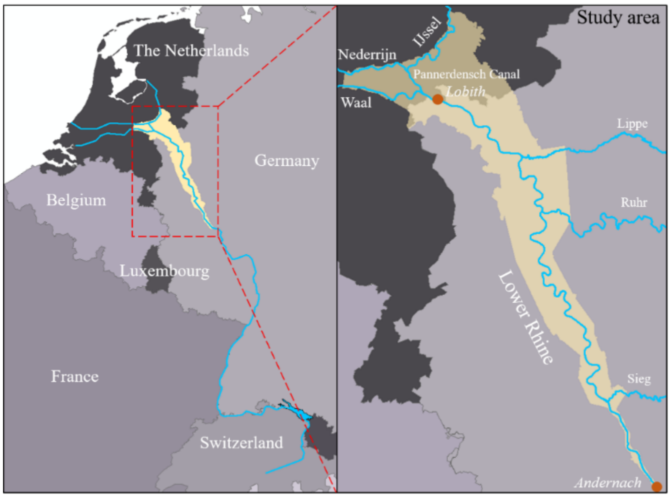


Figure 2: The Rhine river catchment area (left) and the study area (right).

A drawback of reconstructing historic flood events with the use of hydraulic models is the high computational cost. Because historical information is limited and uncertain, many model runs have to be performed to include these uncertainties in the analysis. Therefore, the hydraulic model must be efficient in terms of model accuracy and computational time. We first developed a fully two dimensional (2D) hydraulic model, after which we simplified this model to achieve a data-driven model, referred to as a response surface surrogate model. By simplifying the model, computational times are reduced. However, it is important that the simplified models are still capable of reproducing the desired output.

Various 2D grids were evaluated in terms of model performance. Structured rectangular, unstructured triangular and hybrid (consisting of both structured and unstructured grid cells) grids with a high and low resolution were compared (Figure 3) as well as the performance of both non-calibrated and calibrated models based on simulated water levels. Furthermore, flow velocities in a meander bend were evaluated to assess the correctness of the physical processes. It was found that there are three important grid generated features that influence model results, namely: (1) bathymetry accuracy and (2) numerical friction, both as a result of grid resolution, and (3) numerical viscosity as a result of grid shape. Numerical friction and numerical viscosity have the same effect on model results as physical bed friction has, namely attenuating the discharge wave and in increasing the simulated water levels.

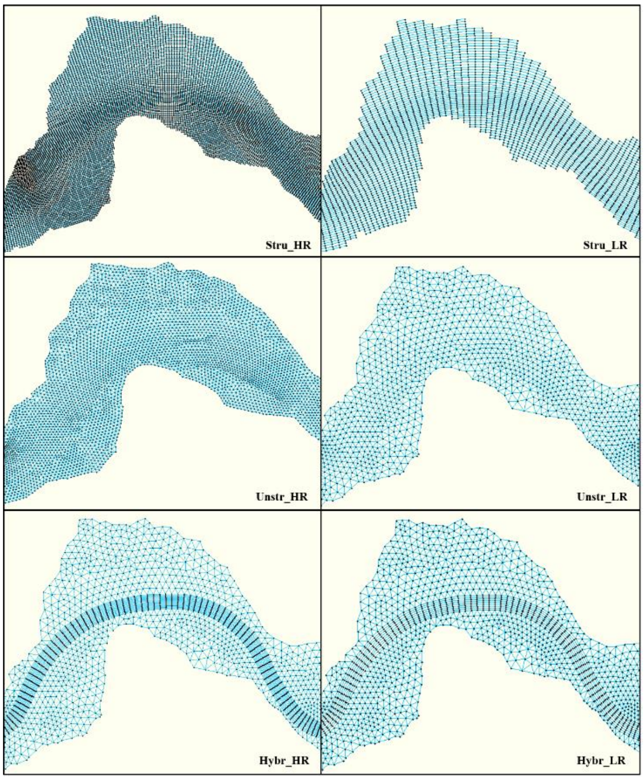


Figure 3: The six grids evaluated in which "Stru" refers to the triangular structured grids, "Unstr" to the curvilinear unstructured grids and "Hybr" to the grids having curvilinear grid cells in the main channel and triangular grid cells in the floodplains. HR and LR refers to high resolution and low resolution, respectively.

A 1D-2D coupled model was developed, also referred to as a lower-fidelity physically based surrogate model, to study whether this model can replace a fully 2D model to reduce computational time. In this simplified model, the main channel and floodplains are schematized by 1D profiles and the embanked areas are discretized on a 2D grid. This model was used to perform a sensitivity analysis to analyse which parameter has the largest impact on the maximum discharges during a flood event. In 1926 the largest measured flood event has occurred. This flood event was used as a case study. It was concluded that the model output is most sensitive to the roughness class with the largest share in surface area. Furthermore, it was found that the 1D-2D coupled model is capable of producing model results with the same accuracy as a fully 2D model. Therefore, the 1D-2D coupled model was used as a high-fidelity model.

A surface response surrogate model was developed with training data created with the 1D-2D coupled model. This data-driven surrogate model has no physical interpretation, but reproduces the input-output relations of the high-fidelity model based on simple mathematical functions. As a result, many model runs can be performed within a couple of seconds. The maximum discharge at Lobith (German-Dutch border) during the 1809 flood event was reconstructed based on measured water levels of surrounding locations. The confidence intervals of the 1809 maximum discharge are reduced compared to the results of existing methods that did not use hydraulic models to perform the reconstruction.

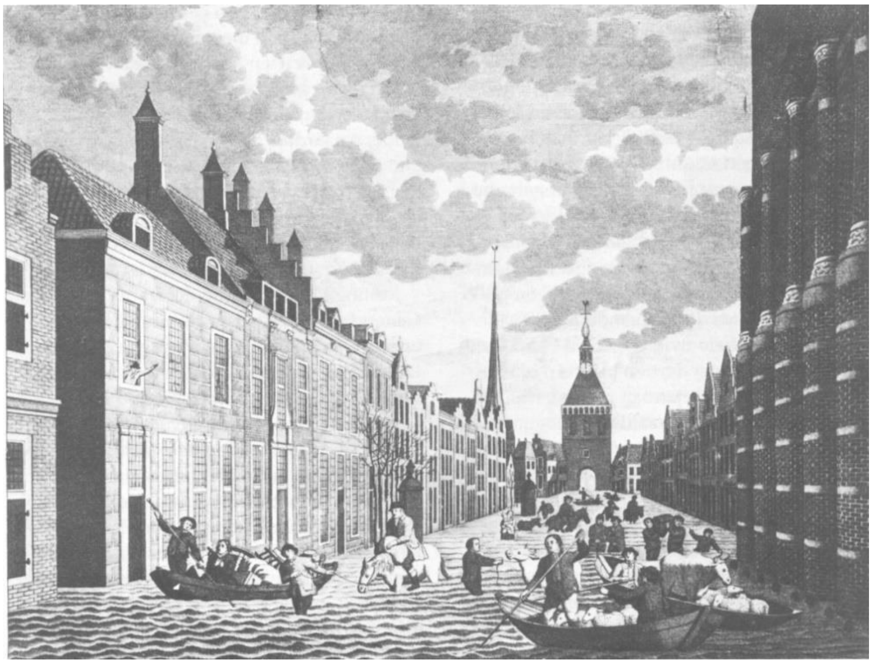


Figure 4: Painting of the 1809 flood event at Culemborg by J. G. Visser.

Before historic flood events can be added to the data set of annual maximum discharges, they must be normalized for natural and anthropogenic changes in the river system. To do so, the upstream discharges of the historic flood events were routed over the present geometry using the 1D-2D coupled modelling approach. The present system behaviour of the Rhine delta as a result of various upstream discharges has been shown. Dike breaches are included in the model domain as random input parameter resulting in various overland flow patterns. As a result, dike breaches and corresponding overland flow patterns may significantly change the discharge partitioning of and flood risk along the Dutch Rhine river branches (Figure 5).

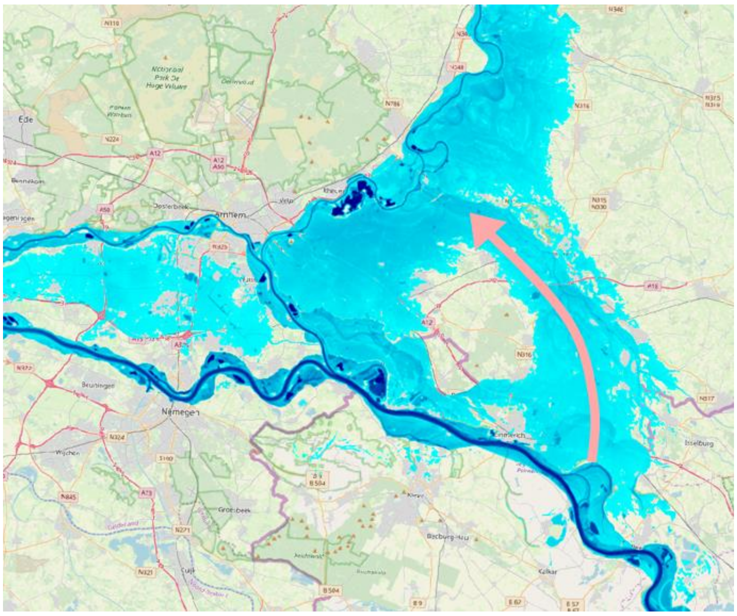


Figure 5: Dike breaches in Germany may result in significant overland flow (pink arrow) resulting in inundations of the Dutch hinterland influencing Dutch flood risk.

Finally, a continuous data set of annual maximum discharges of approximately 700 years was created. The data from the period 1772-2018, comprising of discharge and water level measurements, were extended with 12 historic flood events. The 12 historic flood events were normalized using the modelling approach developed in Chapter 5. Next, a bootstrap approach was used to sample discharges for the missing years in the historical time period, resulting in a continuous series. A flood frequency analysis was performed with the extended data set. The results of this analysis were compared with the flood frequency relation created by solely using measured discharges. It was found that uncertainty in flood frequency relations decreases if the length of the considered data set of annual maximum discharges is extended (Figure 6). Therefore, it is recommended to include as many historic flood events as possible in the considered data set such that the uncertainty intervals of flood frequency relations are reduced. This even applies if the magnitude of the historic flood event itself is highly uncertain. In this manner, future design discharges can be predicted with less uncertainty.

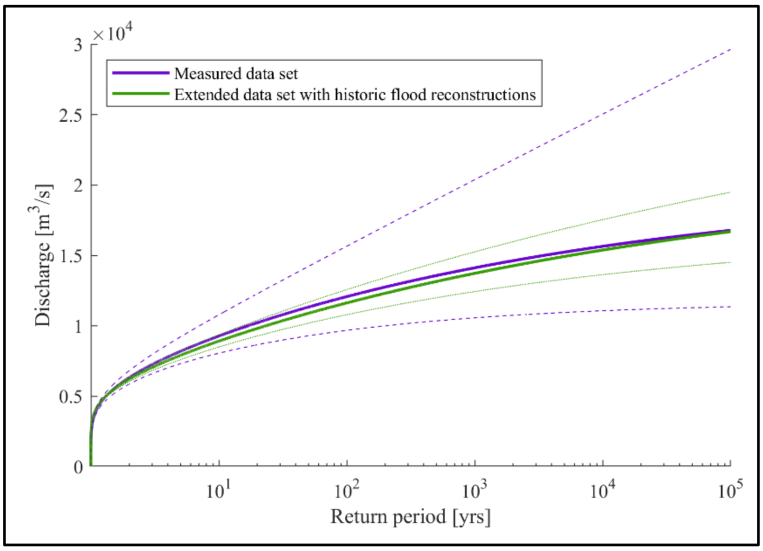


Figure 6: Flood frequency analysis performed with the measured data set of annual maximum discharges of ~120 years and with the data set extended with historic flood events resulting in a data set having a length of ~700 years.

The PhD research by Anouk Bomers was part of the research programme Floods of the Past-Design for the future with project number 14506 (Water 2015), which was (partly) financed by the Netherlands Organisation for Scientific Research (NWO). Anouk Bomers was the recipient of the KIVI Hoogendoorn Fluid Mechanics Award for the best PhD thesis defended in the academic year 2019-2020. For more details on the work, see Refs [1] – [6].

References

1. Bomers, A. Hydraulic modelling approaches to decrease uncertainty in flood frequency relations, PhD Thesis, University of Twente, 2000. https://research.utwente.nl/en/publications/hydraulic-modelling-approaches-to-decrease-uncertainty-in-flood-f
2. Bomers, A., Van der Meulen. B., Schielen, R.M.J., Hulscher, S.J.M.H. (2019). Historic flood reconstruction with the use of an artificial neural network. *Water Resources Research* **55** (11), 9673-9688. DOI: 10.1029/2019WR025656
3. Bomers, A., Schielen, R.M.J., Hulscher, S.J.M.H. (2019). Decreasing uncertainty in flood frequency analyses by including historic flood events in an efficient bootstrap approach. *Natural Hazards and Earth System Sciences* **19**, 1895-1908. DOI: 10.5194/nhess-19-1895-2019
4. Bomers, A., Schielen, R.M.J., Hulscher, S.J.M.H. (2019). Consequences of dike breaches and overflow in a bifurcating river system. *Natural Hazards* **97**, 309-334. DOI: 10.1016/j.envsoft.2019.03.019
5. Bomers, A., Schielen, R.M.J., Hulscher, S.J.M.H. (2019). Application of a lower-fidelity surrogate hydraulic model for historic flood reconstruction. *Environmental Modelling and Software* **117**, 223-236. DOI: 10.1007/s11069-019-03643-y
6. Bomers, A., Schielen, R.M.J., Hulscher, S.J.M.H. (2019). The influence of grid shape and grid size on hydraulic river modelling performance. *Environmental Fluid Mechanics* **19**, 1273-1294. DOI: 10.1007/s10652-019-09670-4

Energy entrainment from low-level jets benefits wind farm performance

Srinidhi Gadde and Richard Stevens[1]

[1] Department of Science and Technology, University of Twente

Low-level jets are the wind maxima in the lower regions of the atmosphere. Due to their significant influence on the power production of wind farms, it is crucial to understand the interaction between low-level jets and wind farms, which we study using large-eddy simulations. We find that the power production in the back of the wind farm is relatively high when the jet is above the wind farm. When the low-level jet is at turbine hub height, the production of the turbines in the back is limited. However, the production of turbines in the back is higher than anticipated when the jet flows under the wind turbines. The reason is that the negative shear of the jet creates a significant upward entrainment flux, which facilitates energy extraction from the jet by downstream turbines. While low-level jets are beneficial from the point of view of power production, our simulations also show that their presence induces significant cyclical variation in aerodynamic loading. This means that low-level jets increase the fatigue loading experienced by the turbines and this can negatively affect the turbine lifetime. Overall, our work underlines the importance of fundamental fluid dynamics research for the understanding of the flow dynamics in wind farms.

Introduction

Wind energy has been used for many different purposes since the Middle Ages, for, among other things, reclaiming land, sawing wood, and grinding grain. The Netherlands have a rich history and a leading role in its development, like, for example, the famous Kinderdijk. In recent decades, wind energy technology has undergone major developments. Today, wind energy is mainly used for electricity generation, and there is great interest in this technology because of its enormous potential as a clean energy source. To achieve the European climate targets, generating much more clean energy is necessary, and wind energy will play a crucial role in this development. However, to realize wind energy potential, many technological and scientific challenges remain. Here we discuss some of our recent work [2, 3] on modeling wind farm flow dynamics, showing the crucial role of fluid mechanics in this development.

Large eddy simulations of wind farms

Studying turbulent phenomena in the atmosphere is of fundamental importance for optimizing the design of wind farms and providing reliable production forecasts [4]. The flow in a wind farm is highly turbulent and strongly depends on atmospheric conditions. Figure 1 shows a visualization of the turbulent flow in a wind farm obtained from a large eddy simulation. The energy extraction by the wind turbines from the incoming flow creates wind turbine wakes, i.e. the low-velocity wind speed regions created behind the turbines. In the figure, these wind turbine wakes are visible as blue 'clouds' behind the turbines. The visualization shows the complex turbulent nature of the wind turbine wakes, which affects the performance of downstream turbines.

Figure 1: Flow visualization of turbine wakes from a large eddy simulation of a wind farm. Wind turbines extract energy from the incoming wind and produce a low-speed wake (blue smoke in the figure). Figure adapted from Gadde (2021) [1].

When downstream turbines operate in the wakes of the upstream turbines, their power production is reduced compared to turbines operating in free stream conditions. The reason for the lower power production, which is also known as 'wake effect', is the velocity deficit in the wake. Naively one would expect that the power production of turbines farther downstream in the wind farm continuously decreases as then there are more upstream turbines that harvest energy from the flow. However, after the first few rows, the combination of wake deficit and high turbulence intensity in the wake creates a downward turbulent entrainment that brings a high energetic flow from above the wind farm towards the wind turbine region. This vertical kinetic energy entrainment counteracts the wake effects and is beneficial for the power production of turbines farther downstream in the wind farm.

Influence of low-level jets on wind farm performance

Under typical atmospheric conditions, vertical kinetic energy transport in a wind farm is downward [5], pulling down the high-energetic geostrophic winds in the higher atmosphere towards the wind farm. However, the direction of this vertical kinetic energy flux depends on the wind shear in the atmospheric boundary layer. Typically, the wind shear at turbine height is positive, i.e. the wind velocity increases with height, which leads to the negative entrainment flux. However, atmospheric temperature variation, also known as thermal stratification, can give rise to negative wind shear situations under stable conditions. The effect of negative wind shear on entrainment flux in a wind farm is hitherto unexplored. Negative wind shear can be created in the atmosphere due to interesting lower-atmospheric phenomena such as low-level jets [6]. Low-level jets can be considered `high-speed atmospheric rivers', which are frequently observed worldwide [7, 8, 9]. Understanding the influence of low-level jets on wind farm performance is crucial as their strong winds provide a crucial contribution to the energy potential of a site. Figure 2 shows a typical low-level jet profile and the three main scenarios we consider, i.e. the low-level jet above, at the middle, or below the turbine rotor swept area. We studied these three scenarios in detailed large-eddy simulations [10], shown in Figure 2, which reveals that the flow physics for these three cases are pronounced differently.

Figure 3(a) shows the row-averaged power normalized by the first row's power production for the three cases. When the low-level jet is above the turbines ($z_{jet} > z_h$), the relative power production for turbines farther downstream in the wind farm is relatively high. The reason for this is the strong downward vertical kinetic energy flux that transports energy from the jet towards the turbines and thereby counteracts the wake effects. When $z_{jet} \approx z_h$, the power production continuously reduces towards the rear of the wind farm. The corresponding wake recovery shows that the velocity continuously drops in the downstream direction, which indicates that the wake recovery is negligible. Interestingly, when the jet is below the turbines ($z_{jet} < z_h$), the power production of the second row is severely affected due to the absence of turbulence in the wake of the first turbine row. However, the power production increases farther downstream due to wake-generated turbulence, and it shows an upward trend towards the back of the wind farm. For this case, the wake turbulence becomes significant behind the second turbine row, and subsequently, the turbines entrain high-momentum wind from the low-level jet below the turbines.

Energy budget analysis

To further understand the power production of downstream turbines, the planar-averaged vertical turbulent flux of streamwise momentum $\langle\overline{u'w'}\rangle$, normalized with the $\langle\overline{u'w'}\rangle$ value at the wall, is plotted in Figure 3(b). When the low-level jet is above the turbine rotor swept area ($z_{jet} > z_h$), there is a significant negative (downward) momentum flux, which extracts the jet's momentum and eliminates it towards the rear of the wind farm. However, when the low-level jet is below the turbines ($z_{jet} < z_h$), the turbines operate in the negative shear region, and a significant positive entrainment flux is created. As a result, the jet's energy is entrained towards the turbines, and the power production shows an upward trend towards the end of the wind farm, see Figure 3(a). In essence, when the low-level jet is below the turbine rotor swept area, the momentum deficit by the turbines in combination with the negative shear creates a significant positive turbulent flux from the low-level jet towards the turbines.

Essentially, we identified two main scenarios in which wake recovery is enhanced due to the entrainment from energy from the jet. For continuous production of turbulence, $\langle\overline{u'w'}\rangle(\partial\bar{u}/\partial z)$ should be negative, where $\partial\bar{u}/\partial z$ is the wind shear. Therefore, in the presence of positive shear ($\partial\bar{u}/\partial z$ is positive), $\langle\overline{u'w'}\rangle$ should be negative to produce turbulence. However, when the shear is negative ($\partial\bar{u}/\partial z$ is negative), $\langle\overline{u'w'}\rangle$ should be positive to sustain turbulence. The velocity deficit in the turbine wakes tends to create a positive entrainment flux below the hub height and a negative entrainment flux above the hub height. This leads to the following two scenarios:

1. When the low-level jet is above the turbine rotor swept area ($z_{jet} > z_h$), the low-level jet energy is pulled towards the turbines due to the momentum deficit created by the turbines. This creates a significant downward entrainment flux that benefits the production of turbines further downstream in the wind farm. In this case, $\langle\overline{u'w'}\rangle$ is negative, and the horizontally averaged $\partial\bar{u}/\partial z$ is positive, and this creates a net downward vertical flux towards the turbines.
2. When the low-level jet is below the turbine rotor swept area ($z_{jet} < z_h$), the high momentum low-level jet with the positive entrainment flux from below aid power production. The turbines extract the low-level jet energy transported by the positive entrainment fluxes. In this case, $\langle\overline{u'w'}\rangle$ is positive and the horizontally averaged $\partial\bar{u}/\partial z$ is negative. The negative shear created by the wind turbine wakes contributes to the negative shear already present above the low-level jet, which amplifies the effect.

An energy budget analysis of the turbulent entrainment flux over a control volume surrounding each turbine row provides the conclusive evidence of the phenomena described above. Figure 3(c) shows the development of the turbulent kinetic energy flux as a function of the positions in the wind farm. The solid lines represent the energy flux entering the control volume below, and the dashed lines represent the energy flux entering the control volume above. We can observe that the turbines operate in a positive shear region when the low-level jet is above the turbines. The downward flux is strong, and the vertical entrainment flux is enhanced in the presence of low-level jets. However, the net entrainment is severely limited when the low-level jet is in the middle of the turbine rotor swept area, which prevents turbines downstream in the wind farm from benefiting from the jet's energy. Furthermore, the figure shows that a positive entrainment flux is created when the low-level jet is below the turbine rotor swept area, i.e., in the negative shear region of the jet. This new mechanism extracts energy from the low-level jet. Figure 3(d) shows the integrated net entrainment flux as a function of the downstream location in the wind farm. The figure clearly shows that the energy entrainment from the jet is significantly higher for the cases $z_{jet} > z_h$ and $z_{jet} < z_h$ than when $z_{jet} \approx z_h$. In essence, we discovered an interesting turbulent phenomenon wherein the high-energy wind is pulled up (or down), aiding the wind farm power production, when the low-level jets are below (or above) the turbines. This hitherto unknown effect is critical for wind resource planning in areas where low-level jets events are frequent [2]. Therefore, installing turbines with heights slightly above the low-level jet height in wake recovery is advisable. Otherwise, the wake recovery of the downstream turbines is severely affected, increasing the so-called `wake losses.'

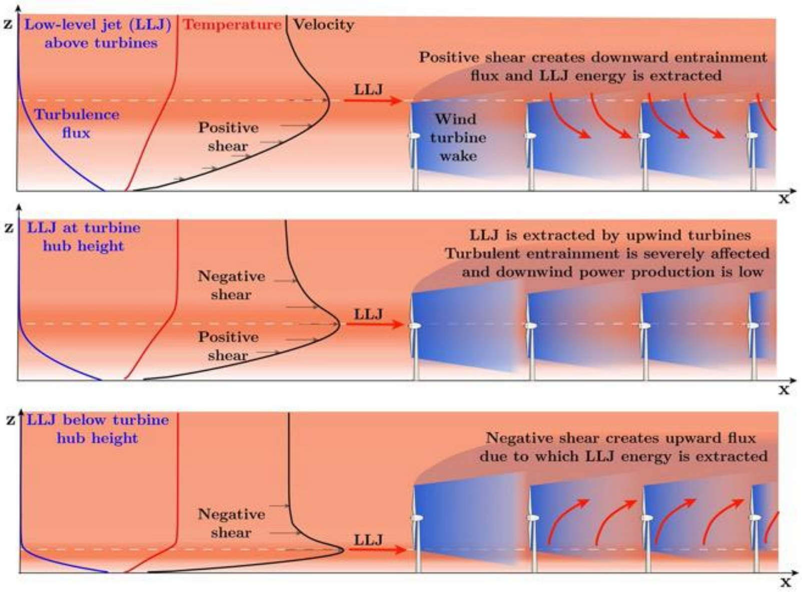


Figure 2: Schematic showing the interaction between a low-level jet, i.e. the wind maximum in the lowest regions of the atmosphere, with a wind farm. Three scenarios are considered. In the upper figure, the jet is above the wind turbines. In this case, a downward vertical kinetic energy flux is created, which benefits the production of the turbines further downstream in the wind farm. In the middle figure, the low-level jet is roughly at the turbine height, and in this case, only the leading rows can harvest the jet's energy. The lower figure shows a jet below the turbine. In this case, the wake generated turbulence created a vertical kinetic energy flux that is directed upwards, and this again benefits the turbines downstream. Figure adapted from [2].

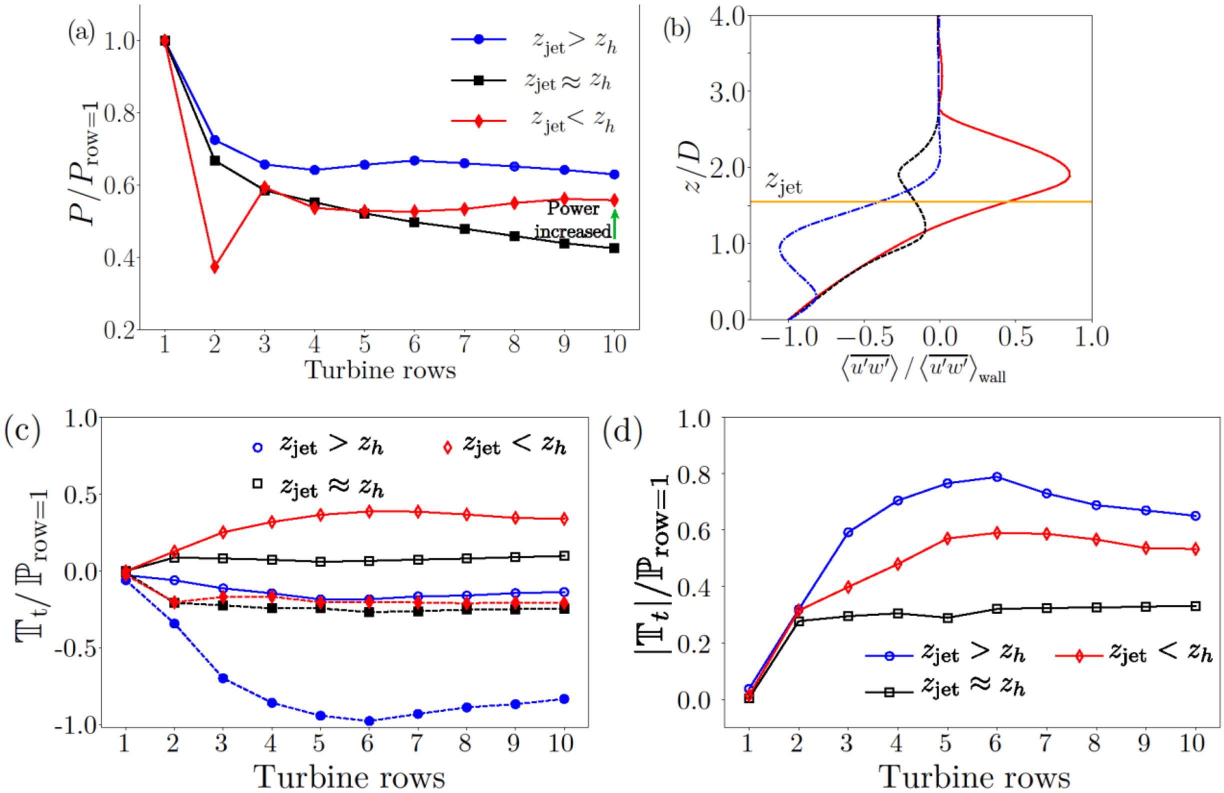


Figure 3: (a) The row-averaged power normalized with the power production of the first row. (b) Planar-averaged streamwise vertical momentum flux versus height. (c) Turbulent kinetic energy flux as a function of the downstream location. The solid lines indicate the energy that enters the control volume around the respective turbines from below. The dashed lines indicate the energy that comes from above. (d) Net energy flux as a function of downstream position. Figures adapted from Gadde and Stevens (2021) [2].

Effect of low-level jet on aerodynamic blade loading

So far, we focused our discussion on the large-scale interaction between the wind farm and the low-level jet. However, it is also crucial to understand how the low-level jet affects the fatigue loading experienced by the turbines. To study this we analyzed the influence of the low-level jet height on the external aerodynamic blade loading of the turbine blades for the three scenarios considered above [3]. Figure 4 shows that the flow structures created behind a stand-alone turbine are significantly affected by the low-level jet height. The wakes are mostly turbulent with weak tip and root vortices when the low-level jets are above the turbine. However, with decreasing low-level jet height, the strength and stability of the root and tip vortices increase, and this can cause higher fatigue loads on the downstream turbines. Overall, our study shows that the turbines are subjected to higher aerodynamic loads in the presence of low-level jets than in typical scenarios without low-level jets. This means that the influence of low-level jets needs to be considered in the life-cycle analysis of wind turbines. These new insights can help to design wind turbines and wind farms.

Outlook

In our recent studies [2, 3, 11] we focused on the influence of low-level jets on wind farm power production. However, in more realistic situations, the atmospheric boundary layer contains mesoscale advection tendencies, and baroclinicity [12] due to complex terrain or land-to-sea transitions. While there have been preliminary studies on the effect of baroclinicity on wind farm performance [13], the combined effect of baroclinicity and thermal stability is largely unexplored. For example, low-level jets are enhanced by baroclinicity and differential heating [14] and the influence of this phenomena on wind farm performance needs further investigation. Future studies should focus on incorporating atmospheric mesoscale effects in studies of wind farms, which will be crucial in designing futuristic wind farms.

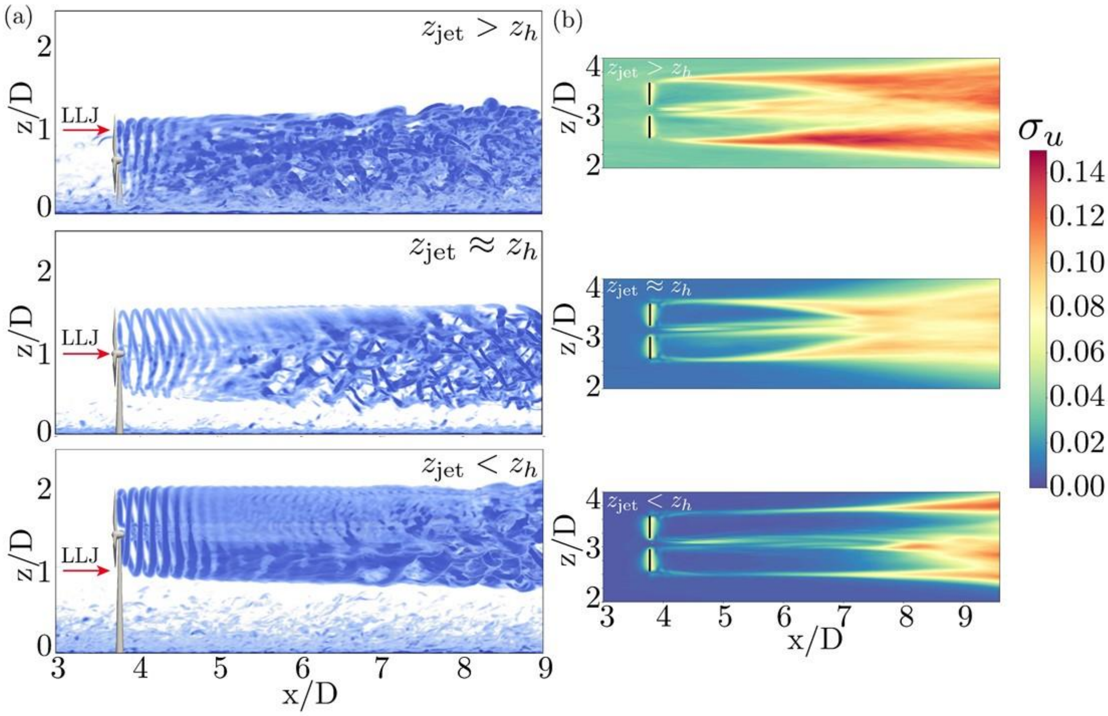


Figure 4: (a) Visualization of vorticity magnitude Ω_{mag} for three different scenarios. From top to bottom: low level jet is above, in the middle, or below the rotor swept area, and (b) corresponding streamwise turbulence intensity $\sigma_u = u'/u_{hub}$ at hub-height, where u' represents the fluctuating streamwise velocity. Figures adapted from Gadde et al. [3].

References

1. Gadde, S.N. (2021). Large Eddy Simulations of Stratified Atmospheric Boundary Layers and Wind Farms. PhD Thesis, University of Twente, doi:10.3990/1.9789036551809.
2. Gadde, S.N., Stevens, R.J.A.M. (2021) Effect of turbine-height on wind farm performance in the presence of a low-level jet. *J. Renew. Sustain. Energy* **13**, 013305, doi:10.1063/5.0026232.
3. Gadde, S.N., Liu, L., Stevens, R.J.A.M. (2021). Effect of low-level jet on turbine aerodynamic blade loading using large-eddy simulations. *J. Phys. Conf. Ser.* **1934**, 012001 (2021), doi:10.1088/1742-6596/1934/1/012001.
4. Stevens, R.J.A.M., Meneveau, C. (2017). Flow structure and turbulence in wind farms. *Annu. Rev. Fluid Mech.* **49**, 311 (2017), doi:10.1146/annurev-_uid-010816-060206.
5. Calaf, M., Meneveau, C., Meyers, J. (2010). Large eddy simulations of fully developed wind-turbine array boundary. *Phys. Fluids* **22**, 015110, doi:10.1063/1.3291077.

6. Blackadar, A.K. (1957). Boundary layer wind maxima and their significance for the growth of nocturnal inversions, *Bull. Am. Meteorol. Soc.* **38**, 283 (1957), doi:10.1175/1520-0477-38.5.283.
7. Prabha, T.V., Goswami, B.N., Murthy, B.S., Kulkarni, J.R. (2011). Nocturnal low-level jet and 'atmospheric streams' over the rain shadow region of Indian Western Ghats, *Q. J. R. Meteorol. Soc*. **137**, 1273, doi:10.1002/qj.818.
8. Kalverla, P.C., Duncan Jr. J.B., Steeneveld, G.-J., Holtslag, A.A. (2019). Low-level jets over the North Sea based on ERA5 and observations: together they do better, *Wind Energy Science* **4**, 193, doi:10.5194/wes-4-193-2019.
9. Wagner, D., Steinfeld, G., Witha, B., Wurps, H., Reuder, J. (2019). Low level jets over the Southern North Sea. *Meteorol. Zeitschrift* **28**, 389, doi:10.1127/metz/2019/0948.
10. Gadde, S.N., Stieren, A., Stevens, R.J.A.M. (2021). Large-eddy simulations of stratified atmospheric boundary layers: Comparison of different subgrid models. *Boundary-Layer Meteorol.* **178**, 363–382, doi:10.1007/s10546-020-00570-5.
11. Gadde S.N., Stevens, R.J.A.M. (2021). Interaction between low-level jets and wind farms in a stable atmospheric boundary layer. *Phys. Rev. Fluids* **6**, 014603, doi:10.1103/PhysRevFluids.6.014603.
12. Floors, S. E. G. R. , Peña, A. (2015). The effect of baroclinicity on the wind in the planetary boundary layer. *Q. J. R. Meteorol. Soc.* **141**, 619, doi:10.1002/qj.2386.
13. Pedersen, J.G. (2013). Large-eddy simulation of the atmospheric boundary layer: Influence of unsteady forcing, baroclinicity, inversion strength and stability on the wind profile. PhD Thesis, DTU Wind Energy, ISBN:978-87-92896-70-4.
14. Gebauer, J.G., Shapiro, A. (2019). Clarifying the baroclinic contribution to the Great Plains low-level jet frequency maximum. *Mon. Weather Rev.* **147**, 3481, doi:10.1175/MWR-D-19-0024.1.

Interaction between shock waves and supersonic boundary layers over forward-facing and backward-facing steps

Weibo Hu, Bas van Oudheusden, Stefan Hickel[1]

[1]Aerodynamics Group, Faculty of Aerospace Engineering, Delft University of Technology

Introduction

Shock wave / boundary layer interactions (SWBLI) have been an active research topic in the aerospace community over the past decades. SWBLI form a dynamic system of mutually interacting boundary layers, free shear layers, shock waves, an expansion fan, and a separation bubble. It is widely reported that this dynamic system features a significant low-frequency unsteadiness, whose frequency is typically two orders lower than the characteristic frequency of the incoming turbulent boundary layer, and that may detrimentally affect the structural integrity of aircraft and rockets. Several theories have been proposed to explain the origin of this unsteadiness. These theories can be classified based on whether they consider upstream and downstream mechanisms as the main driver (see [1] for an overview).

The validity and general applicability of these mechanisms is a scientific controversary. In literature, there is no consensus even for extensively studied canonical incident shock and compression ramp cases. In addition, these so-called upstream and downstream mechanisms may not apply in the same manner to more complex geometries, such as SWBLI over forward-facing steps (FFS) or backward-facing steps (BFS), in view of their different flow topology. To enhance our understanding of the underlying physical mechanisms, we performed extensive numerical studies on the low-frequency unsteady dynamics of SWBLI in laminar and turbulent supersonic FFS/BFS cases [2, 3, 4, 5]; the work as described here was part of the PhD thesis by the first author [6].

Flow configuration and computational method

We consider an open FFS and BFS (i.e., no upper wall) with the same supersonic freestream conditions at $Ma = 1.7$ and $Re = 1.37 \times 10^7$ m^{-1}. The incoming boundary layer is either a fully laminar flow, a laminar flow superimposed with linear instability modes with different amplitude, or a fully turbulent boundary layer. Common to all cases of this study is Re_δ = 13718, based on the 99% velocity boundary layer thickness δ_0 at the inlet.

We employ the large eddy simulation (LES) method of Hickel et al. [7] and the finite volume solver INCA (www.inca-cfd.com). INCA is a mature solver that has been extensively applied and validated for various compressible flow cases, including SWBLI at compression ramps and SWBLI on planar walls. INCA uses adaptive block-Cartesian meshes with hanging nodes at block interfaces to accommodate adaptive mesh refinement (AMR). For the current simulations, the mesh resolution is uniform within each block except in blocks near walls, where hyperbolic grid stretching is used, such that the mesh resolution is $y^+ < 1$ in the wall-normal direction. For fully turbulent cases, inflow conditions are provided by the digital filter technique of Klein et al. [8]. The step and wall are modelled as no-slip adiabatic surfaces. Non-reflecting freestream boundary conditions based on Riemann invariants are used and periodic boundary conditions are imposed in the spanwise direction. For more details on the setup, see [2, 3, 4, 5, 6].

Selected results

Mean and instantaneous flow

The mean flow field in Figure 1 provides an overview of the main flow topology, including a centred Prandtl-Meyer expansion from the step edge, the separation bubble and shock waves. For the FFS case, the boundary layer separates at x/δ_0 = 15.9 and reattaches on the step wall at y/δ_0 = 2.1. The separation region is encompassed by two shocks, i.e., the separation and reattachment shock. In contrast, the flow over the BFS separates at the step edge and reattaches at x/δ_0 = 8.9, and there is only a reattachment shock that originates from the compression waves near the reattachment. Figure 2 shows instantaneous vortical structures for a transitional case with laminar inflow and for a fully turbulent case. From the instantaneous flow fields in Figure 3, two main types of unsteady behaviour are observed: a global motion involving the breathing of the separation bubble and the flapping of the shock and a relatively localized unsteadiness related to the shedding of vortices along the shear layer.

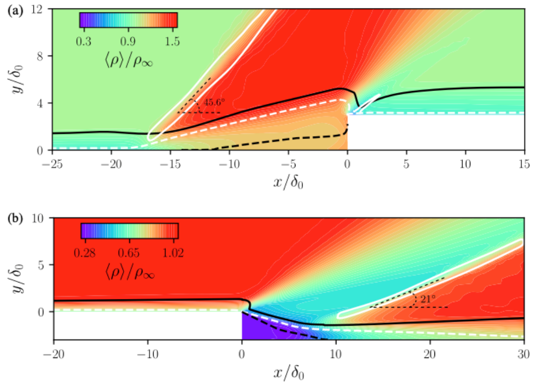


Figure 1: Density contours of the time- and spanwise- averaged flow field for two turbulent cases: (a) FFS and (b) BFS (the flow is from left to right). The black dashed line represents $u = 0$ and visualizes the separation bubble, the black solid line shows the boundary layer edge, and the white dashed line is the sonic line ($Ma = 1$). The separation shock (FFS) and reattachment shocks (FFS and BFS) are visualized by white solid lines, which enclose regions of large pressure gradients.

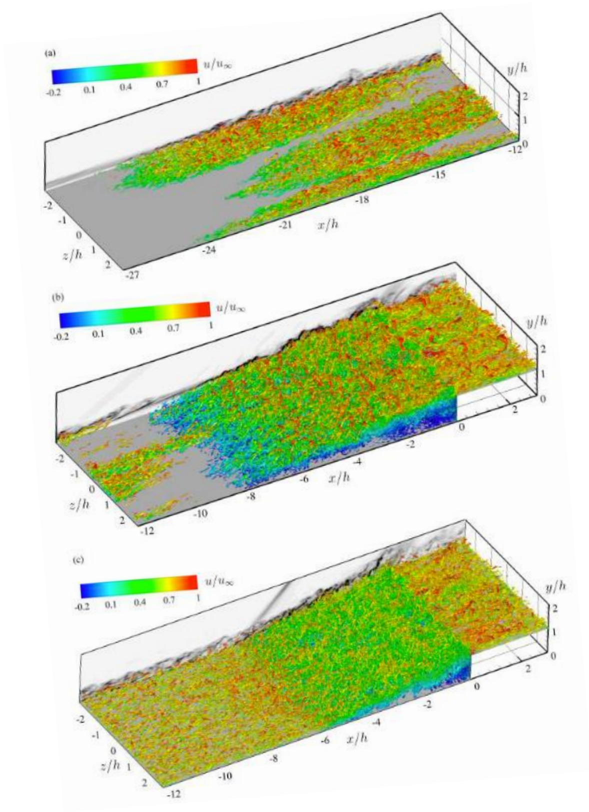


Figure 2: Instantaneous vortical structures for (a, b) a transitional case with laminar inflow and (c) a fully turbulent FFS case.

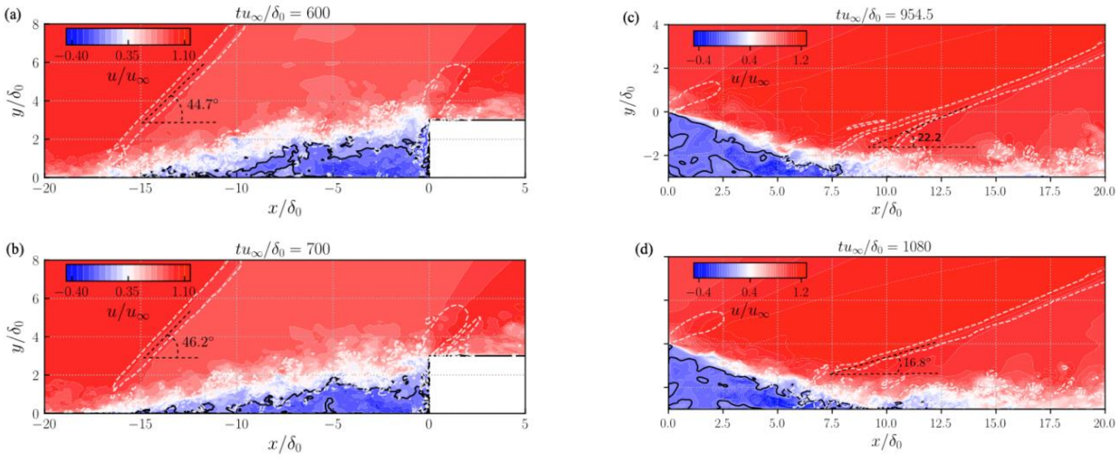


Figure 3: Contours of the instantaneous streamwise velocity for two turbulent cases. (a) FFS: $t\,u_\infty/\delta_0$ = 600, (b) FFS: $t\,u_\infty/\delta_0$ = 700, (c) BFS: $t\,u_\infty/\delta_0$ = 955, and (d) BFS: $t\,u_\infty/\delta_0$ = 1080. Black solid lines denote the isolines of $u = 0$ and white dashed lines signify the isolines of $|\nabla p|\delta_0/p_\infty$ = 0.4.

Spectral analysis

The frequency weighted power spectral density (FWPSD) of the wall pressure in Figure 4 shows a broadband frequency spectrum in these two unsteady SWBLI systems. Two distinct low-frequency ranges are identified from the spectral map, i.e., a lower one ($0.005 \le f\ \delta_0/u_\infty \le 0.05$ for the FFS and $0.06 \le f\ \delta_0/u_\infty \le 0.08$ for the BFS) and a medium one ($0.06 \le f\ \delta_0/u_\infty \le 0.25$ for the FFS and $0.1 \le f\ \delta_0/u_\infty \le 0.3$ for the BFS). The energetic low-frequency range is in good agreement with results for incident shock and ramp SWBLI cases (Priebe and Martin [9]; Pasquariello et al. [10]), which is widely reported to be approximately two orders lower than the characteristic frequency of the incoming energetic turbulent scales u_∞/δ_0.

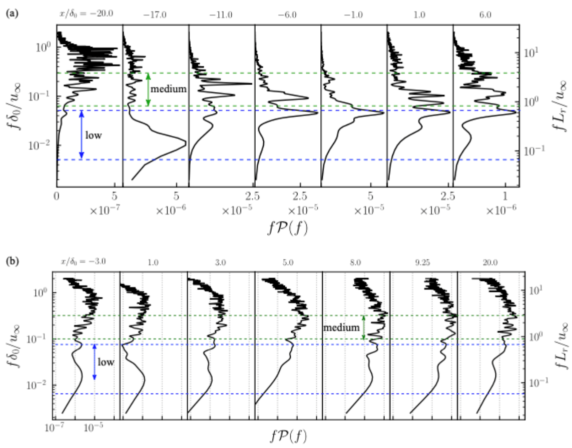


Figure 4: Frequency weighted power spectral density of the wall pressure with the streamwise distance for two fully turbulent (a) FFS and (b) BFS cases.

To further scrutinize the physical links between various dynamics and frequencies of the unsteady motions, several typical flow parameters, including the reattachment location, separation bubble volume and shock angle, were extracted from the present results. The FWPSD of the reattachment location for turbulent FFS and BFS cases has a medium frequency peak located around 0.1 u_∞/δ_0, which suggests that the reattached shear-layer flow is related to the medium-frequency unsteadiness in SWBLI. The intermediate frequencies are associated with the passage of shedding vortices formed in the separated shear layer. The low-frequency unsteadiness is associated with the breathing of the separation bubble and flapping motion of the shock wave. The PSD for the volume of the separation bubble shows a single spectral peak at 0.2 u_∞/δ_0 for both FFS and BFS cases. Similarly, the motions of the shock angle feature a dominant low frequency.

Dynamic mode decomposition

To further analyze the flow features that contribute to the unsteady dynamics at different frequencies, a dynamic mode decomposition (DMD) of the three- dimensional FFS and BFS flow fields was carried out based on snapshots for a spatial subdomain. For both the FFS and BFS cases, three branches of modes with different frequencies are identified from the DMD frequency spectrum.

For the FFS case, we selected one representative mode from each branch, based on the growth rate and magnitudes of the modes, marked as mode ϕ_1, ϕ_2, and ϕ_3. For the low-frequency mode ϕ_1 ($f\delta_0/u_\infty$ = 0.013), iso-surfaces of the modal pressure fluctuations show significant structures around the separation shock and compression waves caused by the reattachment. The variation of the large structures with the phase angles indicates the unsteady motions of the shock both in the spanwise and streamwise direction. Considering the modal velocity fluctuations for the same mode in Figure 5, iso-surfaces of u'/u_∞= 0.2 visualize the counter-rotating streamwise Görtler vortices, distributed from the free shear layer to downstream of the reattachment point. In addition, we also determined the variation of the Görtler number G_t along the streamlines, and found that values larger than the critical value (G_t= 0.6) occur near the separation and reattachment region, suggesting a strong Görtler instability in the interaction region. These observations suggest that the low-frequency flapping motions of the shock are related to streamwise-elongated Görtler vortices in the present case. These unsteady longitudinal vortices are usually induced by the centrifugal forces from the strong curvature of the streamlines in the separation region, which provides a potential physical mechanism for the low-frequency unsteadiness in this FFS case.

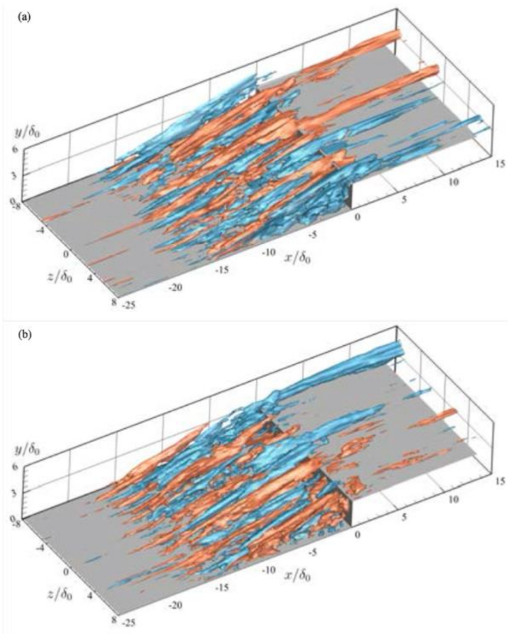


Figure 5: Iso-surfaces of the modal streamwise velocity fluctuations from DMD mode ϕ_1 with phase angle (a) $\theta\ =\ 0$ and (b) $\theta\ =\ 3\pi/4$ for a turbulent FFS case (orange: u'/u_∞= 0.2, blue: u'/u_∞= −0.2).

For the medium-frequency mode ϕ_2, the pressure iso-surfaces show high levels of modal fluctuations along the free shear layer. Positive and negative fluctuations are alternating in the streamwise direction, which represents the propagation of acoustic waves. The radiation of the acoustic waves is in agreement with the results from a global linear stability analysis of an impinging shock case in laminar regime. Regarding the modal fluctuations of the streamwise velocity, shown in Figure 6, Λ-shaped structures are observed in the free shear layer and alternate along both the spanwise and streamwise directions. Based on these observations, we believe this mode represents the convection of the shear layer vortices. Pasquariello et al. [10] addressed similar conclusions from the two-dimensional DMD analysis of a flat-plate, incident shock case at higher Reynolds number.

Similarly, for the BFS case, the flapping of the shock wave and counter-rotating Görtler vortices are responsible for the low-frequency motions, while the shedding vortices along the shear layer and the induced Mach-like waves contribute to the medium-frequency behaviour.

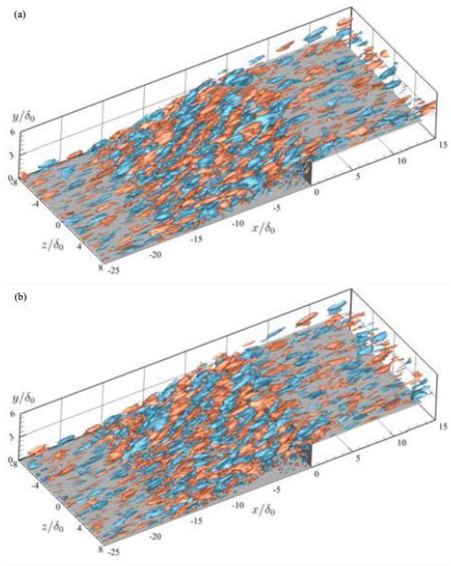


Figure 6: Iso-surfaces of the modal streamwise velocity fluctuations from DMD mode ϕ_2, with phase angle (a) $\theta = 0$ and (b) $\theta = 3\pi/4$ for a turbulent FFS case (orange: u'/u_∞ = 0.2, blue: u'/u_∞ = −0.2).

Conclusions

The low-frequency unsteady dynamics of SWBLI over forward-facing and backward-facing steps was investigated at $Ma = 1.7$ and Re_δ = 13718 using a well-resolved LES. The main flow topology of the SWBLI region contains shock waves (a separation shock in the FFS case and reattachment shocks in both cases), a main separation bubble and a centred Prandtl-Meyer expansion fan. The instantaneous visualizations indicate that the unsteady behaviour involves vortex shedding in the separated shear layer, breathing of the separation bubble, shock wrinkling, and flapping shock motions. From the spectral analysis, we observe broadband low-frequency motions in the interaction region, which can be classified into two branches based on their dominant frequency. Medium-frequency contents are associated with shear layer vortices, and the lower frequency dynamics is associated with the unsteady separation region.

Three-dimensional DMD analysis was applied to identify the individual single-frequency modes that have the largest contributions to the observed unsteady behaviour. Our results show a statistical link between the shock motions (shown by pressure fluctuations) and unsteady Görtler-like vortices (shown by the streamwise velocity fluctuations) along the shear layer. The medium-frequency modes represent the shedding of shear-layer vortices and the radiation of the induced Mach waves.

Based on the above observations and discussion, we believe that the physical mechanism of the low-frequency unsteadiness of SWBLI in FFS/BFS cases is similar as for other canonical SWBLI cases, i.e., unsteady Görtler-like vortices in the shear layer act as an unsteady forcing that is required to drive and sustain the low-frequency motions of shocks and separation bubble. A more detailed analysis is provided by Hu et al. [2, 3, 4, 5, 6].

Acknowledgments

This work was carried out on the Dutch national e-infrastructure with the support of SURF Cooperative and was partly financed by the Dutch Research Council (NWO) project number 2019.045.

References

1. Clemens, N. T., Narayanaswamy, V. (2014). Low-frequency unsteadiness of shock wave/turbulent boundary layer interactions. *Annual Review of Fluid Mechanics* **46**(1):469–492.
2. Hu, W., Hickel, S., van Oudheusden, B. (2019). Dynamics of a supersonic transitional flow over a backward-facing step. *Physical Review Fluids* **4**(10):103904.
3. Hu, W., Hickel, S., van Oudheusden, B. (2020). Influence of upstream disturbances on the primary and secondary instabilities in a supersonic separated flow over a backward-facing step. *Physics of Fluids* **32**(5): 56102.
4. Hu, W., Hickel, S., van Oudheusden, B. (2021). Low-frequency unsteadiness mechanisms in shock wave/turbulent boundary layer interactions over a backward-facing step. *Journal of Fluid Mechanics* **915**: A107.
5. Hu, W., Hickel, S., van Oudheusden, B. (2022). Unsteady mechanisms in shock wave boundary layer interactions over a forward-facing step. *Journal of Fluid Mechanics*.

6. Hu., W. (2020). Dynamics of a supersonic flow over a backward/forward-facing step. PhD Thesis, Delft University of Technology.
7. Hickel, S., Egerer, C. P. & Larsson, J. (2014). Subgrid-scale modeling for implicit large eddy simulation of compressible flows and shock-turbulence interaction. *Physics of Fluids* **26**(10): 106101.
8. Klein, M., Sadiki, A., Janicka, J. (2003). A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations. *Journal of Computational Physics* **186**(2):652–665.
9. Priebe, S., Martin, M. P. (2012). Low-frequency unsteadiness in shock wave–turbulent boundary layer interaction. *Journal of Fluid Mechanics* **699**: 1– 49.
10. Pasquariello, V., Hickel, S., Adams, N. A. (2017). Unsteady effects of strong shock-wave/boundary- layer interaction at high Reynolds number. *Journal of Fluid Mechanics* **823**: 617–657.

High temperature turbulence
coupling of radiative and convective heat transfer in turbulent flows

Simone Silvestri, René Pecnik, Dirk Roekaerts[1]

[1] Department of Process and Energy, Delft University of Technology

This research (which was part of the PhD project by the first author) was focused on the role of radiation in high temperature participating media and its influence on turbulent heat transfer. One of my research goals was to design a method to allow the simulation of fully coupled radiative and convective transfer resolving all thermal scales. To overcome the challenging computational requirements involved, we developed a custom innovative CUDA implementation of a Quasi-Monte Carlo radiative solver and coupled it to an efficient CPU Direct Numerical Simulation solver. This massively parallel Monte Carlo solver is the fastest to-date and ran on the largest heterogeneous HPC clusters in Europe, including Piz-Daint in Switzerland and the new Marconi-100 in Italy. With the help of this code, we were able to formulate a theory that describes the impact of Turbulence-Radiation-Interactions (TRI) on the overall heat transfer process in high temperature turbulent flows. The theoretical analysis, based on the high-fidelity DNS Monte Carlo data, unveiled the mechanisms governing the temperature field. This was the basis for the development of a low-order approximation that enabled the closure for the radiation-turbulence coupling. This model was proven to predict exceptionally the temperature field for all investigated systems, where standard models fail.

Research objectives

The continuous demand to increase the efficiency of energy conversion systems and the productivity of process plants forces engineers and scientists to use fluids at increasingly higher pressures and temperatures. For instance, to increase the thermal efficiency of power plants, engineers are currently developing a thermodynamic power cycle that operates with carbon dioxide at pressures and temperatures high enough to exceed the thermodynamic critical point. Another example where pressures and temperatures of fluids continuously increase is in the development of more powerful rocket engines.

At these high temperatures, radiation is the most important heat transfer mechanism. In addition, gases like CO_2, H_2O, or CH_4 strongly “participate” in radiative heat transfer, i.e. they emit and absorb thermal radiation at large rates. Therefore, being able to predict the impact of radiative heat transfer on the overall heat transfer process as well as on the flow behaviour is of vital importance to successfully realize these new engineering applications.

In turbulent flows of highly participating fluids, the presence of temperature fluctuations induces fluctuations in the emitted thermal radiation. The radiated thermal energy undergoes re-absorption within the fluid, causing a long range energy transfer, which acts as a damping mechanism for temperature fluctuations. This damping mechanism affects turbulence and mixing levels in case of strong density gradients. A vivid example of this mechanism (taken from our detailed simulations) is presented in Figure 1. These so called turbulence-radiation interactions have only recently been investigated using expensive numerical simulations, made possible by the increased computing power, necessary to handle the problem.

On the other hand, a comprehensive knowledge of the underlying physics is still lacking for significantly participating media (optically intermediate and thick). The reason is that for flows in general geometries with highly participating non-gray fluids, analytical representations are inaccessible, experimental investigations are difficult, and detailed numerical simulations are “simply” computationally unfeasible with conventional codes. The combination of non-gray effects (highly variable absorption spectrum) and large optical depths results in a total disruption of turbulent convection and an unexpected redistribution of the energy within the fluid. This completely changes the well-known patterns of heat transfer and variable property turbulence.

In conclusion, the effect of radiation on turbulent convection in optically dense, non-gray fluids is still unknown. This results in the complete lack of models able to predict turbulence-radiation interactions through the whole scale of motion and optical thickness range. For this reason, several important quantities, such as the mean temperature and the mean radiative fields, cannot be predicted accurately. As such the research objectives of this project were the following:

1. Develop an optimized tool to allow the full coupling of radiative heat transfer and turbulence.

2. Perform high-fidelity Direct Numerical Simulations (DNS) that give access to a complete description of the turbulence-radiation interactions.
3. Provide a clear characterization of the coupling between turbulence and radiation, for different optical thicknesses, variable and spectral radiative properties and including the effects of anisotropic turbulent structures.
4. Reconcile the effect of radiation on temperature by providing scaling relations, which are able to describe the turbulence-radiation coupling independently of the above specified parameters.
5. Develop a simplified model, which allows the calculation of high-temperature turbulent participating flows without the necessity to perform heavy duty simulations.

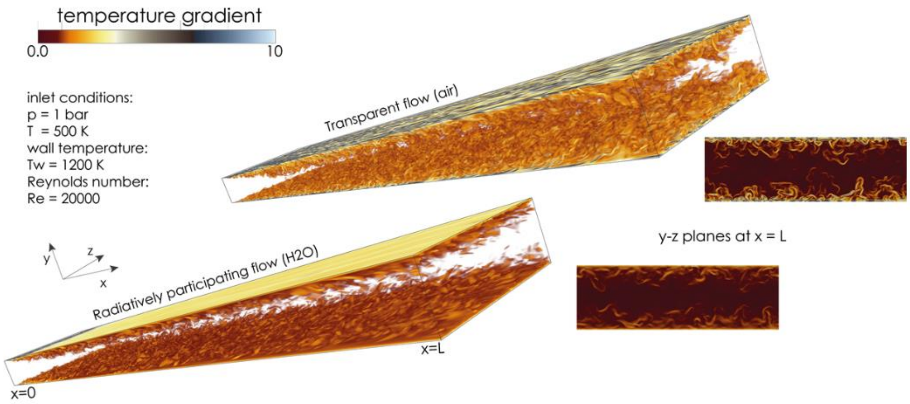


Figure 1: Thermally developing channel flow for two different media. The only difference between the transparent flow (air, top) and the participating flow (H_2O, bottom) is the presence of emission and absorption from the fluid. It is clear that the temperature field is completely disrupted as the gradient decreases significantly. This in turn leads to a great reduction in turbulence intensity for all velocity components. These results were generated with 10000 rays/cell on a mesh with 750 million cells.

Research route and methodology

To study the interaction between turbulence and radiation in high temperature participating flows, the solution of radiative heat transfer for all turbulent length scales had to be made available. Unfortunately, the Radiative Transfer Equation (RTE) is six-dimensional (including directional and wavelength dependency) and a one-to-one coupling of a radiative solver with a DNS solver (which solves the complete range of turbulent scales) had never been attempted due to the exorbitant computational requirements. To circumvent the problem, we developed an innovative method that exploits new generations of High Performance Computing (HPC) centres, by solving the spectral RTE on GPUs with a photon Monte Carlo (MC) method paralleled to a DNS flow solver running on CPUs (schematic shown in Figure 2). The GPU Monte Carlo solver [2] required extensive optimization, both in terms of computer science and exploitation of the physics of the problem to speed up the numerical methods, in order to render the solution feasible. The final version of the code had a staggering 1000 × speedup compared to a usual CPU code and allowed first-of-its-kind high-fidelity simulations of one-to-one coupled turbulence and radiative heat transfer.

With this new solver we were able to perform simulations of high temperature, fully developed channel flow with radiative heat transfer. The setup, consisted of a statistically one dimensional channel flow, periodic in the streamwise and spanwise directions and bounded by a hot and a cold isothermal wall. The participating turbulent flow was simultaneously heated by the hot wall and cooled by the cold wall and capable of absorbing radiation from the hot wall and emitting towards the cold wall. In addition, fluid-to-fluid radiative heat transfer modified the redistribution of the energy within the channel. The thermodynamic and radiative parameters of the turbulent flow (such as the density, optical thickness, and spectral variability) were varied to identify a scaling for the radiative-turbulence coupling.

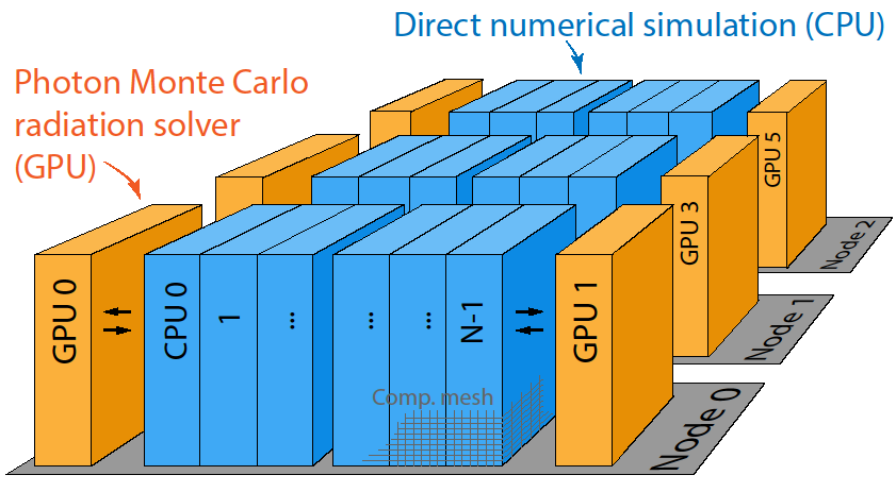


Figure 2: Schematic of the implemented heterogeneous solution method, exemplifying a system with 3 nodes composed of 6 CPU cores and 2 GPUs each.

Main results and achievements

Due to radiation, new terms appear in the energy equation, which are dependent on the "optical thickness" (τ) of the flow, or of the ability of the medium to absorb incoming radiation. We used this last parameter to categorize the effects of radiation on the turbulent heat transfer process by highlighting the contrasting effects of radiative emission and absorption.

The first step was to identify quantities that would encompass the physics of turbulence-radiation interactions. We found one of these quantities to be the "radiative dissipation" ($R_\theta = \langle Q_R' T' \rangle$) , which mathematically represents the correlation of the temperature field fluctuations with the radiative field fluctuations. This term includes both aspects of Turbulence-Radiation Interactions (TRI): (1) the development of a fluctuating radiative field by action of turbulence, and (2) the destruction of the thermal fluctuation field caused by the presence of radiative heat transfer.

It is well known that the radiative heat source has a non-monotonic dependency on the absorption coefficient due to the non-local nature of radiation. As such, radiative heat transfer can be divided into three regimes: optically thin $\tau \ll 1$, optically intermediate $\tau \approx 1$, and optically thick $\tau \gg 1$, where the optical thickness can be seen as the ratio of geometrical to radiative length scales. We extended this categorization to TRI by investigating the differences in the radiative dissipation for optically thin, intermediate and thick turbulent flows. It was found that the optical thickness provided much information on the TRI mechanism as the three different regimes were different not only quantitatively but also in terms of behaviour [1]. On the other hand, we unfortunately discovered that the optical thickness alone was not the correct parameter to predict the effects of TRI. In Figure 3 (left) the radiative dissipation integrated over the channel height for many different simulations (legend in the central figure) is shown versus the optical thickness of the channel. A clear scaling pattern based on optical thickness is not evident.

After careful analysis, we identified the cause of the TRI mechanism in the fluctuation of the incident radiation that is the radiative energy absorbed by the flow per unit area. If normalized correctly, this quantity scales monotonically with a modified optical thickness (τ_g), which was introduced to account for the variability of the absorption spectra, and is a function of the absorption coefficient and the turbulent length scale. Normalized absorption fluctuations are shown as function of τ_g in the central plot of Figure 3. Note that the gray line in Figure 3 does not stem from fitting of the data but is mathematically derived. $f(\tau_g) \sim (\tau_g / \langle \omega_c \rangle) \text{atan}(\langle \omega_c \rangle / \tau_g)$ and ω_c represent the characteristic length scales of the temperature field [3] (here averaged among the different simulations). This analysis allowed to construct a mathematical model, based on a scaling parameter, which is able to predict radiative dissipation and, therefore, provides a framework to model the effects of TRI in high temperature turbulent flows. The aforementioned parameter can be considered as a TRI equivalent optical thickness (τ^*). As seen on the right plot in Figure 3, when using τ^*, it is possible to predict the value of the bulk radiative dissipation with a mathematically derived function of τ^*, which reads:

$$f(\tau^*) = \tau^* [1 - (\tau^* / \langle \omega_c \rangle) \text{atan}(\langle \omega_c \rangle / \tau^*)].$$

This parameter was the base for a Reynolds-Averaged Navier-Stokes (RANS) model, which could be applied as a closure to the turbulent heat flux in high temperature turbulent flows. This model has been tested thoroughly with all the DNS performed and showed exceptional results not only in terms of mean temperature field, but also in terms of second and higher order statistics [4]. An example is shown, for three representative simulations, in Figure 4. Here, DNS results for three different participating turbulent channel flows are compared to the RANS simulation using (1) the most used model in literature (0-equation model with $\alpha_t = \mu_t/0.9$, where μ_t is the eddy viscosity and α_t is the eddy diffusivity); (2) a more elaborate model, which calculates α_t from the evolution of budget equations for temperature variance and temperature dissipation; (3) the model developed in this project.

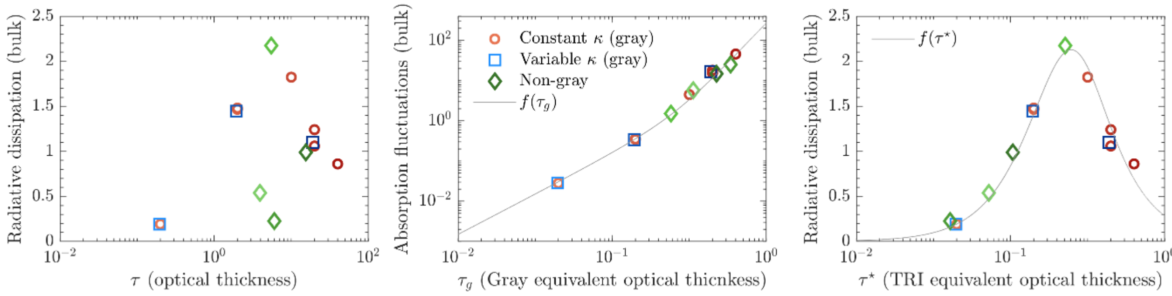


Figure 3: Bulk parameters encompassing TRI for the entire data-set of coupled radiative-turbulent DNS (a mix of gray and non-gray turbulent channel flows, each data point is a single independent simulation where the bulk value is integrated over the whole domain). Left: radiative dissipation plotted against optical thickness, centre: normalized absorption fluctuations plotted against the gray equivalent optical thickness; right: radiative dissipation plotted against the new TRI mean optical thickness.

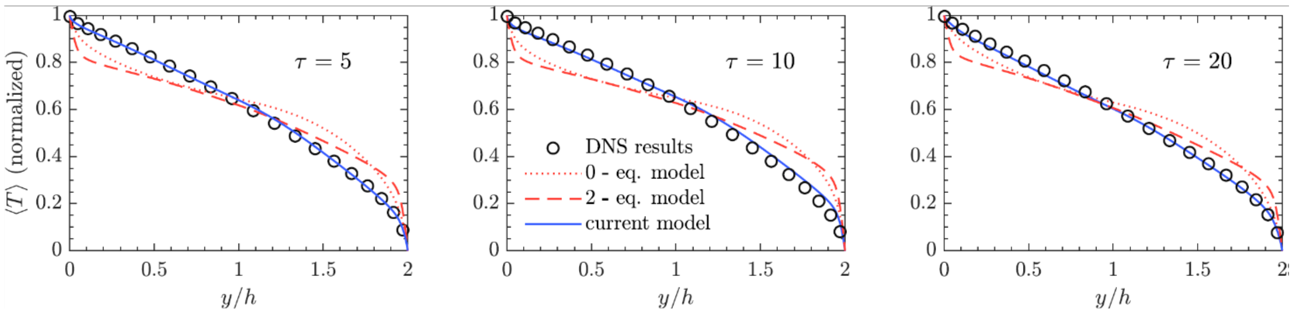


Figure 4: Averaged temperature profile (normalized) in three different fully developed turbulent channel flow simulations with radiative heat transfer. These simulations are periodic in the streamwise and spanwise direction and bounded by a hot and a cold wall in the wall-normal direction. The results are averaged over the periodic directions and shown plotted versus the wall-normal normalized coordinate. From left to right the optical thickness of the channel $\tau = \int_0^2 (I_b)^{-1} \int_0^\infty \kappa_\lambda I_{b\lambda} d\lambda dy / h$ (h is the half channel height, I_b black-body radiation and κ the absorption coefficient) changes from 5 to 10 to 20. DNS results are plotted with circles, while lines come from RANS simulations. The dotted line is a 0-equation model obtained with setting a constant turbulent Prandtl number. The dashed line shows the state-of-art of turbulent heat flux modeling (a 2-equation model calculating temperature variance and temperature dissipation). The blue line is the model developed in this project.

References

15. Silvestri S., Patel A., Roekaerts D.J.E.M, Pecnik R. (2018), Turbulence-radiation interaction in channel flow with various optical depths. *Journal of Fluid Mechanics* **834**, 359-384.
16. Silvestri S., Pecnik R. (2019), A fast GPU Monte Carlo radiative heat transfer implementation for coupling with direct numerical simulation. *Journal of Computational Physics: X*, 100032.
17. Silvestri S., Roekaerts D.J.E.M, Pecnik R. (2019), Assessing turbulence-radiation interactions in turbulent flows of non-gray media. *Journal of Quantitative Spectroscopy and Radiative Transfer* **233**, 134-148.
18. Silvestri S., Roekaerts D.J.E.M, Pecnik R. (2021), Modelling turbulent heat flux accounting for turbulence-radiation interactions. *International Journal of Heat and Fluid Flow* **89**, 108728.
19. Silvestri S., Pecnik R. (2021), Turbulence modulation in thermally expanding and contracting flows. *Journal of Fluid Mechanics*, accepted.

Microfluidic models of metastasis: in vitro approaches to study the tumor microenvironment

Jelle Sleeboom and Jaap den Toonder[1]

[1]Microsystems, Mechanical Engineering, Eindhoven University of Technology

Cancer is still one of the deadliest diseases in the world, accounting for nearly 1 in 6 of all deaths worldwide. The majority of these cancer-related deaths are not directly caused by the primary tumor, but by the formation of secondary tumors through metastasis, see Figure 1. However, our understanding of this complex process is still lacking, such that the successful treatment or prevention of metastasis has been difficult to achieve. One of the main reasons for the gaps in our knowledge is the extreme complexity of the metastatic cascade: cancer cells invade into surrounding tissue, then intravasate into the blood circulation, survive in the circulation, extravasate into another tissue, to finally proliferate and develop into a secondary tumor. During all of these steps, the cells are affected and steered by their environment, provided by other cells, extracellular matrix (ECM), soluble factors, and physical forces, see Figure 2. In the primary tumor, the collection of these cues is referred to as the tumor microenvironment (TME) [1].

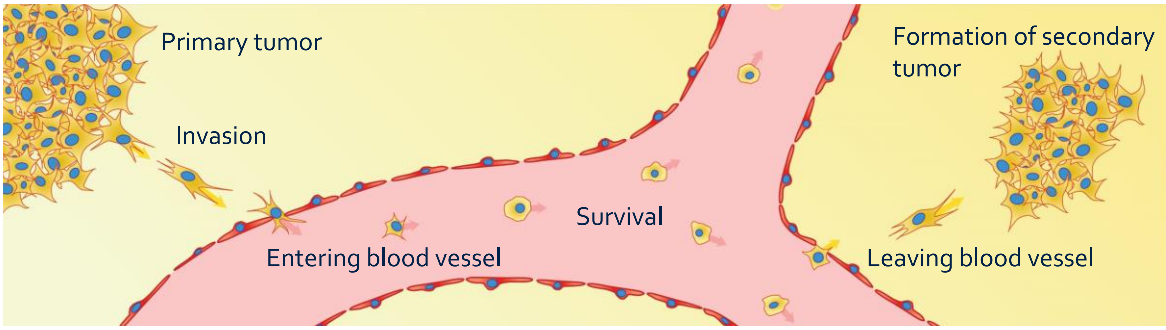


Figure 1: The process of metastasis, spreading of cancer through the body through the blood circulation, encompasses a cascade of events.

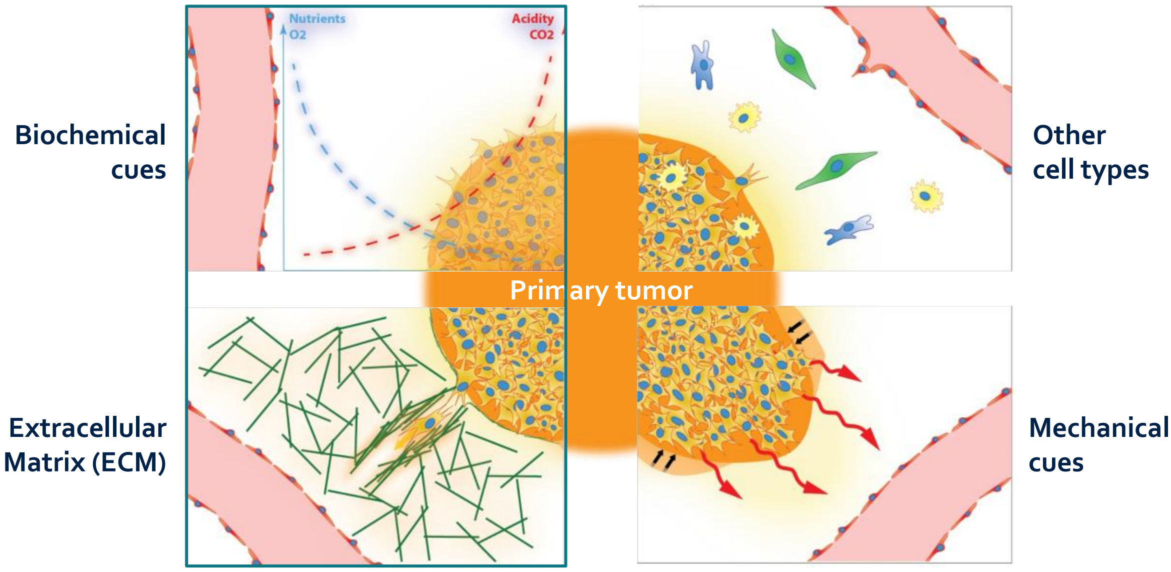


Figure 2: The first step of metastasis is invasion, which is directed by many different cues in the Tumor Microenvironment (TME); in our research, we focus on the influence of chemical cues, in particular oxygen gradient, and the nature of the Extracellular Matrix (ECM).

In our research, we have explored the factors of the TME that are relevant to the onset of metastasis, and developed methods to study them in vitro, using **Cancer-on-a-Chip (CoC) models**. These in vitro models, based on **microfluidic chips**, contain small chambers for cell culture as well as fluid channels, enabling control over local gradients, fluid flow, tissue mechanics, and composition of the local environment. These properties make CoC devices promising tools to investigate the TME. We have focused on the development of novel in vitro systems to investigate two specific factors from the TME: local oxygen gradients, and the ECM (see Figure 2).

In order to study the effect of **oxygen gradients** on cancer cell migration, we have developed a microfluidic platform that is capable of generating a controlled and stable gradient between hypoxic (<1%) and normoxic (~7%) oxygen saturation, shown in Figure 3(a) [2]. Cells are cultured in a microfluidic chamber in which the oxygen gradient is generated using a neighbouring channel, separated from the cell chamber by a thin polydimethylsiloxane (PDMS) barrier, in which a continuous flow is maintained of leaching fluid that consumes oxygen. The design of the chip was guided using numerical simulations (Figure 3(b)) and the oxygen gradient was confirmed by experiments (Figure 3(c)). Using the developed system, we have studied the migration of MDA-MB-231 breast cancer cells in this defined oxygen gradient. Figure 3(d) shows the tracks of three cells within the cell chamber, and Figure 3(e) shows many cell tracks superposed and shifted to the origin. The migration of the cancer cells was found to have a bias towards low oxygen levels, as shown in Figure 3(f). In addition, we have investigated the migration of another breast cancer cell line (MCF-7), and two glioblastoma cell lines (U87, and U251). Here we found that the U251 cells exhibit behaviour similar to the MDA-MB-231 cells. In contrast, the MCF-7 and U87 cells did not show the bias towards low oxygen, but migrated in random patterns. We also demonstrate that the platform enables staining for relevant markers, such as E-cadherin, N cadherin, and Vimentin.

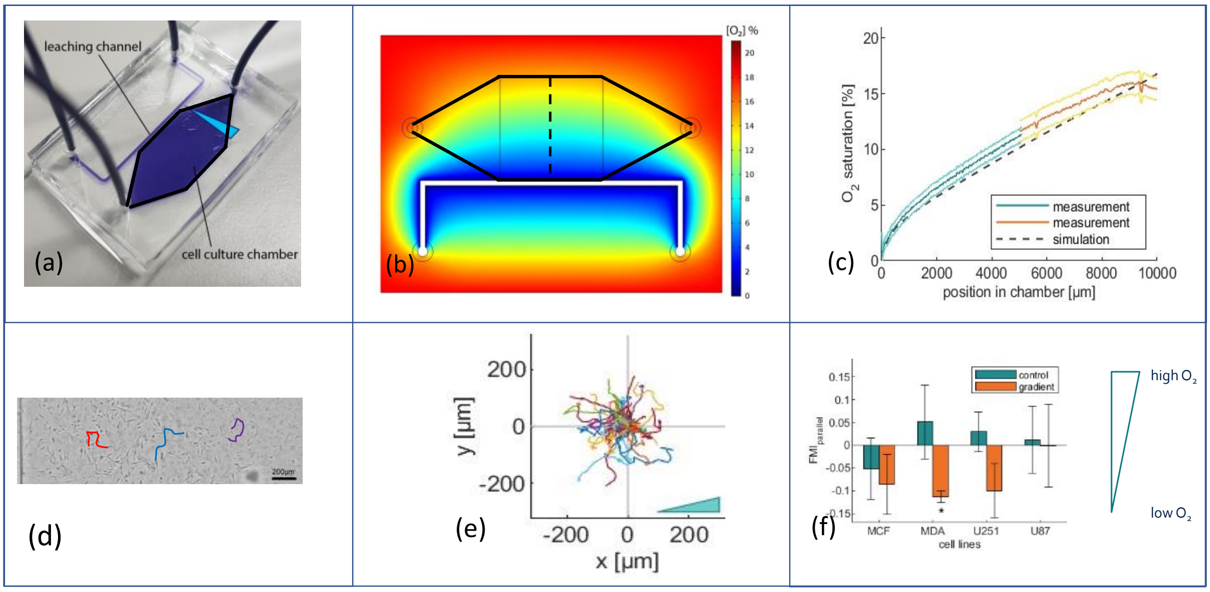


Figure 3: Oxygen gradient microfluidic device used to study cancer cell migration. (a) A picture of the microfluidic device filled with Brilliant Black BN dye. The chip is 25x30 mm. The gradient direction in the chamber is indicated with a blue triangle; (b) Simulation of the steady-state oxygen saturation in the chip. The chamber outline is shown in black; (c) The average oxygen concentration across the entire chamber, based on 10 measurements done within 1 hour; (d) Traces of three MDA-MB-231 breast cancer cells in the device; (e) Representative migration tracks for MDA-MB-231 breast cancer cells, the direction of the oxygen gradient is indicated by the blue triangle: Low oxygen at the tip, and high oxygen at the base. The starting points of the migration tracks of each cell are offset to (0,0). The graph contains the data of 50 tracked cells in a single experiment. (f) The average forward migration index parallel to the oxygen gradient, extractes from the tracks for four cell lines: MCF-7, MDA-MB-231, U251, and U87. Here, a negative value indicates a migration bias towards lower oxygen concentration.

Next to the oxygen gradient chip, we have developed devices that enable us to study the effect of the ECM on the first step in metastasis: invasion. In vivo, cancer cells initially reside in a soft basement membrane before invading the fibrous and stiffer stromal ECM [3]. To recapitulate this heterogeneous pre-invasive ECM composition, we have developed a microfluidic cell encapsulation method that creates a similar tissue morphology. In our model, MCF-7 breast cancer cells are encapsulated in Matrigel beads that mimic the basement membrane, which are then embedded in a fibrous collagen I hydrogel, mimicking the stromal ECM, as shown schematically in Figure 4(a). Figure 4(b) shows the actual cell encapsulation device, which has a flow focusing region in which droplets of liquid Matrigel are formed encapsulating cancer cells; this is region is cold so that Matrigel is liquid. In a subsequent meandering channel kept at higher temperature, the Matrigel gelates. After harvesting these Matrigel beads containing cancer cells and embedding them in collagen I matrix, we were able to make live observations of the early stages of invasion of MCF-7 breast cancer cells, Figure. 4(c), and we demonstrated the possibility to quantify individual cell positions after invasion using image analysis, Figure 4(d).

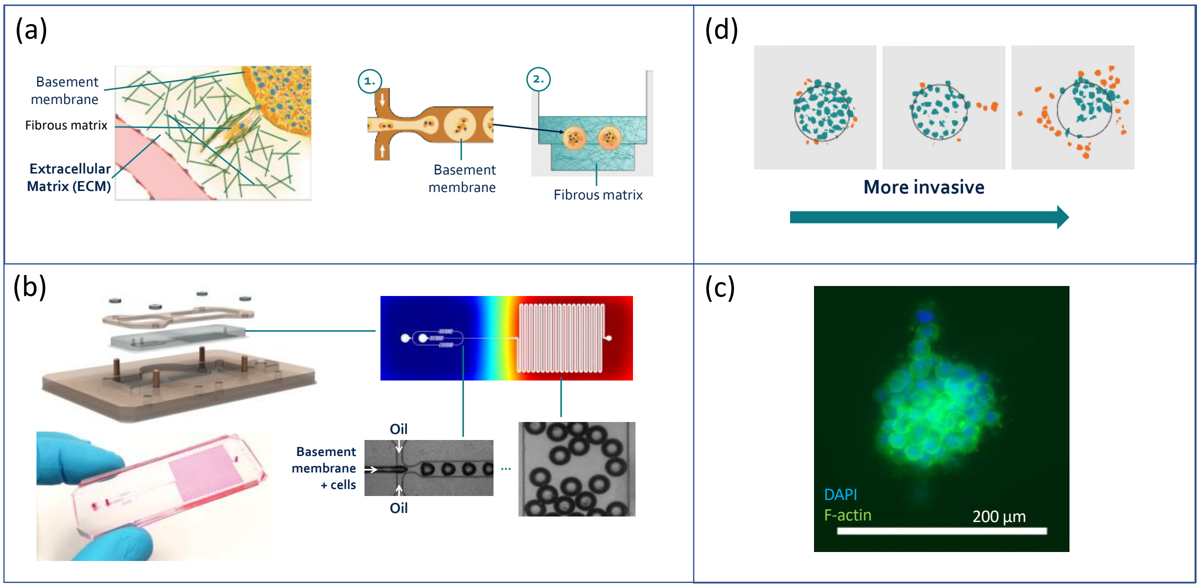


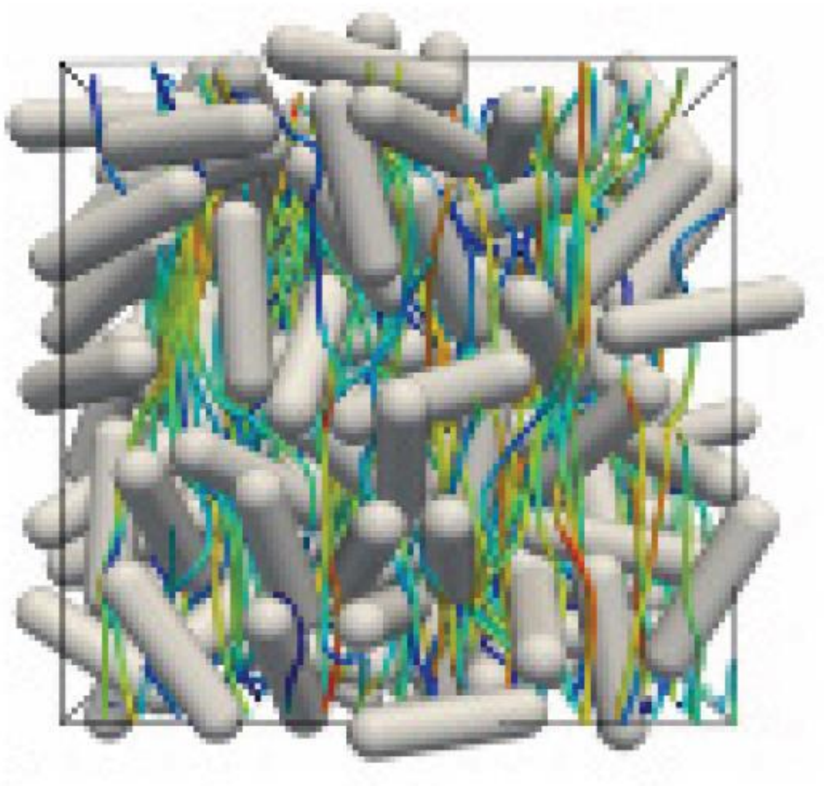
Figure 4: Breast cancer invasion chip. (a) Schematic of cancer cell invasion in which the cell breaks through the basement membrane and enters the fibrous stromal matrix. We recreate this using two steps: encapsulating cells in Matrigel beads using a microfluidic flow focusing device, followed by harvesting the beads and embedding them in fibrous collagen I. (b) Microfluidic cell encapsulation device (75mm x 25 mm); Matrigel beads encapsulating cells are formed in a flow focusing structure at <4 ℃ at which Matrigel is liquid; in a subsequent meandering channel at 37 ℃, Matrigel gelation happens. (c) We can follow invasion live; the fluorescent image is taken at the moment cells break through the basement membrane (Matrigel) and invade into the stromal matrix (collagen I); (d) Images can be analyzed and invasion can be quantified.

Both systems, for oxygen control and ECM modeling, are designed to fit together, such that the effects of oxygen gradients on cancer invasion can be studied in the future. In a broader perspective, the technologies in this research could be further developed to envelope more of the TME cues, such as additional biochemical cues, cell types, and (fluid-) mechanical cues. Potentially, they could also be applied to study other (patho-) physiological processes in the field of organ-on-a-chip. Many tissues have some form of epithelial tissue and different oxygen levels throughout, such as the lungs, liver, and gut. Summarized, our technologies can aid in deciphering the mechanisms that govern metastasis, and potentially other pathological processes [4].

References

1. Sleeboom, J.J.F., Eslami Amirabadi, H., Nair, P., Sahlgren, C.M., & Toonder, J.M.J. den (2018). Metastasis in context: Modeling the tumor microenvironment with cancer-on-a-chip approaches. Disease Models & Mechanisms 11. dmm033100. doi:10.1242/dmm.033100.
2. Sleeboom, J.J.F., Toonder, J.M.J. den & Sahlgren, C.M. (2018). MDA-MB-231 Breast Cancer Cells and Their CSC Population Migrate Towards Low Oxygen in a Microfluidic Gradient Device. Int. J. Mol. Sci. 19, 3047. doi:10.3390/ijms19103047.
3. Sleeboom, J.J.F., Sahlgren, C.M., & Toonder, J.M.J. den (2021). A microfluidic model for breast cancer invasion from a heterogeneous ECM. Under review.
4. Sleeboom, J.J.F. (2020). Microfluidic models of metastasis: In vitro approaches to study the tumor microenvironment. PhD Thesis, Eindhoven University of Technology.

RESEARCH



Research

The research programme of the JMBC has been grouped into three main research themes. The reason for this structure is to present a combination of projects which have coherence, either in terms of physical models or in terms of mathematical methods.

The three themes are:

1. Complex dynamics of fluids
2. Complex structures of fluids
3. Mathematical and computational methods for fluid flow analysis

Fluid flows in the environment or in industrial applications are almost always characterised by some complexity. Frequently, it is this complexity that makes the flow an interesting topic of research. The first form of complex dynamics that comes to mind is turbulence, in contrast to laminar flow. In particular aspects like laminar-turbulence transition, effects of thermal buoyancy, compressibility and rotation, density stratification and the interaction with chemical reactions are topics that are actively being studied by a number of the JMBC groups.

Also, the presence of different phases, e.g. in the form of particles, bubbles or drops, may add to the complexity of flows. This class of flows, generally denoted as 'dispersed multi-phase flow' forms a strong focal point of research within the JMBC. Also, non-Newtonian and granular flows form a special class of complex flows that is being studied by a number of groups.

During the last two decades micro- and nanofluidics has become a topic that has attracted substantial attention, not in the least because of its important industrial and biomedical applications.

Advanced mathematical and computational techniques have become indispensable instruments for the description and understanding of complicated flow phenomena. The rapid increase of computational power has significantly stimulated the use of simulation techniques. In areas such as turbulent flow simulation, important progress has been made through refined modelling via Large-Eddy Simulation (LES), Direct Numerical Simulation (DNS) and stochastic methods. In other areas similar trends have become feasible, such as PDF modelling in combustion, and particle-based methods , like the Lattice-Boltzmann method.

Experimental techniques also play a crucial role in modern fluid-dynamics research. Many experimental methods are based on various forms of laser diagnostics, like e.g. PIV and PTV for flow measurements and CARS and LIF for measurements of temperatures and concentrations. Also, recording of ultrafast flow phenomena via high-speed camera techniques is playing an essential role in present-day fluid mechanics.

Within the JMBC, the various groups have built up extensive expertise on these aspects of experimental , theoretical and computational fluid dynamics. Within the framework of the network provided by the research school, stimulated by the contact groups, all groups benefit from this common pool of knowledge and expertise.

Review of progress in research projects

As usual in the scientific community, progress in the research projects is reported in the form of PhD theses, journal publications, contributions to conference proceedings, (chapters of) books, and in the form of presentations at conferences. The (refereed) scientific output of the JMBC groups is presented in the Annual Reports, which are downloadable from this website. In a number of industry-funded projects, some of the JMBC groups produce output in the form of special reports for industries and technological institutes. These reports are not included in the groups' output presented in the Annual Reports. For more information, please contact the relevant Project leaders.

REPORTS OF INDIVIDUAL RESEARCH GROUPS

TU Delft

Delft University of Technology

<u>Faculty Mechanical Maritime and Material Engineering (3mE)</u>

Department Process & Energy

Energy Technology
Fluid Mechanics
Multiphase Systems
Complex Fluid Processing

Department Maritime and Transport Technology

<u>Faculty of Applied Sciences</u>

Department Chemical Engineering

Transport Phenomena
Product and Process Engineering

<u>Faculty of Electrical Engineering, Mathematics and Computer Science (EEMCS)</u>

Delft Institute of Applied Mathematics (DIAM)

Numerical Analysis
Mathematical Physics

<u>Faculty of Aerospace Engineering</u>

Department Aerodynamics, Wind Energy, Flight Performance and Propulsion

Aerodynamics
Wind Energy (Aeroacoustics)

<u>Faculty of Civil Engineering and Geosciences</u>

Department Hydraulic Engineering

Environmental Fluid Mechanics

TUD - Mechanical, Maritime and Materials Engineering (3mE) – Process & Energy

The Process & Energy department within the faculty Mechanical, Maritime and Materials Engineering (3mE) of TUD aims at enabling the energy transition by educating future (mechanical) engineers and by developing novel processes and equipment for the production and consumption of synthetic fuels, chemicals and materials. Its research covers fundamental aspects (thermodynamics and fluid dynamics) and technologies (energy technology and storage, process intensification and multiphase systems). Within the P&E department, the sections below actively participate in the J.M. Burgerscentrum.

Energy Technology
Prof.dr.ir. BJ Boersma

The Energy Technology section focuses its research and education efforts on the design and modeling of thermal energy conversion systems, with a focus on renewable energy. The section's research is performed with state-of the-art numerical tools and experimental techniques.

Fluid Mechanics
Prof.dr.ir. J Westerweel
Prof.dr.ir. RAWM Henkes *(part-time)*
Prof.dr. DJEM Roekaerts
Prof.dr.ir. W van de Water *(part-time)*
Prof.dr.ir. G Ooms *(em)*

The Fluid Mechanics (FM) section at the Laboratory for Aero &Hydrodynamics performs research on the topics of turbulence and complex flows. The research is carried out at a fundamental level using modern experimental and numerical methods and has a clear connection to processes in an applied or industrial context. The research projects generally focus on six disciplines: turbulence, multiphase flow, combustion, microfluidics, biological flows, and fluid dynamics of sports. Experimental methods include particle image velocimetry and laser induced fluorescence; numerical methods include direct numerical simulation and large-eddy simulation.

Multiphase Systems
Prof.dr.ir. C Poelma

The Multiphase Systems section, housed in the Laboratory for Aero & Hydrodynamics, studies fundamental and applied aspects of multiphase flows, inspired by industrial applications. Research is performed using state-of-the art numerical tools (DNS, immersed boundary methods) and experimental techniques (e.g. MRI, X-ray, ultrasound imaging and particle image velocimetry). The current focus is on dense suspensions, in particular on its effects on e.g. the transition to turbulence.

Complex Fluid Processing
Prof.dr.ir. JT Padding

The goal of the section Complex Fluid Processing is to understand the complex interplay between transport processes and transformations on the nano and mesoscale, and to apply this understanding to scale up and improve fluid processing devices and equipment.

The mesoscale is the scale of 0.01 micrometre to a few millimetre. This is the scale where most process limitations occur, i.e. where different transport mechanisms, such as flow, diffusion, migration, and mixing, are competing with each other, and are possibly also competing with transformations, such as chemical reactions, phase change, and aggregation. Moreover, changing surface properties at the nanoscale (nanometer or smaller) can lead to significantly altered (effective) boundary conditions at meso- and macroscales. By accurately modelling transport and surface effects, and by connecting nano- to meso- to macroscales, we are able to scale up and improve equipment for the processing of particles, liquids, gases, multi-phase fluids and non-Newtonian fluids.

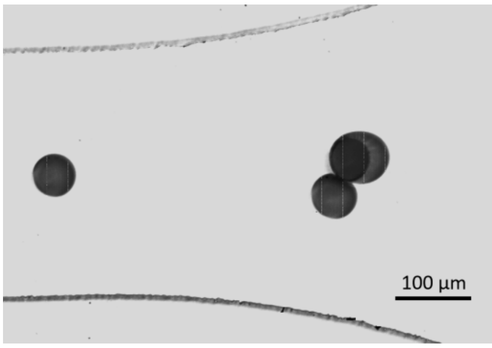
The current focus of our group is on understanding and scaling up processing of non-spherical particles (e.g. for biomass gasification in fluidized beds, microparticle separation, and processing of fluids containing carbon platelets), non-Newtonian droplets (e.g. for spray-drying of complex dispersions such as milk), crystallization (e.g. for controlled nucleation in industrial processes) and transport in porous media, including interfacial

transport phenomena and charged wall effects (e.g. for electrochemical conversion of CO2 using porous electrodes and catalyst layers).
Staff members

- Johan Padding (full professor, chair). Johan has a PhD in Applied Physics and does fundamental research in multiphase flows, mesoscale transport, rheology, and heterogeneous catalysis, with applications in fixed and fluidised beds, spray drying, crystallisation, and electrochemical reactor design. He specializes in developing particle-based and stochastic simulation methods for these applications.
- Lorenzo Botto (associate professor). Lorenzo has a PhD in Mechanical Engineering and is interested in multiphase fluid dynamics and its applications to materials science and the environment. He specializes in processing of flexible carbon platelet nanomaterials by flow, and theoretical treatment of fluid-structure coupling.
- Burak Eral (assistant professor). Burak has a PhD in Chemical Engineering and focuses on out-of-equilibrium manufacturing and separation processes, in particular on soft matter, microfluidics and crystallization applications.
- Remco Hartkamp (assistant professor). Remco has a PhD in Mechanical Engineering and is interested in interfacial transport phenomena and its application to electrochemical conversion, nanofluidic transport, and water desalination. He specializes in molecular dynamics simulations.

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders Dr. D.S.W. Tam Research theme Complex dynamics of fluids Participants B. Mulder (PhD candidate) D.S.W. Tam (PI) Jerry Westerweel (Promotor) Funded by ERC Funding % EU 100% Start of the project 2019 Information Name: Daniel Tam E-mail: d.s.w.tam@tudelft.nl	Project title Collective Dynamics of Biologically Active Fluids Project aim Collective motion is ubiquitous in the natural world; from the flocking dynamics of animal groups to the micron-scale swarm behaviour of single-cell organisms, and the metachronal coordination of beating cilia. On these small scales, hydrodynamic forces are dominant and determine the motion of micro swimmers, from single cells to swarms of millions of cells. This PhD project will investigate experimentally the interplay between hydrodynamics and cell motility. Progress • second set of improved silicon wafers are being fabricated. The wafers act as a porous membrane where microorganisms are trapped on by means of a pressure grad. applied over the membrane. • Finished prototype flow-device that controls the pressure gradient across the abovementioned wafers and lets the user inject and control fluids on both sides of the wafer. • Started using multiple organisms: Chlamydomonas R.WT.,Volvox (2x) and Gonium Pectorale. • Microfluidic geometries have been fabricated and tested in combination with the silicon wafers. • literature review has been performed where specific key areas have been identified

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders Prof. dr. ir. J. Westerweel Dr. D. S. W. Tam Research theme Complex dynamics of fluids Participants Ankur Kislaya Dr. Teng Dong Funded by Shell Technology Centre Amsterdam Funding % Industry 100 % Start of the project 2017 Information Ankur Kislaya +31 617290056 a.kislaya@tudelft.nl	Project title Particle manipulation-on-chip: Using programmable hydrodynamic forcing in a close loop Project aim To design a microfluidic device that has no real channels with physical walls such as Y-junction or T-junction devices, instead have "virtual channels". These virtual channels are dynamic and can be used as multi-purpose channels on a single device in order to integrate multiple functionalities such as trapping, separation, or sorting onto a single device. In this work, hydrodynamic manipulation is used to manipulate the flow field to create such virtual channels. Progress Implemented a PID control scheme for the microfluidic device, which assists *a priori* particle manipulation algorithm (developed in 2019) to improve the accuracy and precision of the experiment particle trajectory from the pre-computed optimized particle trajectory. The PID controller gives flow rate corrections that added to the pre-computed flow rate based on particle deviation from their prescribed path during the ongoing experiment. Currently writing dissertation as my PhD contract was finished on 1st January 2021. 2020 Publications A. Kislaya, A. Deka, P. Veenstra, D.S.W. Tam, J. Westerweel. "Psi-PIV: A novel framework to study unsteady microfluidic flows." *Experiments in Fluids*, Volume 61, Number 2, 2020. W.J. Geelhoed, K.E.A. van der Bogt, T.C. Rothuizen, F.F.R. Damanik, J.F. Hamming, C.D. Mota, M.S. van Agen, H.C.de Boer, M.Tobón Restrepo, B. Hinz, A. Kislaya, C. Poelma, A.J. van Zonneveld, T.J. Rabelink, L. Moroni, J.I. Rotmans. "A novel method for engineering autologous non-thrombogenic in situ tissue-engineered blood vessels for arteriovenous grafting". *Biomaterials*, Volume 229, 2020.

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders Prof. dr. ir. Jerry Westerweel Dr. Daniel Tam Research theme Complex dynamics of fluids Participants Teng Dong Funded by Shell Technology Centre Amsterdam Funding % Industry 100% Start of the project 2017 Information Teng Dong +31 617219903 T.Dong@tudelft.nl	Project title Flow manipulation to study the coalescence of liquid/liquid mixture in a microfluidic device Project aim From the industrial perspective, this Project aims to develop a novel microfluidic device for the diagnosis of single fluid or mixed fluids (such as the water-crude oil mixture). In fundamental research, the Project aims to understand the coalescence mechanisms between the micro-droplets surrounded by a third immiscible phase. The experimental part of the project focuses on the manipulation and the capturing of the droplet for the benefit of the observation of the coalescence. Progress By far the old version of the micro-fluidic device is largely improved by scaling down the global dimension and by changing the method of droplet generation. With the 2nd version of the device, the droplets are successfully captured and locked at a specific position. The rapid coalescence between the micro-droplets is captured by the high-speed camera, the flow fields both inside and outside of the dispersed droplets are obtained by PIV measurements. A scaling law that describes the relationship between the impact position of the droplet and the corresponding coalescence time is found to exist.  *The sliding of the free moving droplet along the trapped one.*

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders Daniel Tam Research theme Complex dynamics of fluids Participants Abel-John Buchner Junaid Mehmood Parviz Ghoddoosi Dehnavi Bob Mulder Cooperation Marie-Eve Aubin-Tam, Department of Bionanoscience TU Delft Funded by ERC Funding % EU 100% Start of the project 2017 Information Daniel Tam d.s.w.tam@tudelft.nl	Project title Origins of Collective Motion in Active Biofluids Project aim Collective motion and synchronization arising within and between simple motile organisms occurs ubiquitously and is crucial to many biological and industrial processes. This project examines the origins of spontaneous coherent motion in three model biofluidic systems: (1) flagellar synchronisation of the alga *C. Reinhardtii*, (2) metachronal wave dynamics in the cilia of protist Paramecium, and (3) collective motion of swimming microorganisms in active suspensions. Using optical tweezers and a unique μ-Tomographic PTV, the 3D micro-scale motion of these microorganisms is tracked. Progress • 3D tracking data has been acquired for the hydrodynamic “puller” species C. Reinhardtii. • A tracking algorithm has been developed to study the pairwise interactions among C. Reinhardtii. • Experiments are completed for *C. Reinhardtii* swimming in viscoelastic fluid • Hydrodynamic interactions among *C. Reinhardtii* are under investigation. 2020 Publications Mehmood, J., Muller, K., Buchner, A. J., & Tam, D. (2020). Experimental investigation of pair-wise interactions between Chlamydomonas reinhardtii. Bulletin of the American Physical Society, 65.

TUD – 3mE – Process & Energy - Fluid Mechanics

Project leaders
RAWM Henkes, WP Breugem

Research theme
Complex dynamics of fluids

Participants
MHW Hendrix

Funded by
Shell Global Solutions International BV

Funding %
Industry 100%

Start of the project
2013

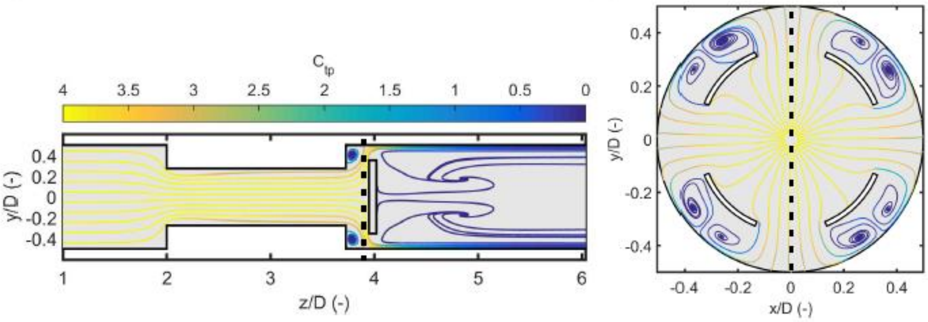
Information
R.A.W.M. Henkes, 0652096201,
R.A.W.M.Henkes@tudelft.nl

Project title
Modelling and experiments for by-pass pigging with speed control

Project aim
A Pipeline Inspection Gauge (pig) is a cylindrical device that fits onshore or offshore pipelines for the transport of gas, oil, water in the oil and gas industry. The pig is launched at the inlet and received at the outlet. It is used for various purposes, such as water removal to prevent corrosion, removal of wax deposition along the pipe walls, removal of other solids, and inspection of the pipe wall condition. The use of a pig with an opening in the centre (a so-called by-pass pig) will allow some of the fast-moving gas to flow through the pig during the pigging operation. The aim of this PhD project is to develop an optimum way to control the speed of the by-pass pig.

Progress
PhD student Maurice Hendrix has completed and defended his PhD Thesis. The thesis contains:

- Single phase CFD simulations for the pressure loss coefficient of various by-pass pig configurations.
- Lab experiments for the friction between the pig disk and the pipe wall.
- Finite element simulations for the deformation of the pig disks.
- Lab experiments and simulations for low-pressure pigging, showing the slip-stick behaviour.
- Accurate numerical solver for the 1D two-fluid model for pigging of twophase pipe flow.



Simulation of the flow around a by-pass pig in a pipe

2020 Dissertations
Hendriks, M.H.W., Experiments and modelling for by-pass pigging of pipelines, PhD Thesis, Delft University of Technology, 2020

TUD – 3mE – Process & Energy - Fluid Mechanics

Project leaders
R.A.W.M. Henkes, G. Ooms, M. Pourquié

Research theme
Complex dynamics of fluids

Participants
Haoyu Li (aio)
Fan Jiang (guest researcher)

Funded by
China Scholarship Council

Funding %
Industry 25%
Scholarships 75%

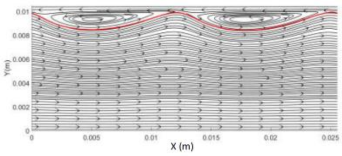
Start of the project
2018

Information
R.A.W.M. Henkes, 0652096201,
R.A.W.M.Henkes@tudelft.nl

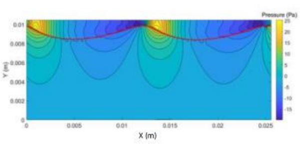
Project title
Liquid-liquid interfaces

Project aim
The aim of the project is to investigate the liquid-liquid interfaces which are found in the core-annular flow of a very viscous oil and water, both experimentally and numerically. The focus is on the interaction between the interface waves and the turbulent water film layer along the pipe perimeter. Comparisons of the experimental and numerical results will give new insights into the properties of the flow, such as the levitation mechanisms and the drag experienced. Studying the influence of the fluid properties on the oil-core flow is another aim. That means that carrying out experiments with different values of the density difference, core-viscosity and interfacial tension. Furthermore, various turbulence models will be considered in the numerical calculations. These turbulence models need to be investigated further in order to legitimate their usage.

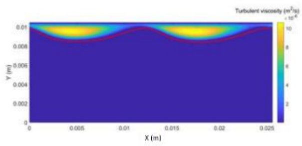
Progress
Lab experiments for horizontal water-oil pipe flow were carried with a high-speed camera from which the structure of the travelling waves at the interface as well as the eccentricity of the oil core could be quantified. Detailed CFD simulations were carried out with OpenFOAM for a horizontal pipe to determine the relation between the oil flow rate, the water cut, the pressure drop and the water holdup fraction. The structure of the turbulent water annulus was analysed. In the CFD model the turbulence was represented by a low-Reynolds number k-ε model. Both the simulations and the experiments show that the wave velocity is almost constant such that the temporal and spatial wave structure can be directly converted in one another, which makes the analysis easier. Simulations were also carried out to investigate the conditions that the oil waves touch the pipe wall, breaking the oil-core annular flow structure. Two papers on the results are in preparation. The next step is to carry out simulations at similar conditions using a Large-Eddy Simulation approach.



(a)



(b)



(c)

2D results with respect to observer that travels with the wave velocity; (a) streamlines, (b) isobars, (c) turbulent viscosity.

TUD – 3mE – Process & Energy - Fluid Mechanics

Project leaders
R.A.W.M. Henkes
L. Portela
J. van 't Westende
A. Greidanus

Research theme
Complex structures of fluids

Participants
A. Greidanus

Cooperation
TNO

Funded by
TKI project

Funding %
TKI 100%

Start of the project
2019

Information
R.A.W.M. Henkes, 0652096201,
R.A.W.M.Henkes@tudelft.nl

Project title
Multiphase flow in an annulus

Project aim
This project is a combined experimental/modeling study for the multiphase flow through an annulus. When gas reservoirs become older, the diameter of the production tubing becomes too large for the remaining gas production, and due to the resulting low gas velocities, liquids are no longer lifted with the gas, but instead they start to accumulate; this is the so-called liquid loading problem. To prevent this an additional pipe (so-called velocity string) is placed in the centre of the original pipe, which creates an annulus. As the cross sectional area of the annulus is smaller than the cross sectional area of the original pipe, the gas velocity will increase and liquid loading is prevented. The aim of the project is to create a design model, and validate this against lab experiments for the flow regime, pressure drop, and liquid accumulation.

Progress
The annulus flow facility in the Process & Energy lab consists of a vertical pipe-in-pipe with a length of 13 m, outer diameter of the inner pipe is 124 mm, and the inner diameter of the outer pipe is 100 mm. Air and water are the working fluids. The inner pipe was replaced with one with a smaller diameter of 80 mm. Master student Friedrich Anastasopoulos has measured the pressure drop, liquid holdup and flow structure for the new configuration. By comparison the results for the two hydraulic diameters the measured quantities could be scaled such that the basic physics of the interaction between the gas core and the liquid film (through the wavy interface) could be described, as well as the onset of the transition from annular flow to churn flow at reduced gas flow. Journal papers on the scaling laws are in preparation.

Flow facility in the Process & Energy lab at TU Delft

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders R. Pecnik and D.J.E.M. Roekaerts Research theme Complex dynamics of fluids Participants Simone Silvestri Funded by TU Delft Funding % University 100% Start of the project 2016 Information S. Silvestri 0626345924 s.silvestri@tudelft.nl	Project title Coupled Radiative and Convective heat transfer in high temperature turbulent flows Project aim The presence of radiative heat transfer in turbulent flows can lead to Turbulence Radiation Interactions (TRI). Whereas influence of turbulence on radiative sources and flux has been studied widely the modification of the properties of the turbulent velocity and scalar fields by thermal radiative transport is less will studied. Our aim is to investigate the physical mechanism underlying TRI in its full generality in order to advance predictive models. Progress We studied the effect of thermal expansion and contraction on thermally developing turbulent channel flows. Using different constitutive relations for viscosity we analyzed the response of variable property flows to streamwise acceleration/deceleration by separating it from the effect of wall-normal property variations. We demonstrated that, beyond a certain streamwise location, the flow can be considered in a state of “quasi-equilibrium” regarding semi-locally scaled variables. As such, we claimed that the development of turbulent quantities due to streamwise acceleration / deceleration is localized to the region of impulsive heating/cooling, while changes in turbulence occurring further downstream can be attributed solely to property variations. This finding allows to study turbulence modulation in accelerating/decelerating flows using the semi-local scaling framework. By investigating the energy redistribution among the turbulent velocity fluctuations, we concluded that a change in bulk streamwise velocity has a non-local effect which originates from the change in mean shear and modifies the energy pathways through velocity- pressure-gradient correlations. On the other hand, the wall-normal property gradients have a local effect and act through the modification of the viscous dissipation. We show that it is possible to superimpose and compare the two different effects when using the semi-local scaling framework. We also performed thermally developing simulations of radiatively participating media to study the effect of radiative heat transfer and turbulence-radiation-interactions on turbulence modulation and the fluctuating temperature field. Preliminary results show the complete laminarization and recovery of turbulence due to the extreme acceleration in a surprisingly short space and, as such, highlight a new “meta-stable” state of turbulence in which the flow is laminar at relatively high Reynolds number and undergoes a bypass transition localized where perturbations from the centre of the channel reach sufficiently near the walls. Additionally, we implemented an innovative compressible DNS channel code which runs exclusively on GPUs with the native GPU CUDA language. This efficient code has the potential to be the fastest DNS code as it is almost 2X faster than current compressible GPU channel codes found in literature. The dissertation: “High Temperature Turbulence: coupling of radiative and convective heat transfer in turbulent flows” was written and will be defended in 2021. 2020 Publications S. Silvestri, D.J.E.M. Roekaerts, R. Pecnik, Modelling turbulent heat flux accounting for Turbulence-Radiation Interactions, International Journal of Heat and Fluid Flow, Volume 89, 2021, 108728, ISSN 0142-727X. 2020, https://doi.org/10.1016/j.ijheatfluidflow.2020.108728.

TUD – 3mE – Process & Energy - Fluid Mechanics	
Project leaders D.J.E.M.Roekaerts Research theme Complex structures of fluids Participants Shuzhong Wang, Xi'an Jiaotong University, China Cooperation J.A. van Oijen, TU/e Funded by China Scholarship Council Funding % Scholarships 100% Start of the project 2017 Information D.J.E.M. Roekaerts d.j.e.m.roekaerts@tudelft.nl	Project title Computational modeling of hydrothermal combustion Project aim This project aims at providing computational fluid dynamics models for hydrothermal combustion, the combustion in supercritical water, i.e. at pressure higher than the critical pressure of water (22.1MPa). Appropriate thermodynamic, transport and flow models are developed and combined in simulations of representative reacting flow simulations and validated by experimental data. Progress The Eddy Dissipation Concept (EDC) model and the flamelet generated manifolds (FGM) model have been applied to study the temperature profiles and extinction limits of non-premixed hydrothermal methanol flames in a lab-scale reactor. Predictions by the two models have been evaluated by comparison with experimental data in the literature. The FGM model shows relatively better prediction of temperature than the EDC model in the near nozzle field. The EDC model can predict extinction temperatures with a deviation of 10–33 K. The extinction flow rates predicted by the FGM model are higher than those by the EDC model. Flow fields and reaction source terms are analyzed to identify the inherent mechanism leading different results by the two models. It is shown that the positive effect of turbulence on the reaction rate near the nozzle predicted by the FGM model is the essential reason causing different flame characteristics compared to the EDC model for which the turbulence only has negative effect on reaction rate. 2020 Publications Mengmeng Ren, Shuzhong Wang, N. Romero-Anton, Junxue Zhao, Chong Zou, Dirk Roekaerts Numerical study of a turbulent co-axial non-premixed flame for methanol hydrothermal combustion: Comparison of the EDC and FGM models, The Journal of Supercritical Fluids 169 (2020) 105132

TUD – 3mE – Process & Energy - Multiphase Systems

Project leaders
Wim-Paul Breugem

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Tariq Shajahan

Funded by
NWO

Funding %
NWO 100%

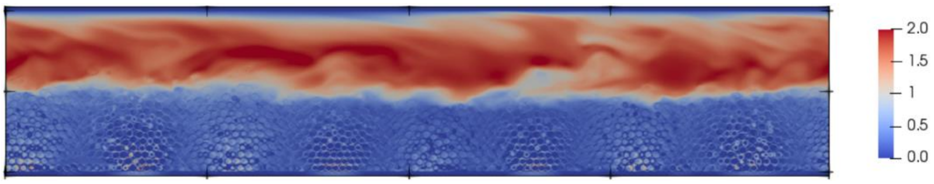
Start of the project
2017

Information
Tariq Shajahan
06 173 215 18
M.T.Shajahan@tudelft.nl

Project title
Modelling of turbulent hyperconcentrated sediment transport in horizontal pipes

Project aim
In dredging applications, deep sea mining and land reclamation projects typically large amounts of sediments are transported through pipes in the form of hyper-concentrated sediment-water mixtures or slurry. Depending on the flow and sediment concentrations, the different transport regimes can be identified. The goal of this project is to study this transport phenomenon by the development of accurate physics-based prediction tools for the wide range of operating conditions in practice. To this purpose fully-resolved numerical simulations will be performed of turbulent sediment transport through a small-scale horizontal pipe.

Progress
We studied the pressure driven transport of dense particulate suspensions of heavy particles by means of fully-resolved Direct Numerical Simulation (DNS) based on an Immersed Boundary Method. We compared the DNS with experiment carried out in a closed slurry flow loop. The experiment and DNS show good qualitative agreement for the three expected regimes: fixed bed, sliding bed and fully-suspended regimes. We have analyzed the results and are preparing a manuscript for the same. Following our previous study on sedimentation, we extended the DNS to a wider parameter space. We have been preparing a draft based on our new study on sedimentation.



2020 Publications
T. Shajahan, W.-P. Breugem, Influence of concentration on sedimentation of a dense suspension in a viscous fluid, Flow, Turbulence and Combustion 105 (2) (2020) 537–554.

TUD – 3mE – Process & Energy - Multiphase Systems

Project leaders
dr. ir. WP Breugem
prof. dr. AEDM van der Heijden

Research theme
Complex dynamics of fluids

Participants
ir. W Peerbooms
dr. ir. WP Breugem
prof. dr. AEDM van der Heijden

Funded by
TNO

Funding %
TNO 100%

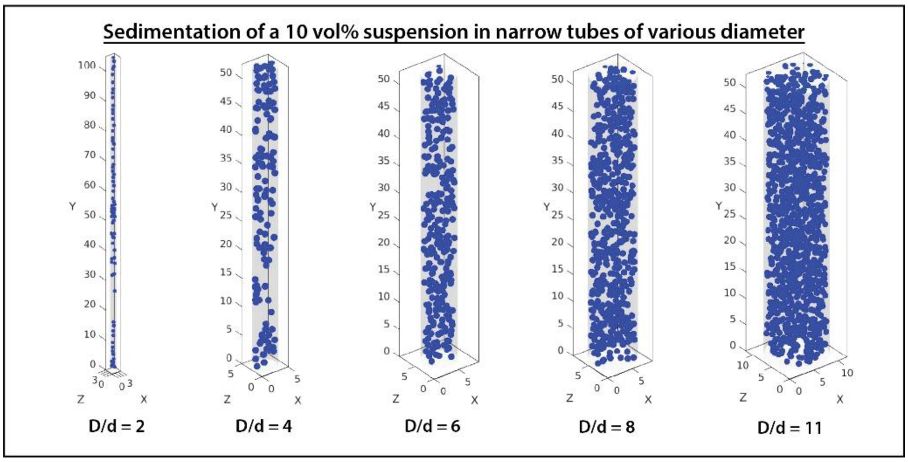
Start of the project
2019

Information
W Peerbooms
w.peerbooms@tudelft.nl

Project title
Rheology of dense suspensions for 3D-printing of energetic materials

Project aim
The aim of this project is to get a better understanding of the physical phenomena at the particle scale that are responsible for the non-Newtonian macroscopic rheology of a dense suspension in the nozzle of a 3D printer. Accurate modelling of the suspension rheology is needed for optimizing the printing process by tuning the process conditions and material properties. We will study the macroscopic rheology in relation to the suspension microstructure by means of fully-resolved simulations (DNS) of the suspension flow. In addition, experiments will be conducted to measure the rheological suspension behaviour.

Progress
A large portion of 2020 has been used to adapt the DNS code to enable the investigation of sedimentation of dense suspensions in narrow tubes. Next, sedimentation of dense suspensions has been studied and compared with the gravitational settling of a single particle in narrow tubes. The tubes have a diameter ranging from 2 to 11 particles diameters and the solid content is 5 to 30 volume percent for the suspensions. A scaling relation is found for the sedimentation velocity of a single particle. Scaling relations and particle microstructure are being studied for dense suspensions.



TUD – 3mE – Process & Energy - Multiphase Systems

<u>Project leaders</u>
Christian Poelma
Jerry Westerweel

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Mike van Meerkerk
Wout Cornel
Christian Poelma
Jerry Westerweel

<u>Funded by</u>
STW – Perspectief SLING

<u>Funding %</u>
NWO 100 %

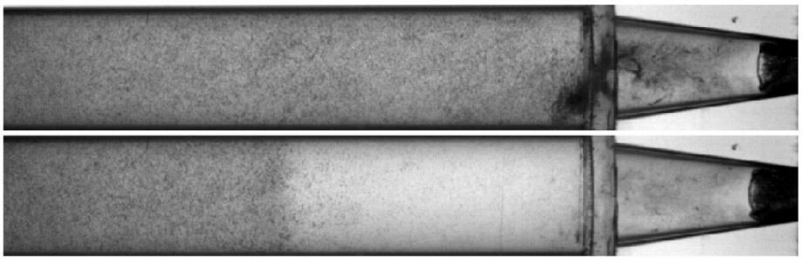
<u>Start of the project</u>
2016

<u>Information</u>
Mike van Meerkerk
M.vanMeerkerk@tudelft.nl
Wout Cornel
W.A.Cornel@tudelft.nl

<u>Project title</u>
Sloshing of Liquefied Natural Gas – Variability and Multiphase dynamics

<u>Project aim</u>
Measure the instabilities on top of a single impact wave to relate the variability of the liquid free-surface to those of the impact pressure. The vapor-liquid free surface is dynamically mapped in 3D using a non-intrusive instantaneous measurement technique. Furthermore, impact experiments on controlled pre-aerated liquids are performed to study the bubble dynamics. The effect of liquid compressibility on the liquid loads during sloshing events is studied by varying the gas fraction and density ratio with and without phase changes.

<u>Progress</u>
Novel experimental techniques are developed for the complex measurements in the multiphase wave lab. Measurements with the three-dimensional, non-intrusive technique show the ability to measure small- and large-scale structures over a relatively large domain. The free surface measurement technique is used in a wave flume at the Environmental Fluid Mechanics Laboratory of the TU Delft. Also, shock waves are measured by high-speed imaging of (micro)bubbles at various conditions. The effect of phase changes on the shock wave front speed and pressure magnitude is studied by the collapses of large vapor bubble cloud structures in a venturi setup.



<u>2020 Publications</u>
van Meerkerk, M., Poelma, C. & Westerweel, J. Scanning stereo-PLIF method for free surface measurements in large 3D domains. *Exp. Fluids* 61, 19 (2020)

van Meerkerk, M., Poelma, C., Hofland, B. & Westerweel, J. Experimental investigation of wave tip variability of impacting waves. Physics of Fluids 32 (2020)

TUD – 3mE – Process & Energy - Multiphase Systems	
Project leaders Prof. Christian Poelma Research theme Complex dynamics of fluids Participants Dr Angeliki Laskari Dr Sebastian Schreier Esra Uksul Funded by 3ME Faculty, TU Delft Funding % University 100% Start of the project 2020 Information Christian Poelma	Project title Boundary layer effects on flexible floating structures; an experimental investigation using PIV/DIC Project aim For future applications of VLFS's, considerations of viscoelastic responses to ocean waves are necessary, as a wider spectrum of flexibility for floating structures get considered going forward. Comparisons between existing viscoelastic models and experimental studies of flexible covers interacting with waves have highlighted significant discrepancies. The influence of a (turbulent) boundary layer beneath the viscoelastic covers is believed to be the major contributor to the mismatch. The interactions between flexible or compliant surfaces and turbulent boundary layers involve complex two-coupled fluid-structure interactions that will investigated in this project. Progress So far, a literature survey has been conducted and the first preliminary test measurements completed. With the insight and conclusions drawn from them, the upcoming free surface, planar PIV measurements on linear waves were conducted.

TUD – 3mE – Process & Energy - Multiphase Systems

Project leaders
Dr. Ir. Wim-Paul Breugem
Dr. Ir. Burak Eral

Research theme
Complex dynamics of fluids

Participants
Yavuz Emre Kamis

Funded by
Kreber BV

Funded %
University 50 %
Industry 50 %

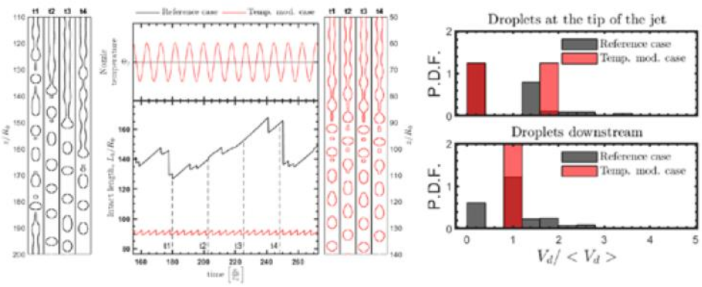
Start of the project
2019

Information
Yavuz Emre Kamis
06 27135505
Y.E.Kamis@tudelft.nl

Project title
Controlled jet formation and break-up of non-Newtonian jets for prilling applications

Project aim
The aim is to investigate the break up of straight and spiralling jets in regards to the prilling process, and find ways to control drop size which is essential to produce a good quality prill (e.g. fertilizers). The multi-faceted approach contains: 1) rheological characterization of working materials, 2) simulating non-Newtonian and nonisothermal jet breakup to investigate the formation of satellite drops and 3) experiments to validate theoretical and numerical predictions.

Progress
The slender jet framework that was implemented last year is extended to accommodate energy equation and to simulate the jet behaviour beyond pinch off. Using this, the concept of using thermal modulation to control jet breakup and droplet formation is numerically investigated. Results are submitted to Physical Review Fluids.



TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Prof. dr.ir. Johan T. Padding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Sathish K.P. Sanjeevi, Vinay V. Mahajan, Ivan Mema, Dr. Barry W. Fitzgerald, Dr. Ahad Zarghami, Dr.Vikrant Verma, Dr. Yousef M.F. El Hasadi

Cooperation
TU Eindhoven, (Hans Kuipers and Kay Buist, for MPT experiments)

Funded by
European Research Council
(ERC Consolidator Grant)

Funding %
EU 100%

Start of the project
2014

Information
Prof. Johan T. Padding
J.T.Padding@tudelft.nl

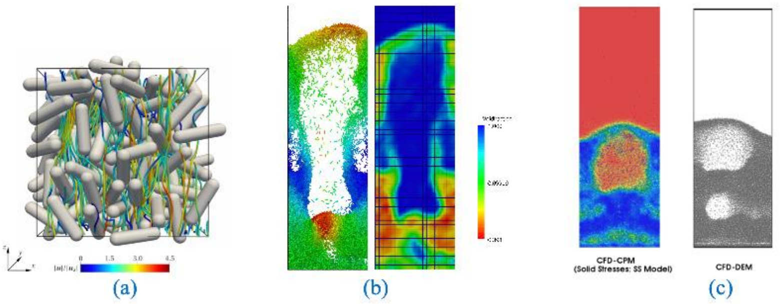
Project title
Multiscale modelling of dense gas-fluidized flows of non-spherical particles

Project aim
The aim of this project is to predict the hydrodynamics of dense fluidized systems containing elongated particles, using a multiscale modelling approach and experiments. The multiscale approach consists of: 1) fully resolved simulations (DNS) to obtain closures for translational and rotational drag for dense particle systems, 2) discrete particle model (CFD-DEM) simulations to validate the drag closures with matching experiments (PIV, MPT and XRT) and to obtain statistics of the particle stress tensor due to inter-particle collisions, 3) a novel Lagrangian method based on the particle stress tensor (CFD-CPM) for large scale simulations.

Progress
In 2020 this ERC project was concluded and the last PhD student graduated. The closures for hydrodynamic drag, lift and torque on assemblies of axisymmetric, non-spherical particles obtained from detailed DNS (Figure a) have been used in CFD-DEM simulations and have been validated by comparison with Magnetic Particle Tracking and X-Ray Tomography experiments. A novel particle stress based continuum particle model (CFDCPM, Figure c) has been published and will be further developed in forthcoming projects.

2020 Dissertations
I. Mema, Spheres vs rods in fluidized beds – numerical and experimental investigations

2020 Publications
1. V. Verma and J.T. Padding, A Novel Approach to MP-PIC: Continuum Particle Model for Dense Particle Flows in Fluidized Beds, Chem. Eng. Sci. X 6, 100053 (2020).
2. I. Mema, K.A. Buist, J.A.M. Kuipers and J.T. Padding, Fluidization of spherical versus elongated particles: Experimental investigation using magnetic particle tracking, AIChE Journal 66, e16895 (2020).
3. S.K. Pacha Sanjeevi and J.T. Padding, Hydrodynamic forces on monodisperse assemblies of axisymmetric elongated particles: orientation and voidage effects, AIChE Journal 66, e16951 (2020).
4. I. Mema, E.C. Wagner, J.R. van Ommen and J.T. Padding, Fluidization of spherical versus elongated particles - experimental investigation using X-ray tomography, Chem. Eng. J. 397, 125203 (2020).
5. I. Mema and J.T. Padding, Spherical versus elongated particles – numerical investigation of mixing characteristics in gas fluidized bed, Chem. Eng. Sci. X
8, 100079 (2020).



(a) (b) (c)

TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Prof. Johan T. Padding, Prof. Jan Peter van der Hoek

Research theme
Complex structures of fluids

Participants
Onno Kramer, Peter de Moel, Eric Baars, Leon Kors, Kay Buist, Tim Nijssen, Edo Boek, Hans Kuipers

Cooperation
Hogeschool Utrecht-ILC, Queen Mary University London, Waternet

Funded by
Waternet (Amsterdam)

Funding %
Industry 100%

Start of the project
2016

Information
Onno Kramer
onno.kramer@waternet.nl

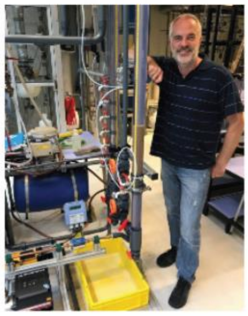
Project title
Hydraulic modelling of liquid-solid fluidisation in drinking water treatment processes

Project aim
Improve drinking water treatment processes to make a contribution to a more sustainable society.

Progress
8 journal articles finished, draft dissertation finished,
planned PhD defence planned on 10 September 2021.

Scientific Publications
1. O.J.I. Kramer, J.T. Padding, W.H. van Vugt, P.J. de Moel, E.T. Baars, E.S. Boek, J.P. van der Hoek, Improvement of voidage prediction in liquid-solid fluidized beds by inclusion of the Froude number in effective drag relations, International Journal of Multiphase Flow. 127 (2020) pp. 1-13. https://doi.org/10.1016/j.ijmultiphaseflow.2020.103261.
2. O.J.I. Kramer, P.J. de Moel, J.T. Padding, E.T. Baars, Y.M.F. el Hasadi, E.S. Boek, J.P. van der Hoek, Accurate voidage prediction in fluidisation systems for full-scale drinking water pellet softening reactors using data driven models, Journal of Water Process Engineering. 37 (2020) pp. 1-15. https://doi.org/10.1016/j.jwpe.2020.101481





TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Prof.dr.ir. Johan T. Padding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Dr. Elyas Moghaddam, Prof. dr. Andrzej Stankiewicz, Prof. dr. Esmail Foumeny

Funded by
TUD

Funded %
University 100 %

Start of the project
2017

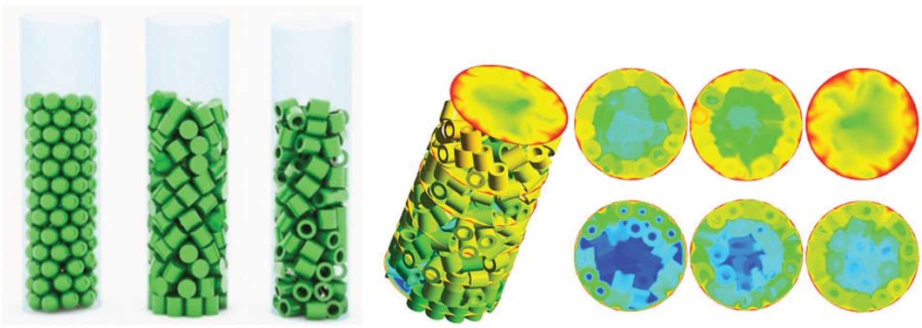
Information
Prof. Johan T. Padding
J.T.Padding@tudelft.nl

Project title
Computational modelling of fixed beds containing non-spherical pellets

Project aim
The aim of this project is to predict the hydrodynamics and heat/mass transfer in fixed beds containing non-spherical pellets. Using these simulations, the accuracy of effective pseudo-continuum approaches is evaluated and correlations for effective transport parameters are determined. Special attention will be given to often-used particle shapes such as cylinders and Raschig rings. Moreover, the effect of reactor to pellet diameter ratio will be investigated.

Progress
The hydrodynamics in packed beds containing Raschig rings has been investigated. Local flow structures have been identified, and show large deviations from azimuthally averaged flow profiles, typically assumed in effective axisymmetric models. We have investigated the heat transfer characteristics in such beds, the results of which are published in 2021.

2020 Publications
E.M. Moghaddam, E.A. Foumeny, A.I. Stankiewicz and J.T. Padding, Hydrodynamics of narrow-tube fixed bed reactors filled with Raschig rings, Chem. Eng. Sci. X 5, 100057 (2020).



TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Prof. dr. ir. Johan T. Padding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
ir. Stephan Sneijders

Cooperation
TetraPak, DSM, Nutricia-Danone, Wageningen University

Funded by
TTW, Industry

Funded %
NWO 70%
Industry 30%

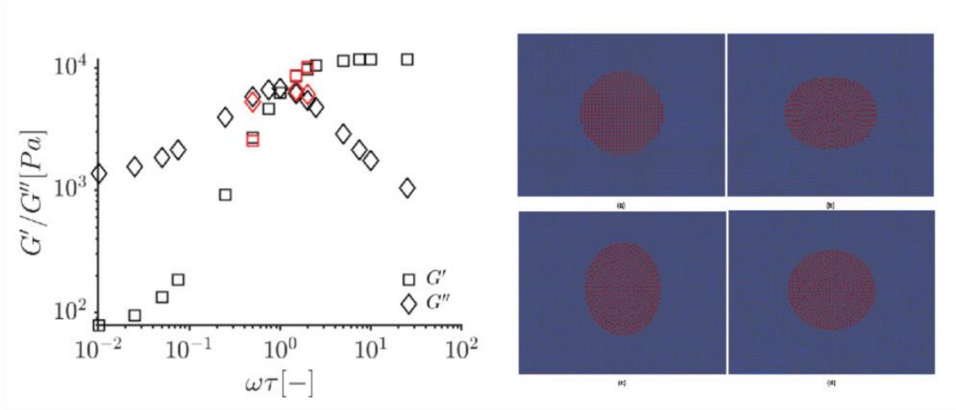
Start of the project
2017

Information
Stephan Sneijders
s.sneijders@tudelft.nl

Project title
Modelling the drying of multicomponent dispersion droplets

Project aim
In an effort to gain better understanding of spray drying processes, a numerical model describing drying and collisions on droplet scale will be developed. The model should be able to predict the droplet drying rate, the morphological changes, and the agglomeration probability. The approach chosen is multidisciplinary and includes experimental work involving well-defined droplet drying and agglomeration studies, soft matter theory development and numerical modeling. The numerical framework should allow the study of droplet collisions and impacts considering multiple physical phenomena, e.g. rheology and evaporation.

Progress
Smoothed particle hydrodynamics has been extended with pair-wise interaction forces in order to simulate viscoelastic fluids. Also, responsive particle hydrodynamics has been incorporated with the same purpose. Both methods are compared and the responsive particle hydrodynamics approach shows to be more useful for our applications. The effect of pair-wise interaction forces on surface tension has also been investigated. The influence on surface tension is there but can be counteracted with appropriate surface tension models like the CSF approach. The CSF method shows to be unaffected by the pair-wise forces.



TUD – 3mE – Process & Energy – Complex Fluid Processing

<ins>Project leaders</ins>
Prof. dr. ir. Johan T. Padding
Dr. ir. H. Burak Eral

<ins>Research theme</ins>
Complex structures of fluids

<ins>Participants</ins>
Rumen N. Georgiev

<ins>Cooperation</ins>
University of Hawaii at Manoa
Universiteit Utrecht

<ins>Funded by</ins>
NWO

<ins>Funded %</ins>
NWO 100 %

<ins>Start of the project</ins>
2016

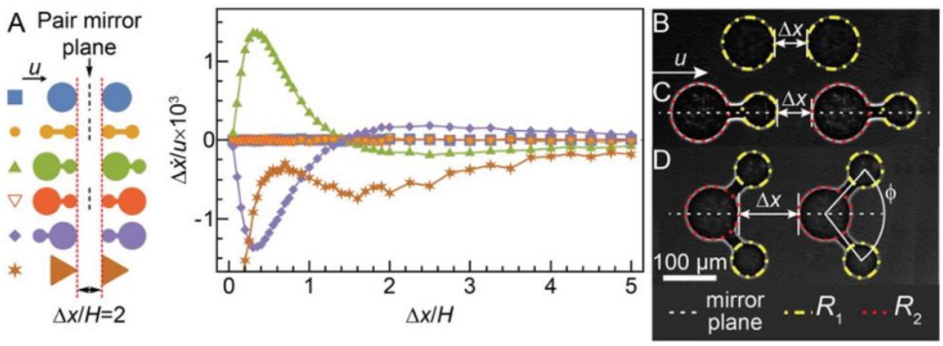
<ins>Information</ins>
Rumen N. Georgiev
r.n.georgiev@tudelft.nl

<ins>Project title</ins>
Hydrodynamic interactions in quasi-2D creeping flow

<ins>Project aim</ins>
To gain insight into sub-millimetre separation of species we consider micron-sized aspherical particles subjected to creeping flow in a microfluidic device. Particles in confined flow can interact hydrodynamically with each other or with surrounding surfaces, altering their trajectories. To establish a quantitative relation between the shape of a particle and its trajectory we employ experimental and numerical methods. Experimental choices are guided by finite element computations, which are, in turned verified by experimental data. Both methods serve as means towards the development of a unified theory of shape-dependent motion at the microscale.

<ins>Progress</ins>
Confined flowing particles of all shapes and sizes are present in various applications: from blood flow through tissue capillaries to industrial-scale processes. A particle in flow is strongly coupled to its surroundings via hydrodynamic interactions, which determine its motion. We investigate these interactions in pairs of non-circular objects in a Hele-Shaw cell. Particles in asymmetric pairs exhibit a relative motion, dependent on both shape and interparticle distance. By combining experimental and numerical techniques we study this shape-specific relative motion and conclude that flowing particles should self-assemble into one-dimensional flowing crystals regardless of their shape.

<ins>2020 Publications</ins>
R.N. Georgiev et al., Universal motion of mirror-symmetric microparticles in confined Stokes flow, Proceedings of the National Academy of Sciences of the United States of America, 117 (36), 21865-21872, 2020.



TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Prof.dr.ir. Johan T. Padding
Dr. Remco M. Hartkamp

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Rong Fan, Githin T. Zachariah, Parsa Habibi

Funded by
China Scholarship Council

Funded %
Scholarships 100 %

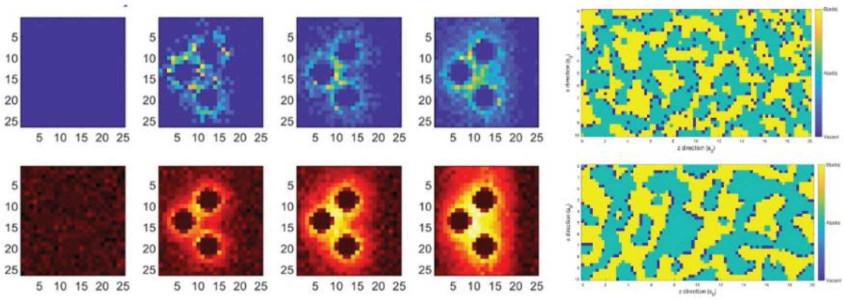
Start of the project
2017

Information
Rong Fan
r.fan@tudelft.nl

Project title
Mesoscopic simulations of transport phenomena in reactive systems

Project aim
The scope of the program is to study the interplay of transport phenomena (convection, diffusion) and reactions taking place near surfaces in structured reactors and porous media. The pore spaces in catalytic pellets may become so small that non-continuum (molecular) effects such as Knudsen diffusion start to play a role. On the other hand, the relevant scales where these phenomena take place are usually so large that detailed Molecular Dynamics simulations are also out of the question. In such cases it will be advantageous to use mesoscale particle-based simulations methods.

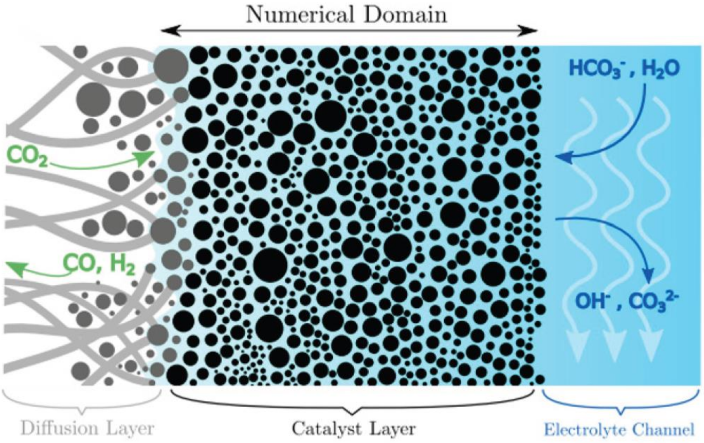
Progress
The SRD (Stochastic rotation dynamics) method for temperature measurement and chemical reactive system has been developed.

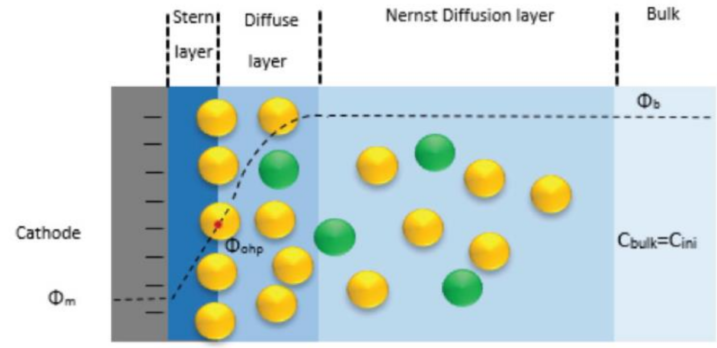


(a) (b)

(a) *Variation of local product particle number density and temperature in cross section of domain with time.*

(b) *Island formation on a surface between two adsorbed species A and B when taking into account nearest neighbour interactions (top figure) and when considering 1st and 2nd nearest neighbours (bottom figure).*

TUD – 3mE – Process & Energy – Complex Fluid Processing	
Project leaders Prof.dr.ir. Johan T. Padding Dr.ir. J. Willem Haverkort Research theme Mathematical and computational methods for fluid flow analysis Participants Joseph (Joe) Blake Funded by NWO-TTW Funded % NWO 100 % Start of the project 2019 Information Joseph (Joe) Blake +44 7954150352 J.W.Blake@tudelft.nl	Project title Optimized and scalable CO 2 reduction by tuning reaction environment and transport properties in gas diffusion cells Project aim The goal of the project is to develop and use 3D models to resolve the pore-scale interplay of diffusion, migration, advection and mean field chemical kinetics in structured porous and bi-porous media. This will allow us to improve e.g. flooded agglomerate models, and to investigate electrokinetic effects in porous electrodes. Progress Development of a numerical model and an analytical approximation for the transport and chemical/electrochemical reaction interplay within a CO2 reduction gas-diffusion electrode catalyst layer. 

TUD – 3mE – Process & Energy – Complex Fluid Processing	
Project leaders Prof.dr.ir. Johan T. Padding Dr. ir. Remco M. Hartkamp Research theme Mathematical and computational methods for fluid flow analysis Participants Esaar Naeem Butt Funded by Shell, Topsectors Chemistry, HTSM and Energy Funded % Industry 100 % Start of the project 2019 Information Esaar Naeem Butt e.n.butt@tudelft.nl	Project title Bridging scales in electrochemical reactor design Project aim Optimization of an electrochemical reactor, maximizing the CO2 electro-reduction for the desired product of formic acid (HCOOH). Such electrochemical models are usually developed based on well-established continuum-scale techniques; however, the predictive power of these models is restricted by several underlying Physico-chemical phenomena, that are occurring at smaller length and time scales. This project would focus on developing phenomenological descriptions and effective correlations of these smaller-scale phenomena, within a continuum-scale framework. Progress A comprehensive electrochemical model for CO2 reduction to formate has been developed. Key features of the model include a more realistic approach towards the Electric Double Layer phenomenon based on the Gouy-Chapman- Stern theory, entropic size effects including that of the solvent molecules, and a more accurate representation of the electrode kinetics based on the Frumkin interpretation of heterogeneous reactions. 

TUD – 3mE – Process & Energy – Complex Fluid Processing	
Project leaders Prof.dr.ir. Johan T. Padding Dr. Remco M. Hartkamp Research theme Mathematical and computational methods for fluid flow analysis Participants Arvind Pari Cooperation Dr. James E. McClure (Virginia Tech) Dr. Zhe Li (Virginia Tech) Prof. Cor P.J.W. van Kruijsdijk (Shell) Funded by NWO, Shell, Nobian Funded % NWO 70% Industry 30% Start of the project 2019 Information Arvind Pari 06 26327072 A.Pari@tudelft.nl	Project title Bubble dynamics in alkaline water electrolysis using the lattice Boltzmann method Project aim This Project aims at developing an open-source lattice Boltzmann model to study the nucleation, pinning, growth, and coalescence of a bubble on an electrode surface, or its detachment from that surface in an electrochemical system. The developed LB model would be capable of simulating multicomponent and multi-phase (gas, liquid and solid) systems, and would be able to handle electric fields and charged components. Simulations can increase our understanding of the limiting processes, and help in optimizing and testing new electrode designs. An important output of the project is a predictive (statistical) correlation for bubble nucleation rates at the electrode surface. Progress An open-source LB porous media code by James McClure (available on GitHub) has been adopted as the starting point for simulation of fluid flow and mass transport in porous media. The code has been extended to account for ion transport under an applied electric field, through the implementation of a Poisson solver for electric potential. In its current state, the code is being used to study the influence of bubble void fraction on the effective conductivity of an electrolyte solution and in turn investigate the validity of the Bruggeman relation in the parametric space (range of bubble void fractions and conductivity of electrolyte solution and bubbles) of interest.

TUD – 3mE – Process & Energy – Complex Fluid Processing	
Project leaders Dr. ir. Remco M. Hartkamp Research theme Complex structures of fluids Participants Max F. Döpke, Johan T. Padding Cooperation Othonas A. Moultos, Johannes Lützenkirchen, Bertrand Siboulet, Jean-François Dufrêche Funded by TUD Funded % University 100 % Start of the project 2018 Information Max F. Döpke m.f.dopke@tudelft.nl	Project title Adsorption and transport phenomena near charged surfaces Project aim Fluid properties within several molecular diameters of charged surfaces are known to be different to those found in the bulk region. Consequently, models that are derived for bulk fluids, such as the Navier-Stokes equations, break down in close proximity of charged surfaces. In this project we aim at shedding light onto fluid properties in this region and expanding existent models to include this non-uniformity of the fluids. Progress In the second leg of this project we scrutinized the commonly used approach of combining different force field parameter sets in Molecular Dynamics simulations of electrolytes and presented our results as reference and best practices guide for future researchers. In the future we aim to further improve simulation models by also scrutinizing fluid-solid interactions and evaluate the impact these have on properties such as dielectric permittivity and viscosity in the interfacial region. 2020 Publications Döpke, M. F., Moultos, O. A., & Hartkamp, R. (2020). On the transferability of ion parameters to the TIP4P/2005 water model using molecular dynamics simulations. The Journal of chemical physics, 152(2), 024501.

TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Dr. ir. H. Burak Eral
Dr. ir. Remco M. Hartkamp
Prof. dr. ir. Johan T. Padding

Research theme
Complex structures of fluids

Participants
Nagaraj Nagalingam

Funded by
NWO-TTW

Funded %
NWO 90%
Industry 10%

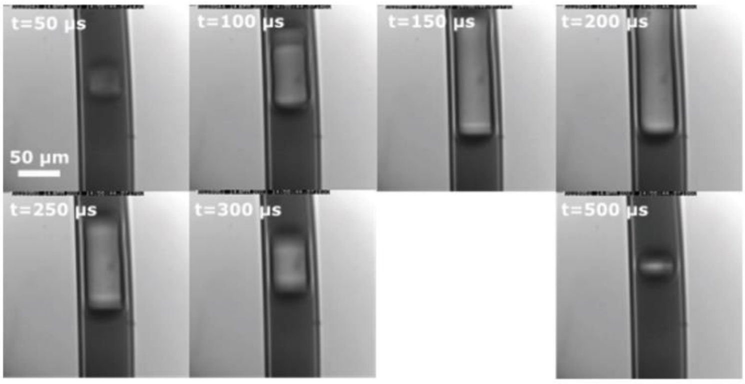
Start of the project
2019

Information
Nagaraj Nagalingam
06 30209925
n.nagalingam@tudelft.nl

Project title
Non-Photochemical Laser-Induced Nucleation (NPLIN)

Project aim
Industrial crystallization, despite its wide-spread use, currently rely on trial and error to ensure desired crystal quality - crystal size distribution, polymorphic form, crystal shape and purity. Over the past two decades, various methods have been proposed to control crystallization process using lasers. Yet, currently existing literature has no consensus on the underlying mechanism. In this work, using numerical simulations and high-speed microscopy experiments, we study the formation of micro-vapor bubble within aqueous solution due to laser irradiation as a precursor to crystal nucleation (known as thermocavitation). This allows us to achieve controlled and predictive production of crystal.

Progress
High-speed microscopy experiments were performed to study the thermocavitation by creating a vapour bubble at the midpoint of a micron size capillary by focusing a 9 ns laser pulse of a 532 nm wavelength (see the figure). Our current experiments focus on the role of laser energy absorbed and system geometry on vapor bubble dynamics, similar to Sun et al (Journal of Fluid Mechanics, 2009). To mathematically represent the expansion and collapse of the vapour bubble under confinement, a simplistic one-dimensional model involving energy balance and momentum balance is being developed. Our future work will extent the current single component model to a multicomponent system focusing on the solute. Using experiment and simulation we will analyse the observed heat and momentum effects over the solute concentration around the vapour liquid interface.



Evolution of vapour bubble inside a circular capillary with an inner diameter 50 μm, recorded with a frame rate of 20 kHz

TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Dr. Lorenzo Botto

Research theme
Complex dynamics of fluids

Participants
Simon Gravelle, Catherine Kamal

Cooperation
Queen Mary University of London

Funded by
European Research Council
(ERC Consolidator Grant)

Funded %
EU 100%

Start of the project
2017

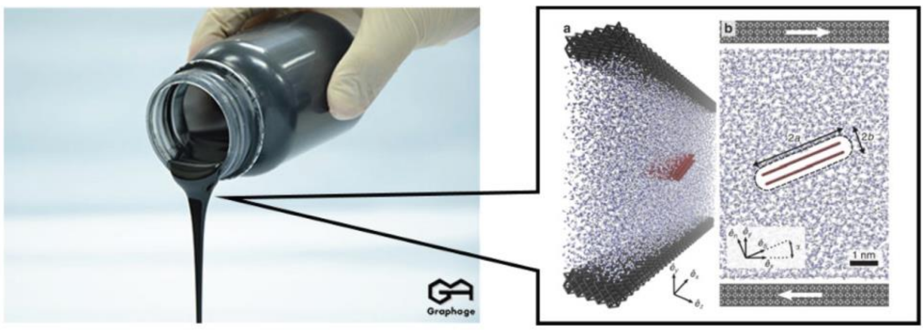
Information
Dr. Lorenzo Botto
l.botto@tudelft.nl

Project title
Graphene hydrodynamics: effect of slip, Brownian fluctuations and bending deformations on the rotational dynamics of thin plate-like colloids

Project aim
What we know about the fluid dynamics of plate-like colloids rests on Jeffery’s theory, which predicts periodic rotations at large Péclet numbers for particles in a shear flow. Does this theory apply to graphene, and the other many 2D nanomaterials? The aim of this project is to combine continuum-level theory and non-equilibrium molecular dynamics to reveal how different, from a fluid mechanics perspective, graphene is from other well-known plate-like colloids (e.g. clays). Graphene has atomic thickness. Its study therefore offers the possibility of exploring an entirely new fluid dynamic regime, in which the fluid molecular scales are comparable to the particle thickness.

Progress
Our molecular dynamics simulations show that rigid graphene platelets suspended in a liquid and subject to a simple shear flow do not rotate, but attain a very specific orientation. The hydrodynamic stress distribution giving rise to this equilibrium orientation has been simulated by boundary integral simulations and analyzed with asymptotic theory. The overarching result of our investigation is that the dynamics of graphene is dominated by hydrodynamic slip at the liquid-solid interface, and that a dramatic change in rotational behaviour occurs when the slip length is comparable to the particle thickness.

2020 Publications
C. Kamal, S. Gravelle, L. Botto. “Hydrodynamic slip can align thin manoplatelets in shear flow.” Nature communications 11.1, 1-10, 2020.
S. Gravelle, C. Kamal, L. Botto “Liquid exfoliation of multilayer graphene: a molecular dynamics investigation“, The Journal of Chemical Physics, 152,104701, 2020.



TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Dr. Lorenzo Botto

Research theme
Complex dynamics of fluids

Participants
Heng Li

Funded by
European Research Council
(ERC Consolidator Grant)

Funded %
EU 100%

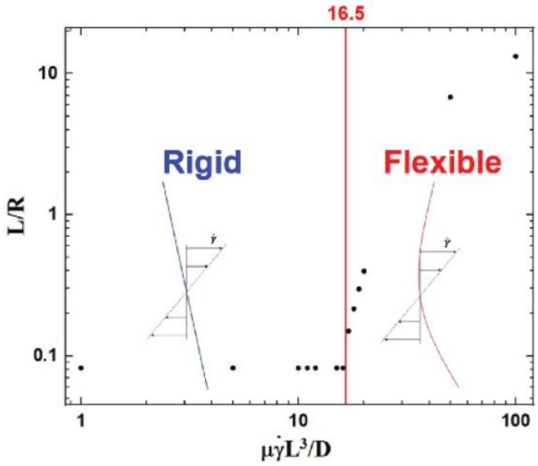
Start of the project
2020

Information
Heng Li
h.li-10@tudelft.nl

Project title
Smart centrifugation of polydispersed suspensions and flexible colloids: simulation of new-generation, low-Reynolds-number dispersed flows

Project aim
The Project aims are to address two problems. One is the theoretical prediction of the particle size evolution during multi-cascade centrifugation, a technique often used to fractionate small micro and nanoparticles such as graphene. This is achieved by comparing one-dimensional mathematical models with continuum Stokes flow simulations. The second is an improved understanding of the role of flexibility in shear flows of flexible plate-like particles, achieved with the use of techniques of the Boundary Integral type. Comparison of simulations with experiments will be done to reach both objectives.

Progress
The evolution of particle size distribution during sedimentation of an initially log-normally distributed system has been predicted, and a two-step centrifugation process has been devised based on these results. The critical non-dimensional shear rate based on the bending rigidity (D) above which the buckling will happen for a single sheet in the shear flow has been identified (see the figure). The interaction of two flexible sheets in shear flows is under investigation.



Numerical predictions for normalised curvature of a single-sheet in simple shear flow as a function of normalised shear rate, showing a distinct buckling threshold

TUD – 3mE – Process & Energy – Complex Fluid Processing

Project leaders
Dr. Lorenzo Botto

Research theme
Complex dynamics of fluids

Participants
Dr. Hugo Perrin

Funded by
European Research Council
(ERC Consolidator Grant)

Funded %
EU 100%

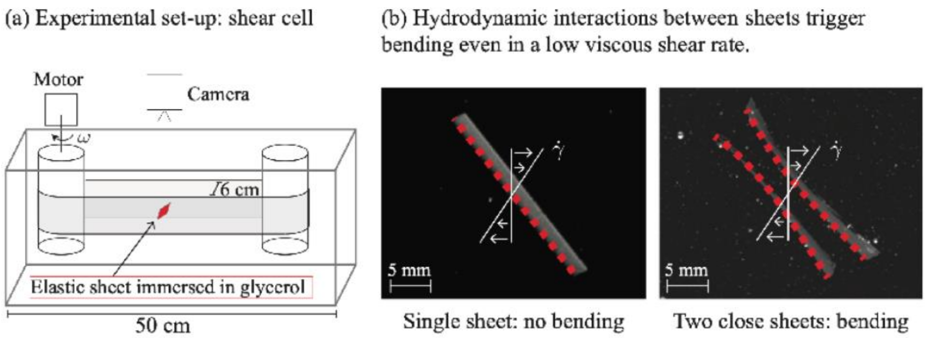
Start of the project
2020

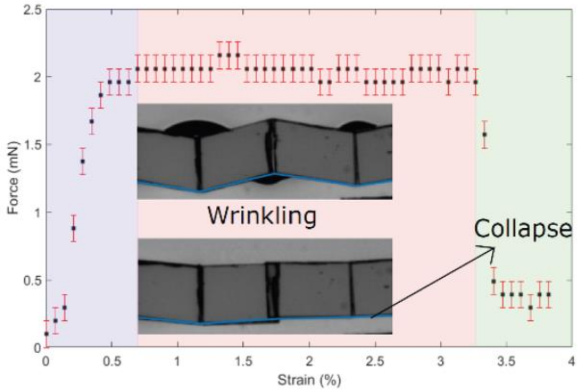
Information
Hugo Perrin
h.r.l.perrin@tudelft.nl

Project title
Elastic sheets in shear flows, from single sheet behaviour to exfoliation process: an experimental study

Project aim
The fabrication of high-quality graphene at industrial scale and affordable price remains a challenge. One of the most promising techniques is so-called liquid-phase exfoliation of graphite. While being poorly understood, the mechanism of exfoliation is known to rely on three physical ingredients: adhesion strength between sheets, their bending rigidity and the large scale imposed viscous shear stress. To investigate the role of the three ingredients, we study experimentally model elastic adhesive sheets in shear flows at the macroscopic scale.

Progress
As a requirement one needs to predict the viscous forces exerted on elastic sheets. We built a shear cell to impose a controlled viscous shear stress on elastic sheets and observed optically (see figure (a)). We studied the deformation of single sheets, and measured the critical shear rate to trigger buckling. Next, we studied the deformation of two close sheets. Our experiments demonstrate that the critical shear rate depends nonmonotonically on the distance between the sheets. As illustrated by figure (b) two close sheets deform while an isolated one remains straight. To rationalize this unexpected observation, we developed a model of the hydrodynamic interactions.



TUD – 3mE – Process & Energy – complex Fluid Processing	
Project leaders Dr. Lorenzo Botto Research theme Complex dynamics of fluids Participants Suriyaprakash Senthil Kumar Cooperation TU Twente, University of Birmingham Funded by European Research Council (ERC Consolidator Grant) & TU Delft department funding Funded % University 50% EU 50% Start of the project 2020 Information Suriyaprakash Senthil Kumar s.senthilkumar@tudelft.nl	Project title Flow physics of plate-like particle suspensions in canonical flows and applications to graphene Project aim Graphene and other 2D materials are nanoscopic objects, whose dynamics is not accessible to direct experimental observation. Therefore, in this project we aim to develop experimental model systems to study the dynamics of 2D materials in canonical interfacial and bulk flows of interest in 2D materials processing. Currently, as mimics for graphene we use thin polyester sheets and elastomers. Later in the project, the insights from the model systems will be applied to experiments with real graphene dispersions. Progress Experimental model systems were developed for studying the buckling dynamics of a monolayer of polyester plates at air-water+glycerol interface. This is of interest to get insight into Langmuir-Blodgett technique to coat surfaces with graphene colloids. A 1D arrangement of plate like particles on the interface under compression shows no change in compressive forces after the out-of-plane deformations starts to appear. A mathematical model is being developed to understand these out-of-plane deformations. Preliminary experiments in extensional flows shows preferred orientations of plates, the hydrodynamic and excluded volume interactions will be studied in the future. Model systems for studying sedimentation of plate type particle suspension are also built and preliminary experiments were carried out.  *Compressive forces on the 1D arrangement of plate like particles plotted against strain, showing three different regimes. The regime where the force increases, a regime with constant force and the regime where force drops as the plate arrangement collapses.*

TUD - Mechanical, Maritime and Materials Engineering (3mE) – Maritime and Transport Technology

Prof.dr.ir. C van Rhee
Prof.dr.ir. TJC van Terwisga

Seakeeping and Manoeuvring

Research of the Chair of Ship motions and manoeuvring is at present focused on the following areas: 1. Non-linear behaviour of fast craft in waves; 2. Extreme wave events in relation to stationary floating structures; 3. Very Large Floating Structures at sea; 4. Prediction of manoeuvring forces based on CFD methods.

High speed ships traveling in waves experience large amplitude motions which can lead to strong non-linear effects in the loads on the hull girder. These non-linear effects are due partly to the large changes in the wetted part of the hull in waves and partly due to non-linear pressure effects. Extreme wave events (breaking waves etc.) can lead to high impact loads on stationary floating structures , e.g. bow loads on Floating Production and Storage vessels for the oil industry. Hydrodynamic analysis of such behaviour is now moving towards application of CFD methods to determine the local flow at the bow. In order to simulate open sea conditions, the CFD region will be connected to an outer region which will be described by potential flow methods. This will allow waves to enter the region local to the bow and reflected waves to travel away from the bow thus minimizing reflections from the CFD boundary. Very Large Floating Structures are being investigated world-wide for various applications such as airports and for floating cities. This research is aimed at developing a novel concept of a large floating structure based on the use of air cushions to support the structure and distribute the wave loads thus optimizing both motion behaviour and structural costs. Manoeuvring models for ships have traditionally been based on equations of motions using experimentally determined drag, mass and lift coefficients. This research aims to investigate the applicability of CFD methods in determining the hydrodynamic coefficients for existing mathematical models. Use is made of a RANS code developed by MARIN and modified to accommodate oblique flow.

Propulsion and Resistance

Research at the Chair of propulsion and resistance is focussed on three areas: 1. Cavitating Flows; 2. Ship-Propeller-Engine system in Service Conditions; 3. Drag reduction through air lubrication.

Cavitation remains an important field of investigation in Marine Technology. Almost all propellers in operation show cavitation in some but mostly in all working conditions. Cavitation often is an important source of vibrations and sometimes even cavitation erosion. Cavitation on propellers should therefore be controlled as much as possible in both the design and during operations. As cavitation often appears to be extremely unsteady and unstable, this poses a continuing challenge to designers and research groups. The more so because an acceptable cavitation control and a high propulsive efficiency are often conflicting requirements. The Delft Cavitation Tunnel has proven to be a strategic tool for enhancing our understanding of the physics and engineering characteristics. There are currently two PhD projects addressing this issued: One on unsteady sheet cavitation (E.J. Foeth) and another on propeller radiated pressure fluctuations (E.v.Wijngaarden). Ship-Propeller-Engine system analysis and simulation is important to reduce e.g. radiated noise from the propeller and to reduce wear of the propulsion system during its operation. To this end, a close cooperation exists with the section of Marine Engineering. There is currently a PhD project concerned with the development of a model podded propeller for testing in operational conditions (G. Oosterhuis), and a recent PhD project on improved propulsion control for Naval Vessels (A.Vrijdag). Initiatives in the third area on Drag reduction through air lubrication are currently under way.

TUD – 3mE – Maritime and Transport Technology

Project leaders
Cees van Rhee
Rudy Helmons

Research theme
Complex dynamics of fluids

Participants
Cees van Rhee
Rudy Helmons
Said Alhaddad
Mohamed Elerian

Cooperation
Royal IHC
TUDelft
RWTHAACHEN universtiy
Royal Netherlands Institute for Sea Research
Spanish National Research Council
Aarhus University (Universiteit van Aarhus)
SEASCAPE Consultants LTD

Funded by
Industry

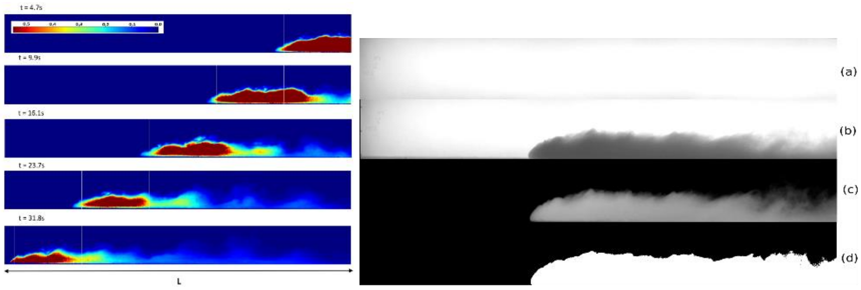
Funded %
Industry 100%

Information
Mohamed Elerian
Telephone number: 0636322299
E-mail: m.f.a.i.elerian@tudelft.nl

Project title
Blue Harvesting project

Project aim
Development and testing of a hydraulic nodule collector for mining of polymetallic nodules. The aim is to significantly reduce the environmental impact generated by the hydraulic collector, mainly the sediment plume.

Progress
Lock-exchange experiment took place in 2020 to investigate the effect of the mining-generated turbidity currents. Different sediment types with different initial conditions were tested.
A new multiphasedriftfluxFoam solver was developed in OpenFOAM that is able to solve to different particle sizes.



2020 Publications
Near-Field Analysis of Turbidity Flows Generated by Polymetallic Nodule Mining Tools
https://repository.tudelft.nl/islandora/object/uuid%3Afa48d9f4-ecbe-4ce4-bb7f-6578f6bcc6cc

TUD – 3mE – Maritime and Transport Technology

Project leaders
Cees van Rhee
Rudy Helmons

Research theme
Complex dynamics of fluids

Funded by
The European Institute of Innovation and Technology, EIT Raw Materials under Project Agreement 18138, Specific Grant Agreement No. [EIT/RAW MATERIALS/SGA2019/1].

Funding %
University 10%
EU 90%

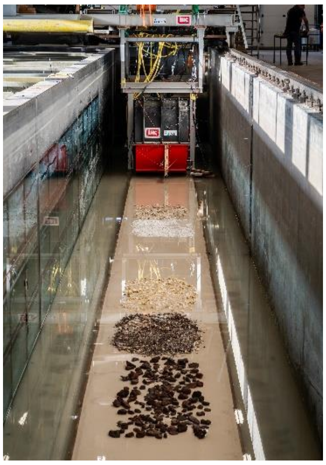
Start of the project
2019

Information
Said Alhaddad
Telephone: +31 (0)15 27 83675
S.M.S.Alhaddad@tudelft.nl

Project title
Blue Harvesting

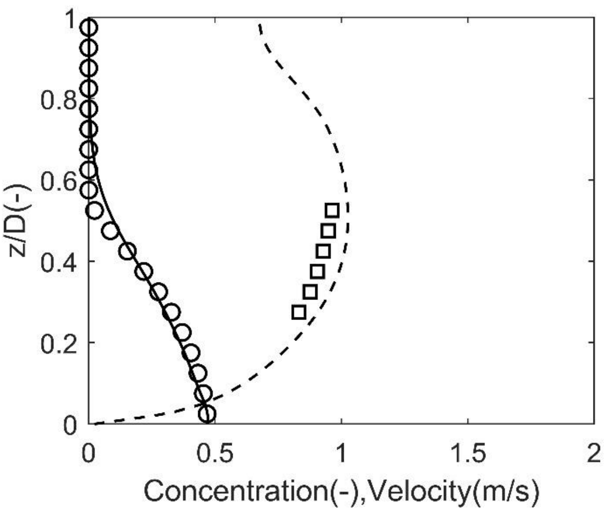
Project aim
Polymetallic nodules are potato-sized rock accretions that form on vast areas of the abyssal plains of the global ocean. These nodules are rich in commercially precious metals, such as nickel, cobalt and copper, making them a target for potential future deep-sea exploitation. Among the existing mechanisms for mining polymetallic nodules (mechanical, hydraulic and hybrid), hydraulic collecting is deemed the most suitable technology in deep sea mining. The main aim of our study is to design a hydraulic collector with an optimal production efficiency while preserving a minimal environmental disturbance.

Progress
Full-scale experiments have been designed to test the functionality and productivity of the designed collector while avoiding having scale effects. A set of full-scale experiments has been conducted at Deltares to obtain a deeper insight into the production efficiency of the hydraulic collector under different operational conditions and to develop a better understanding of the underlying physics. The experimental results confirmed the functionality of the designed collector and pave the way for a more advanced design in the future.



The experimental setup

TUD – 3mE – Maritime and Transport Technology	
Project leaders Cees van Rhee, Sape Miedema Research theme Complex dynamics of fluids Participants Xiuhan Chen Cooperation NWO Seatools B.V. Tree-C Technology B.V. Funded by NWO Funding % NWO 100% Start of the project 2012 Information Xiuhan Chen x.chen-1@tudelft.nl	Project title Numerical Modelling for Underwater Excavation Process Project aim In the underwater excavation process of dredging engineering, cutting force is greatly influenced by the local water pressure especially in deep water. The fluid flow will change the pore pressure distribution and meanwhile create certain forces to the solid particles. Since it is expensive to conduct experiments to measure the cutting force, a numerical model is then needed to describe the physics in it. This project will build up the numerical model of underwater excavation process for sand, clay and rock. Progress The numerical model has been fully established, now capable of simulating the underwater sand, clay and rock cutting tests. Many scenarios are now running for building up a data base, for the material properties, and also for the cutting process. For writing, the draft PhD dissertation has been completed. 2020 Publications Echaroj, S., Ong, H.C., Chen, X., Simulation of Mixing Intensity Profile for Bioethanol Production via Two-Step Fermentation in an Unbaffled Agitator Reactor, Energies, 13, 54-57 (2020) Chen, X., Wang, X., Miedema, S.A., van Rhee, C., Introduction to Deep Sea Mining and the Relevance to Dredging Engineering V: The Cutting Process, China Dredging, 48, 22-33 (2020). Miedema, S.A., Chen, X., Nieuwboer, B.J., Cutter Head Spillage When Dredging Sand or Gravel, China Dredging, 48, 34 - 44 (2020). Miedema, S.A., Wang, F., Hong, G. & Chen, X., Slurry Transport in Inclined Pipes, China Dredging, 51, 45-54 (2020).

TUD – 3mE – Maritime and Transport Technology	
Project leaders Dr. G.K. Keetels Research theme Complex dynamics of fluids Participants Dr. G.K. Keetels Prof. C. van Rhee Dr. W.P. Breugem Ir. T. D. Schouten Cooperation Royal Boskalis Westminster, van Oord Funded by Stichting Speurwerk Baggertechniek (SSB) NWO Funding % NWO 80% Industry 20% Start of the project 2017 Information TD Schouten t.d.schouten@tudelft.nl	Project title Modeling of turbulent hyper concentrated sediment transport in horizontal pipes Project aim In dredging applications typically large amounts of sediments are transported through pipes in the form of hyperconcentrated (40% sediment or more) sediment-water mixtures. These slurry's can flow at three different regimes. 1: fully suspended. 2: partially suspended with a sliding bed. 3: partially suspended with a fixed bed. At the moment it is hard to predict the transport regime, the volume flux of particles and the pressure drop (friction factor) of these slurry's within these regimes. The goal is to establish a 3D continuum model that is able to predict the aforementioned aspects of slurry flow in a wide range of slurry flow conditions. Progress A two-phase Euler-Euler model has been evaluated for all regimes that can be present during the transport of slurry through horizontal pipelines. The model has been compared to laboratory experiments for different types of sediment. It has also been compared with available literature on slurry transport. The model predicts characteristics such as hydraulic gradient, flow velocities and concentration distribution within an error margin of 10%. It also predicts the presence of beds, either fixed or sliding, correctly.  *Concentration (o: experimental, - Computed) and velocity(□ experimental, -- Computed) distribution for suspended 150 µm glass beads.*

TUD – 3mE – Maritime and Transport Technology

Project leaders Cees van Rhee (TUD) Geert Keetels (TUD) Serge Toxopeus (MARIN) Research theme Complex dynamics of fluids Funded by NWO Port of Rotterdam MARIN Funding % NWO 75% Industry 25% Start of the project 2017 Information Stefano Lovato +393471192661 s.l.lovato@tudelft.nl	Project title Sailing through fluid mud Project aim The navigation of ships in restricted areas is often influenced by the presence of mud layers forming on the seabed. The Project aims at developing a computational fluid dynamics (CFD) model to study the effects of these mud layers on the manoeuvrability of ships. Progress One of the main challenges of developing a CFD model to study the effects of mud on ships is that mud has a complex non-Newtonian rheology. We have performed flume experiments in which a plate was towed through mud collected in the port of Rotterdam (see picture below). The plate's resistance computed using the Bingham model as constitute equation for mud showed a fairly good agreement with the experimental data. Thus, despite the complex rheological behaviour of mud, this research suggested that the simple Bingham model may be sufficiently accurate for the purpose of this project.  2020 Publications Lovato, S., Toxopeus, S.L., Settels, J.W., Keetels, G.H. and Vaz, G., 2021. Code Verification of Non-Newtonian Fluid Solvers for Single-and Two-Phase Laminar Flows. Journal of Verification, Validation and Uncertainty Quantification, 6(2), p.021002.

TUD – 3mE – Maritime and Transport Technology	
Project leaders Prof. dr. ir. Cees van Rhee Research theme Mathematical and computational methods for fluid flow analysis Participants Ir. Edwin de Hoog Dr. ir. Arno Talmon Prof. dr. ir. Cees van Rhee Cooperation Royal IHC Funded by Royal IHC, TKI Maritiem Funding % University 40 % Industry 60 % Start of the project 2019 Information Name: Edwin de Hoog E-mail: e.dehoog@royalihc.com	Project title Transient hydraulic transport Project aim Study the effect to of transients on the stability of hydraulic transport pipelines, used in the dredging and deep sea mining industry. The sediment transported through these pipelines can accumulate under certain conditions (density wave amplification), creating an unstable pipeline which can become blocked. The goal of this project is to investigate which processes attribute to destabilizing the transport process, and to predict these transient processes with 1D Driftflux modeling. Progress The year 2020 was dedicated to a literature study to identify key processes which destabilize the hydraulic transport process, and the numerical techniques needed to model these processes. Part of the literature study was submitted as a publication to the Journal of Hydraulic Engineering.

TUD – 3mE – Maritime and Transport Technology	
<u>Project leaders</u> dr. ir. I.Akkerman, dr. ing. S.Schreier <u>Research theme</u> Mathematical and computational methods for fluid flow analysis <u>Funded by</u> NWO, Stichting Bijboegfonds, Damen; Shipyards, Flying Fish Mobility; MARIN <u>Funded %</u> NWO 80% Industry 15% GTI 5% <u>Start of the project</u> 2019 <u>Information</u> Ido Akkerman I.Akkerman@tudelft.nl	<u>Project title</u> Lift control for hydrofoil craft <u>Project aim</u> Hydrofoil craft provide fast and economical transport, in particular in dense urban areas such as the Netherlands. However, the challenge is to maintain comfortable and safe navigation, especially in presence of waves, while maximising robustness and minimising capital and maintenance cost. To address these issues, we investigate in this project novel approaches for hydrofoil lift control and height control, where we use concepts from flight dynamics to study unsteady motions, such as the phugoid mode. For lift control we consider proven aeronautical mechanisms, in addition to novel concepts specific to hydrofoils. <u>Progress</u> The stability envelope and control design mechanisms are explored with a state-space model (SSM), with input from numerical and experimental data. Numerical simulations (PhD-2) focus on potential flow codes in combination with CFD, using finite-element methods and divergence-conforming isogeometric analysis (IGA). Special attention is on interaction of the hydrofoil flow with the free-surface in combination with waves, using turbulence modelling with residual-based variational multi-scale physics and a level-set approach to describe the free-surface. Surface tension is included to accurately predict air injection, ventilation and broaching. Experimental methods (PhD- 1) will be used to provide input to the state-space model, and also validate the numerical results, employing a combination of flow visualization for the overall flow, and stereoscopic particle image velocimetry (PIV) for detailed quantitative whole-field measurements of the flow. Experiments are carried out at TU Delft in the towing tank and newly commissioned multiphase flow tunnel. Furthermore, reduced-order models (ROMs) will be explored to obtain fast and accurate means to exploit the numerical results for utilisation towards the design of new generation hydrofoil craft. A post- doc will work with our industrial and technological partners to implement this in a demonstrator. We expect that this research will lead to new and validated hydrofoil concepts with active lift and height control.

TUD – 3mE – Maritime and Transport Technology

Project leader
Dr. ir. I. Akkerman

Research theme
Complex dynamics of fluids

Participants
dr. ing. S.Schreier, dr. G. Jacobi MSc

Cooperation
Damen Shipyards

Funded by
Stichting Bijboegfonds, TKI

Funded%
NWO 50%
Industry 50%

Start of the project
2019

Information
Ido Akkerman
I.Akkerman@tudelft.nl

Project title
Hydrofoil education and research platform: HE ARP

Project aim
Hydrofoil craft can provide fast and comfortable transportation of people. By exploiting existing waterways (“Blauwe Route”) at a relatively high-speed hydrofoil craft fill a unique niche between standard ships (slow), landbased transport such as cars (cannot use Blauwe Route”) and air-transport such as helicopters (expensive). This transportation niche is particular interesting in dense urban areas such as the Netherlands, but is also relevant in offshore crew supply and naval operations. Apart from the advantages provided by hydrofoil craft there are also some issues that limit its current use. The challenge is to maintain comfortable and safe navigation, especially in presence of waves, in a cost-effective manner. Current solutions are to complex and fragile leading to high capital and maintenance costs. To address this issue, and make hydrofoil craft economically feasible, the TU Delft ship hydrodynamics lab has started a research line investigating alternative hydrofoil craft and lift control concepts. The latter, alternative lift control, is the topic of “Lift control for hydrofoil craft” project.

Progress
A top-level concept has been selected. The platform will have full-rake control of the foils allowing the entire foil to be replaced with alternative designs. Below is an artist impression of the platform.

TUD – 3mE – Maritime and Transport Technology

Project leader
Dr.-Ing. Sebastian Schreier

Research theme
Complex dynamics of fluids

Participants
M. Zhang (TU Delft)

Funded by
Scholarship CSC

Funded %
Scholarship 100%

Start of the project
2018

Information
Sebastian Schreier
s.schreier@tudelft..nl

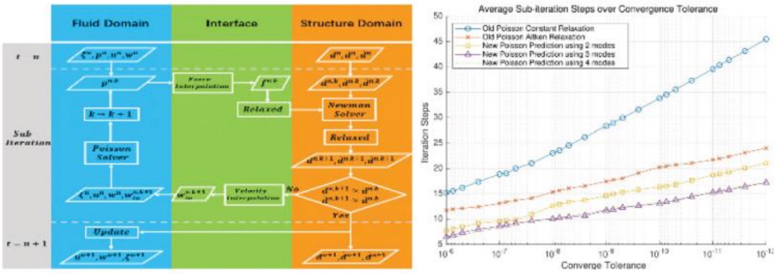
Project title
Wave scattering of very flexible floating structures

Project aim
The aim of the project is to determine the wave scattering properties of Very Flexible Floating Structures, which are able to follow the wave surface elevation almost perfectly due to their low structural bending stiffness.

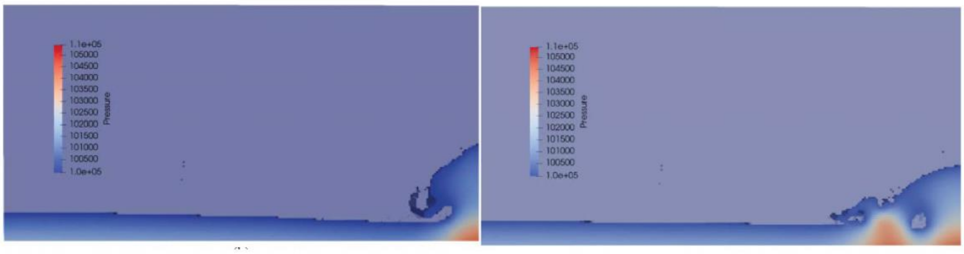
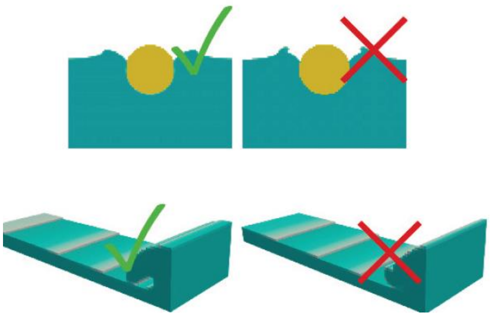
Progress
Based on a literature review on hydroelasticity a suitable theoretical approach to model structural response and influence on the surrounding wave of VFFS was selected. In a preliminary experiment in the towing tank, Digital Image Correlation (DIC) measurements of a VFFS provided a first set of reference data.

Model of VFFS with speckle pattern for DIC measurements in the towing tank of TU Delft. Wave propagating from right to left

TUD – 3mE – Maritime and Transport Technology	
Project leaders Dr. Ir. P.R. Wellens Research theme Mathematical and computational methods for fluid flow analysis Participants Ir. Pengpeng Xu Funded by China Scholarship Council and Delft University of Technology Funded % University 25% Scholarship 75% Start of the project 2017 Information Pengpeng Xu pengpeng.xu@tudelft.nl	Project title Fluid-structure interaction in extreme impacts of water waves Project aim This research is to develop the computational algorithm to reduce the expense of the numerical computation. In the partitioned approach of computational hydroelastics, fluid and structure are computed in each domain respectively. Then two solutions are coupled through the interface, where sub-iterations are necessary. The sub-iterations are usually expensive. In this research, we propose to include few dominating eigenmodes of the structure in the Poisson equation that solves fluid pressures, which is termed as the interaction law. The iteration is reduced because the structure information is predicted and included in the fluid domain computation. Progress A fluid-structure interaction speed-up algorithm is proposed, see the figure. Only the dominating, not all, structure information is modeled as added mass effect and included in the Poisson operator.



The numerical test of rubber-barge in waves shows that the interaction law with only three eigenmodes, i.e., heave, pitch and first order bending can reduce the coupling error by one or two magnitudes in the first sub-iteration. Therefore, the necessary number of sub-steps is reduced.

TUD – 3mE – Maritime and Transport Technology	
Project leaders Dr. Ir. P.R. Wellens (promotor Prof. Dr. Ir. J.J. Hopman) Research theme Mathematical and computational methods for fluid flow analysis Participant Ir. M. van der Eijk Funded by Delft University of Technology Funded % University 100% Start of the project 2019 Information Martin van der Eijk m.vandereijk@tudelft.nl	Project title Extreme impacts of water waves with entrained air on structures Project aim Investigating the effect of aerated water on the time course of the impact pressure caused by free surface events, i.e. freely moving geometries like ships encountering green water or slamming. This will be done by a self-developed novel numerical method. The method needs to be capable to resolve air entrainment during an impact. Progress The relevance of compressibility of air and surface tension during a wave impact in which an air pocket gets entrapped is investigated. At model scale in the 2D dam-break simulation, the effect of compression wave in the air dominates the dynamics. The compression waves and subsequent pressure oscillations were in the same order of magnitude as the initial impact. The inconsistent mass-momentum coupling leading to unphysical free surface distortions as in the first figure is solved. A flux-corrected scheme is used for the density to ensure kinetic energy conservation in high density flows. Furthermore, rigid-body motions are implemented using the same algorithms as for the free-surface flow for the displacement.  *Pressure during entrapment of air pocket by a dam-break*  *Consistent versus non consistent coupling of mass and momentum*

TUD – Chemical Engineering – Transport Phenomena

Prof.dr.ir. CR Kleijn
Prof.dr.ir. HEA van den Akker
Prof.dr.ir. K Hanjalic (em)
Prof.dr. S Kenjeres, Dipl.-Ing

The Transport Phenomena group studies the transport of mass, momentum and heat, on different length and time scales, in physical, biological and phone chemical processes related to advanced materials processing, energy conversion and storage, and health. The main interest is in transport phenomena around (solid-fluid, liquid-gas and liquid-liquid) interfaces, which we wish to understand, control and enhance. The group uses both theoretical and computational models, and nonintrusive experiments based on laser and X-ray techniques. Our expertise is in heat and mass transfer in multiphase flows, turbulent flows, microflows and biological flows.

A partial list of topics which we currently work on:

- Multiphase flow and dynamic contact line phenomena in digital microfluidics and Labs-on-Chips
- Dispersed multiphase flows in large scale chemical processing (bubble columns, fluidized beds, Fischer Tropsch)
- Magnetohydrodynamics in advanced liquid metal processing (welding, casting)
- Magnetic drug targeting
- Oil-water separation
- Turbulence modulation for enhanced heat and mass transfer

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof. dr. ir. C.R. Kleijn
Prof.dr. S. Kenjeres, Dipl.-Ing

Research theme
Complex dynamics of fluids

Participants
A. Blishchik (PhD Student)

Cooperation
TOMOCON project

Funded by
EU Horizon 2020/TU Delft

Funded %
University 25 %
EU 75 %

Start of the project
2018

Information
Artem Blishchik
06 18508176
A.Blishchik@tudelft.nl

Project title
Advanced simulation of liquid melt flows in controlled continuous casting

Project aim
Steel production is an energy intensive process. Continuous casting of steel (metals in general) aims at high production speed at as low as possible energy usage. Stability of the flow in the casting process plays a vital role in product quality and process performance. Electromagnetic actuators can be used to control and stabilize the multiphase metal flow. Computational Fluid Dynamics (CFD) can play a vital role in understanding the interplay between fluid dynamics and electromagnetic forces. Moreover, it allows studying different actuator strategies. The work also involves fundamental scientific analyses and engineering design with groups at Helmholtz-Zentrum Dresden-Rossendorf, at the universities of Liberec and Bath and at different industry partners.

Progress
We have finished and validated our in-house solver capable to simulate various magnetohydrodynamic flows, including multiphase, heat transfer, and turbulence phenomena. Later, we discovered a new flow regime where the initially turbulent MHD flow in a duct can be suppressed or promoted depending on the electrical conductivity of surrounding walls (i.e., Cd wall conductivity ratio), see the figure. Moreover, we investigated how the flow in a continuous casting mold behaves while it is exposed to the presence of arbitrary electrically conductive walls.

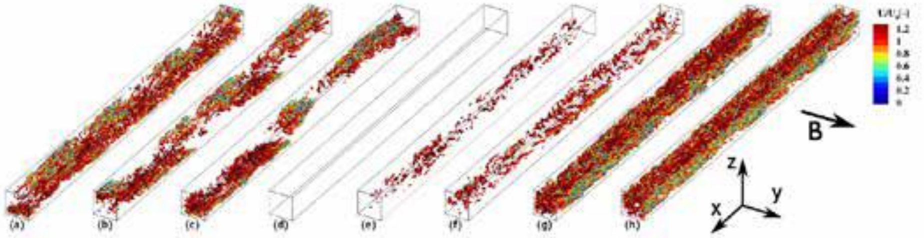


FIG. 1: The instantaneous coherent structures in the MHD duct flow identified as isosurfaces of the second-invariant of the velocity-gradient tensor (Q-criterion, Q=1 s^{-2}), colored with the streamwise velocity at Re=5602 and Ha=21.2. (a) $C_d = 0$, (b) $C_d = 0{,}05$, (c) $C_d = 0.1$, (d) $C_d = 0.15$, (e) $C_d = 0.25$, (f) $C_d = 0.5$, (g) $C_d = 1.5$, (h) $C_d \to \infty$.

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres, Dipl.-Ing, Prof. Dr. H. J. Lamb (LUMC)

Research theme
Complex dynamics of fluids

Participants
Xiaolin Wu

Cooperation
Amsterdam UMC, Leiden University Medical Centre

Funded by
TU Delft, China Scholarship Council

Funded %
University 50 %
Scholarships 50 %

Start of the project
2018

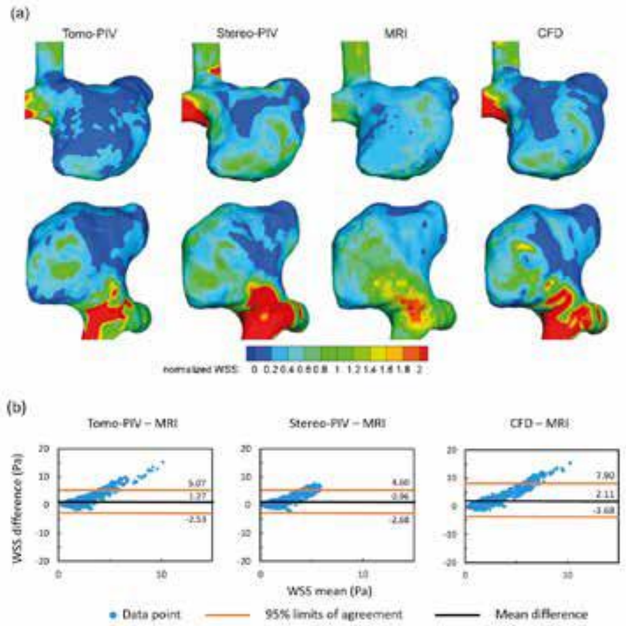
Information
Xiaolin Wu
06 3335 6690
X.Wu-5@tudelft.nl

Project title
Hemodynamic study of a patient-specific intracranial aneurysm: comparative assessment of tomographic PIV, stereoscopic PIV, in vivo MRI and computational fluid dynamics

Project title

Project aim
The objective of this research is to experimentally investigate the instantaneous 3D flow and wall shear stress (WSS) within the patient-specific intracranial aneurysm model by Stereoscopic PIV/Tomographic PIV techniques. This work can provide validation to the developed numerical models and medical imaging techniques by performing multi-modality comparison among aforementioned PIV, CFD and 4D Flow MRI.

Progress
A patient-specific intracranial aneurysm underwent examination by 4D Flow 7T MRI scans at the Academic Medical Center in Amsterdam. Tomographic PIV, Stereoscopic PIV and CFD hemodynamic studies were then carried out based on the reconstructed geometry and boundary conditions of MRI measurements. Velocity field, vorticity, and WSS results obtained by all modalities were compared.



TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres

Research theme
Complex dynamics of fluids

Participants
Dr. S.S.Khalafvand
Prof.dr. S. Kenjeres, Dipl.-Ing

Cooperation
Erasmus MC Rotterdam,
LUMC Leiden, UMC Maastricht

Funded by
TU Delft, ZonMw: The Netherlands Organisation for Health Research and Development

Funded %
NWO 80 %
Industry 20 %

Start of the project
2015

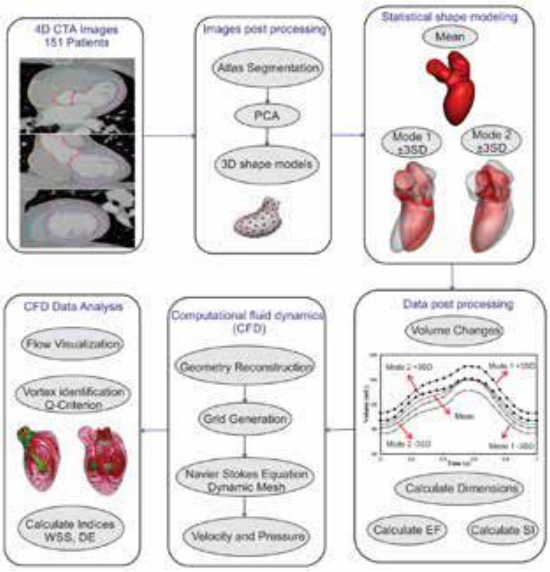
Information
S.Kenjeres@tudelft.nl

Project title
Numerical modelling of 4D left ventricular blood flow using computed tomography imaging and computational fluid dynamics

Project aim
This study is aimed to quantify the characteristics of flow patterns in left ventricle (LV) via Computational Fluid Dynamics (CFD) and an Active Shape Model (ASM) derived from 4-D Computed Tomography (CT) images for 151 patients. Five characteristic 4D shape sets were generated from ASM model: the mean shape, and mode variations of +3 and -3 standard deviations (SD) along the first and second principal component of shape variation in the population. The vortex development pattern during cardiac cycle is investigated through these 5 shapes.

Progress
We have reconstructed the LV geometry for five shapes. For each segmented geometry, separate unstructured grids consisting of tetrahedral cells were generated. The time step resolution of CT data was not fine enough for CFD simulation. Thus intermediate geometries were needed to volume smoothly passed through time steps. For solving the fluid flow domain with finite volume method, the Arbitrary Lagrangian-Eulerian (ALE) formulation of Navier-Stokes was used. The results are analyzed and compared for five characteristics shapes. Novel experimental setup was design to perform combined ultrasound / tomographic PIV measurements.

2020 Publications
Voorneveld J., Saaid H., Schinkel C., Radeljic N., Lippe B., Gijsen F.J.H., van der Steen A. F. W., de Jong N., Claessens T., Vos H. J., Kenjeres S., Bosch J. G. (2020), “4D echo-particle image velocimetry in a left ventricular phantom”, Ultrasound in Medicine & Biology Vol.46 (3), pp.805-817 (doi: https://doi.org/10.1016/j.ultrasmedbio.2019.11.020)



The overall methodology from CT images to blood flow visualization in LV

TUD – Chemical Engineering – Transport Phenomena

<u>Project leaders</u>
Prof.dr. S. Kenjeres, Dipl.-Ing
Prof. Dr. Hildo J. Lamb

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Ir. Romana Perinajova

<u>Cooperation</u>
LUMC

<u>Funded by</u>
Hartstichting

<u>Funded %</u>
Industry 100 %

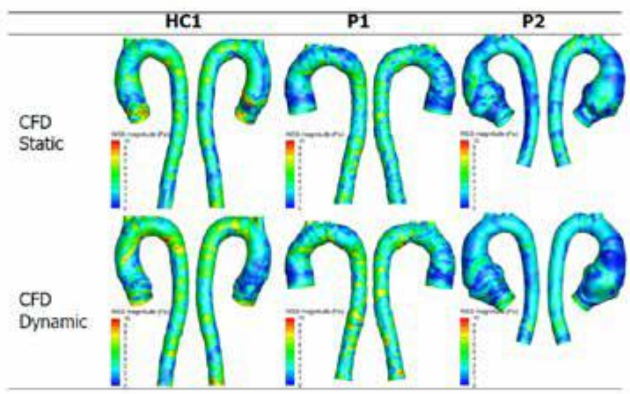
<u>Start of the project</u>
2017

<u>Information</u>
Romana Perinajova
06 14782314
R.Perinajova-1@tudelft.nl

<u>Project title</u>
RADAR – Early Recognition of Aorta Dissection and Aneurysm Rupture

<u>Project aim</u>
An aortic aneurysm is a life-threatening disease. There is an urgent need for new biomarkers that could contribute to an earlier aneurysm detection, preferably before growth, to prevent fatal rupture. The aim of the project is to develop a multi-physics model (fluid-solid-growth) of the whole human aorta that would provide the structural (MRI) and mechanical information of the aortic wall as well as hemodynamic information, transmural pressure, and oxygen concentration. Afterward, with LUMC, define sex-specific physiological normal values and cut-off values for identifying pathological abnormalities based on new and existing data from 10-year follow-up (200 cases).

<u>Progress</u>
A novel method for simulating aortic wall movement has been developed and implemented. With this method, the movement is prescribed using the MRI data. We extract the geometry from MRI at 4 different time-steps and subsequently interpolate the movement for the whole cycle using Radial Basis Function. Simulations were performed on a test dataset consisting of one volunteer and two patients with an identified thoracic aortic aneurysm. In comparison to the simulations with rigid wall assumption, velocity, wall shear stress, as well as oscillatory shear index were affected by the movement. This shows that wall movement is an important factor for simulating aorta.



Contours of wall shear stress (Pa) for Healthy Control (HC1), Patient 1 (P1), and Patient 2 (P2) from CFD static and dynamic at early diastole

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres

Research theme
Complex dynamics of fluids

Participants
Fei Xu

Cooperation
Erasmus Medical Center, Leiden University Medical Center, University of Ghent

Funded by
ZonMw, The Netherlands Organisation for Health Research & Development, China Scholarship Council (CSC)

Funded %
Industry 100 %

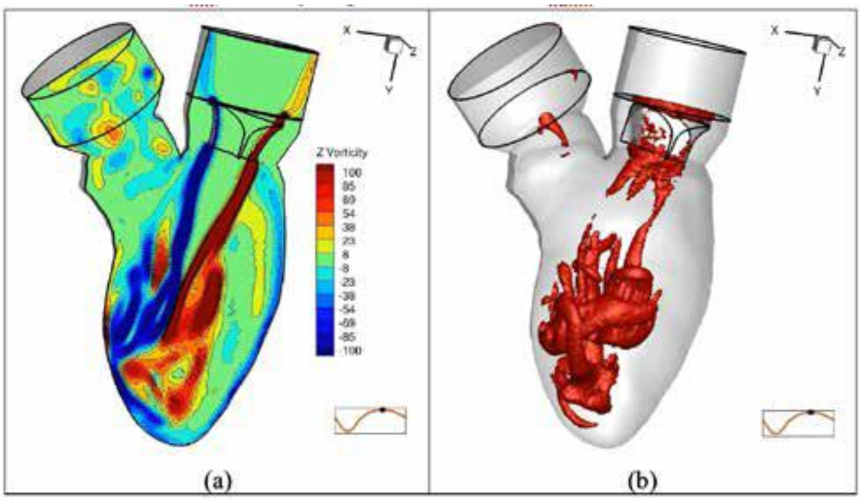
Start of the project
2016

Information
Fei Xu
06 59544684
f.xu-1@tudelft.nl

Project title
Numerical modeling and computer simulations of advanced transport phenomena in bio-medical applications

Project aim
The aim of the project is to provide a fundamental understanding of the hemodynamics inside human heart by means of MRI/Ultrasound based CFD simulations. The MRI based CFD simulations have the capability to reconstruct patient-specific simulation models of the heart. Such models can provide us a deeper understanding of the complex flow phenomenon and provide us with flow details which cannot be supplied by medical imaging alone.

Progress
A novel, cost-effective simulation method for patient-specific heart ventricles is developed. Taking the advantage of the radial basis functions (RBF), the method is able to implement CFD analysis of the left ventricle from medical scans (CT, MRI). The simulation results have been compared with available tomographic PIV as well as MRI measurements. The obtained results provided detailed insights into energetics of the instantaneous flow features of the left ventricle model. The presented method can be applied for future analysis with the patient-specific geometries.



(a) Vorticity component at the peak flowrate on the central plane; (b)Vorticity structure at the peak flowrate by Q criteria
(b)

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres, Dipl.-Ing

Research theme
Complex dynamics of fluids

Participants
Dr. S. Kenjeres, Prof. J. S. Szmyd, Dr. E. Fornalik-Wajs

Cooperation
AGH University of Science and Technology, Krakow, Poland

Funded by
EC Marie Curie, TU Delft, AGH Krakow

Funded %
University 50%
EU 50%

Start of the project
2010

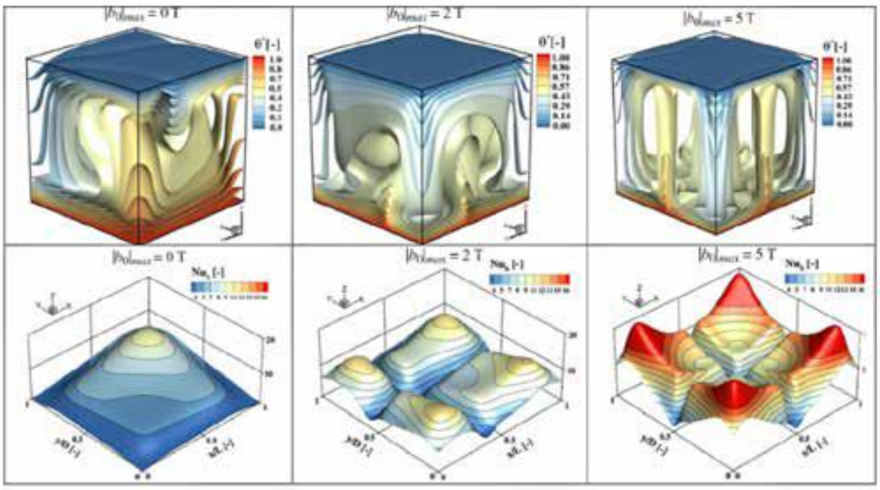
Information
S. Kenjeres
s.kenjeres@tudelft.nl

Project title
Thermo-magnetic convection in the presence of the strong magnetic field gradients: experimental and numerical studies

Project aim
This is a joint project between Transport Phenomena Section, Department of Chemical Engineering at the TU Delft and the Department of Fundamental Research in Energy Engineering, Faculty of Energy and Fuels, AGH University of Science and Technology, Krakow, Poland. The project is aimed at fundamental investigations of flow stability and wall heat transfer of paramagnetic fluids in presence of strong magnetic field gradients.

Progress
Experimental measurements of the integral heat transfer performed over a range of working parameters for a differentially heated cubical enclosure of a paramagnetic fluid subjected to magnetic gradients of different orientation and strength. The DNS studies of the flow and heat transfer were performed and compared to recent experimental data. Detailed insights into the mechanism of the wall-heat transfer enhancement or suppression were analyzed. Finally, the energy budgets were analyzed with the specific focus on the role of the magnetization production term.

2020 Publications
Kenjeres S., Fornalik-Wajs E., Wrobel W., Szmyd J. S. (2020), "Inversion of flow and heat transfer of the paramagnetic fluid in a differentially heated cube", Int. J. Heat and Mass Transfer Vol.151, 119407, pp.1-14 (doi: https:// doi.org/10.1016/j.ijheatmasstransfer.2020.119407).



Top- the iso-surfaces of the temperature for different strengths of the imposed magnetic field, |b0|max = 0, 2, 5 T and a fixed value of Pr = 400 for a cubical enclosure filled with the paramagnetic fluid. Bottom- distributions of the local Nusselt number at the lower (hot) wall. The characteristic value of thermal Rayleigh number for the neutral case is RaT= 2x105, Kenjeres (2016).

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres, Dipl.-Ing

Research theme
Complex dynamics of fluids

Participants
Prof.dr. S. Kenjeres, Dipl.-Ing

Cooperation
Erasmus Medical Center Rotterdam
Leiden University Medical Center

Funded by
TU Delft, ZonMw

Funded %
University 100 %

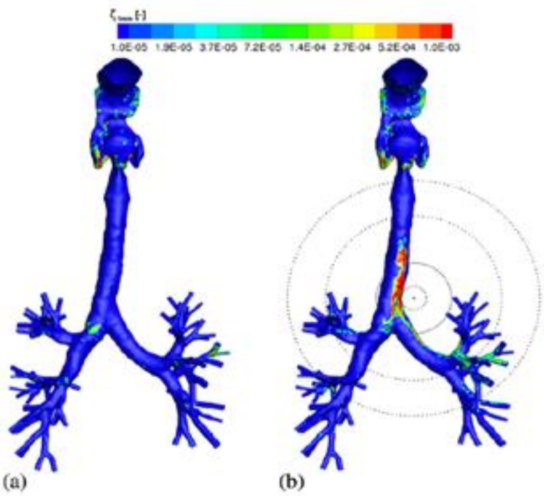
Start of the project
2014

Information
S. Kenjeres
s.kenjeres@tudelft.nl

Project title
Numerical modeling, simulations and experiments of blood and air flow with magnetic particles in simplified and realistic patient arterial and airway geometries: Towards optimized magnetic drug delivery

Project aim
One of the main problems of chemotherapy is often not the lack of efficient drugs, but the inability to precisely deliver and concentrate these drugs in affected areas. Failure to provide localized targeting results in an increase of toxic effects on neighbouring organs and tissues. One promising method to accomplish precise targeting is magnetic drug delivery. Here, a drug is bound to a magnetic compound injected into the blood stream. The targeted areas are subjected to an external magnetic field that is able to affect the blood stream by reducing its flow rate. We believe that mathematical modeling and numerical simulations can significantly contribute to further advancements of this technique.

Progress
We have developed a comprehensive mathematical model for simulations of a blood-flow under a presence of the strong non-uniform magnetic fields. The model consists of a set of Navier-Stokes equations accounting for the Lorentz and magnetization forces, and a simplified set of Maxwell's equations (Biot-Savart/Ampere's law) for treating the imposed magnetic fields. The model is validated for different patient-specific geometries (including a carotid artery and brain vascular system). First simulations of aerosol distribution within a human upper airway system were performed demonstrating that identical tageting concept can be applied for the upper and central human airway system too.



Contours of the local deposition efficiency (ζ) in the upper- and central airways human respiratory system for the magnetic-core particles (dp = 3μm, St=4.3 × 10−2, Mnp=1.7 × 10−1) without (a) and with imposed magnetic field gradient (b), demonstrating potentials of the MDT, Kenjeres (2016)

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres, Dipl.-Ing

Research theme
Complex dynamics of fluids

Participants
Dr. Sasa Kenjeres, Dr. Nikola Mirkov, Dr. Daoming Liu, Dr. Patrick Schrijvers

Cooperation
University of Belgrade and Vinca Institute of Nuclear Science, Belgrade, Serbia, Safety and Emergency Laboratory, Shanghai Advanced Research Institute, Chinese Academy of Sciences, Shanghai, China

Funded by
TU Delft, CSC China

Funded %
University 50 %
Scholarships 50 %

Start of the project
2010

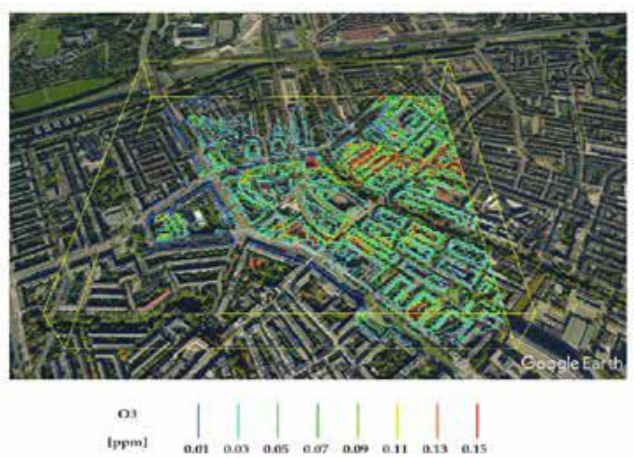
Information
S. Kenjeres
s.kenjeres@tudelft.nl

Project title
Hybrid RANS /LES simulations of turbulent flows over hills and complex urban areas with dispersion of pollutants

Project aim
This project is aimed at the mathematical modelling and numerical simulations of environmental flows, turbulent dispersion of passive and reactive scalars, and the energy balance in the complex urban areas. In this particular project we focus our investigation at turbulent flows over complex terrains and urban areas (street canyons) partially covered with vegetation and with different sources of the passive or reactive scalars. In the last year, the special focus was on the dynamics of reactive scalars to mimic the ozone generation or depletion (the extended GRS-generic reaction set model was applied to model atmospheric chemistry).

Progress
We developed a new class of the seamless hybrid RANS/LES approach suitable for the complex urban areas partially covered with vegetation. We successfully reproduced detailed laboratory scale measurements for different geometrical configurations reported in literature. Also, the mechanism of the ozone generation/depletion in urban areas due to traffic emission is validated.

2020 Publications
Schrijvers P.J.C., Jonker H.J.J., de Rode S. R., Kenjeres S. (2020), "On the daytime micro-climatic conditions inside an idealized 2D urban canyon", Building and Environment Vol. 167, 106427, pp.1-13 (doi: https://doi.org/10.1016/j.buildenv.2019.106427).



The mapping of pollution by iso-lines: concentration contours of O3 (indicating locations with enhanced ozone distribution as result of the chemical reactions) – in the horizontal plane at pedestrian level (z=2 m). Results of the integrated in-house developed CFD/CRD solver for the photochemical smog generation for a city of Rotterdam, Liu and Kenjeres (2017)

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres

Research theme
Complex dynamics of fluids

Participants
Ir. Katrien Van Tichelen (SCK•CEN)
Ir. Steven Keijers (SCK•CEN)
Bouke Kaaks (MSc - TU Delft)

Cooperation
SCK•CEN, VKI

Funded by
ENGIE, SCK•CEN

Funded %
University 100 %

Start of the project
2020

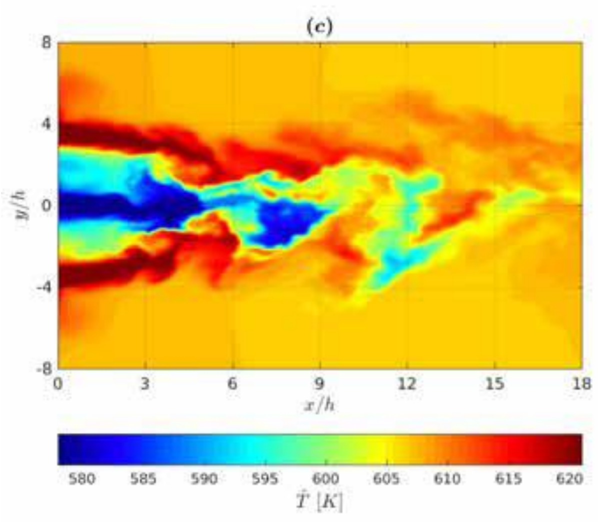
Information
Edoardo Cascioli
+32 470951794
e.cascioli@tudelft.nl

Project title
Study of low Prandtl number heat transfer in the E-SCAPE liquid metal pool facility

Project aim
The Multi-purpose hYbrid Research Reactor for High-tech Applications (MYRRHA) at SCK•CEN is being designed as a pool-type fast reactor prototype cooled by Lead-Bismuth Eutectic (LBE). The aim of the project is to provide a fundamental understanding of, and experimentally validated models for, the Turbulent Heat Transfer (THT) phenomena in pool type fast reactors cooled by liquid metals (low Prandtl number - Pr -fluids). These insights and models will make it possible to develop a reliable Computational Fluid Dynamics (CFD) tool for industrial use.

Progress
The computational work has been completed and efforts are focusing on scientific publications, to enable the drafting of the PhD thesis. Globally, RANS, U-RANS, LES and LES/DNS have been performed on forced jet flows with different fluids (mainly low-Pr values). Comparison and validation of experimental data from collaboration with VKI is ensuring the quality of the generated numerical database, which is valuable for future developments of advanced THT low-Pr models. The most promising RANS models have been tested to also provide indications on the current state-of-the-art for industrial applications.

2020 Publications
E. Cascioli et al. (2020). Numerical and experimental analysis of a planar jet with heated co-flow at medium and low Prandtl-number values. Nuclear Engineering and Design, 361, 110570.



TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof.dr. S. Kenjeres, Dipl.-Ing
Dr. ir. N. Bhattacharya (dept. ImPhys)
Prof. dr. ir. C. R. Kleijn

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Ir. Kevin van As

Funded by
NWO

Funded %
NWO 100 %

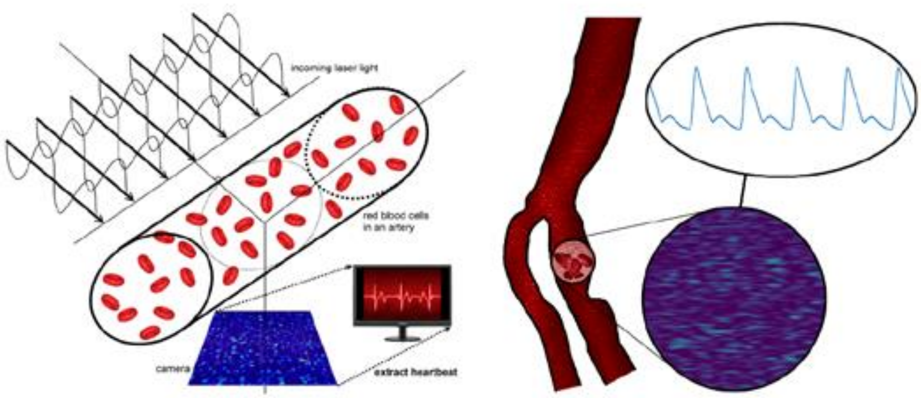
Start of the project
2015

Information
Kevin van As
K.vanAs@tudelft.nl
06 49622049

Project title
Coupling fluid dynamics and optics: dynamic interferometry from blood flow

Project aim
Measuring inside turbid media – such as human tissue – is difficult, as they are non-transparent due to the heavy scattering of light. Coherent light scattering is a technique which makes use of the heavy scattering of the medium, and is thereby rapidly becoming a technique for non-invasive monitoring of patients. The challenge is that the technique is currently still only semi-quantitative. By performing interferometric Mie scattering computer simulations, in combination with computational fluid dynamics, we aim to improve our understanding of the technique and thereby make it a fully quantitative technique.

Progress
We have ued our own Mie optics code for computing the light scattering by spherical particles (representing red blood cells) We made a step towards making Laser Speckle Contrast Imaging (LSCI) a quantitative flow measurement technique, and we made serious progress in using LSCI to detect atherosclerosis.



(left): Our set-up. Coherent light scatters of red blood cells and forms a noise like interferometric diffraction pattern, called speckle. We wish to extract information from the speckle pattern. (right): Complex patient-specific carotid artery with a stenosed region.

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof. dr. ir. C. R. Kleijn
Prof.dr. S. Kenjeres, Dipl.-Ing
Dr.ir. Mark Tummers

Research theme
Complex dynamics of fluids

Participants
Ir. Manu Chakkingal

Cooperation
Tata Steel, M2i

Funded by
STW, Industry

Funded %
NWO 90%
Industry 10%

Start of the project
2016

Information
CR Kleijn
c.r.kleijn@tudelft.nl

Project title
Low Prandtl number mixed convection flow in wall bounded coarse grained porous media – application to blast furnaces

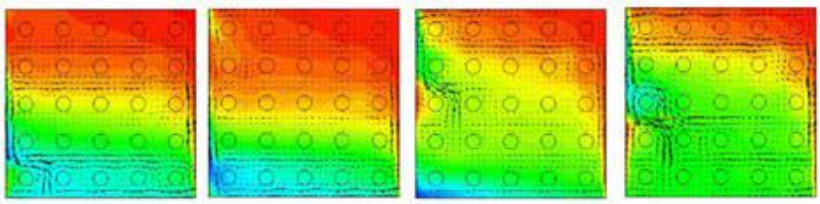
Project aim
The aim of the project is to provide a fundamental understanding of, and experimentally validated models for, the influence of coarse grained porous media on flow, turbulence and heat transfer in liquid steel. These insights and models will make it possible to design and to optimize new strategies for designing blast furnaces and cooling systems, leading to significant energy savings, increased productivity and improved product quality.

Progress
In the final year of the project we have extended our studies on the effect of porous packings on mixed and natural convection heat transfer in enclosures to include:
- The effect of the location of the porous packing in the enclosure
- The effect of double diffusive convection
- Non-uniform wall temperatures
- Assisting and opposing forced convection

Five journal papers and a PhD thesis have been published.

Dissertations
M. Chakkingal: Convective heat transfer in coarse-grained porous media. A numerical investigation of natural and mixed convection. (14 Sept. 2020).

2020 Publications
1. Effect of packing height and location of porous media on heat transfer in a cubical cavity: Are extended Darcy simulations sufficient? Chakkingal, M; Schiavo, S.; Ataei-Dadavi, I.; Tummers, M.J.; Kleijn, C.R. and Kenjeres, S. Int. J. Heat Fluid Flow, Vol. 84, 2020, #108617.
2. Effect of double-diffusive convection with cross gradients on heat and mass transfer in a cubical enclosure with adiabatic cylindrical obstacles. Chakkingal, M; Voigt, R.; Kleijn, C.R. and Kenjeres, S. Int. J. Heat Fluid Flow, Vol. 83, 2020, #108574.
3. Experiments on mixed convection in a vented differentially side-heated cavity filled with a coarse porous medium. Ataei-Dadavi, I.; Chakkingal, M; Kenjeres, S..; Kleijn, C.R.; Tummers, M.J. Int. J. Heat Mass Transfer, Vol. 149, 2020, #119238.
4. Numerical analysis of natural convection in a differentially heated packed bed with non-uniform wall temperature. Chakkingal, M; Kenjeres, S.; Ataei-Dadavi, I.; Tummers, M.J.; and Kleijn, C.R.Int. J. Heat Mass Transfer, Vol. 149, 2020, #119168.
5. Assisting and opposing mixed convection with conjugate heat transfer in a differentially heated cavity filled with coarse-grained porous media. Chakkingal, M; de Geus, J.; Kenjeres, S.; Ataei-Dadavi, I.; Tummers, M.J.; and Kleijn, C.R. Int. Comm. Heat Mass Transfer, Vol. 111, 2020, #104457.



Effect of various sinusoidally varying wall temperatures on natural convection in an enclosure packed with spherical beads.

TUD – Chemical Engineering – Transport Phenomena	
Project leaders David Vermaas, Bernard Dam Research theme Complex structures of fluids Cooperation TOeLS Project Funded by TKI and Shell Funded % Industry 100 % Start of the project 2019 Information Kostadin Petrov +351 915627365 k.v.petrov@tudelft.nl	Project title Membrane Electrode Assemblies (ME As) for stable alkaline CO2 electrolysis Project aim The anion-exchange membranes which are currently used in CO2 reduction cells carry some flaws such as lack of stability in alkaline conditions, or ion-selectivity and a cause for ohmic losses. Solving these issues could potentially make CO2 reduction an economically viable process. Charged inorganic nanoporous materials can be used as an alternative to polymeric IEMs. Their usage for ion transport is a fairly unexplored concept and relies on the ion selectivity in the double layer. This project will explore the relation between pore size, applied potential, surface charge of a material and selectivity of the developed membrane, with the aim of incorporating it into a CO2 reduction cell. Progress Simulations on the ion-transport through a single charged nanopore have been done using ITM software which employs the space-charge model. Deeper understanding on how the pore size, surface charge, electrolyte concentration, pH and applied current has been generated. Experiments to verify this model have also been done, and they seem to match the simulations. A side-project on a polymeric membrane with internal microchannels has been almost completed.

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Dr. Valeria Garbin

Research theme
Complex dynamics of fluids

Participants
Dr. Brice Saint-Michel

Cooperation
Imperial College London (UK)

Funded by
European Research Council (ERC)

Funded %
EU 100 %

Start of the project
2016

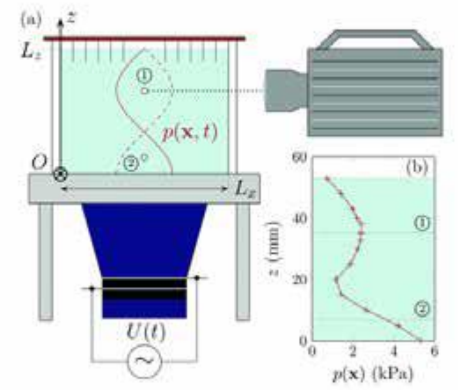
Information
Valeria Garbin
V.Garbin@tudelft.nl

Project title
Bubble dynamics in yield-stress fluids under ultra sound acoustic excitation

Project aim
We study the interaction of bubbles with a yield-stress fluid when they are driven into oscillations by a pressure wave. Such fluids behave as solids under a critical stress and thus naturally entrap bubbles, which can be desirable or unwanted depending on applications. Driving the bubbles into oscillations allows us to use them as local probes of the fluid, and also trigger their release for sufficient oscillation amplitudes. Our project investigates theoretically, numerically and experimentally the information that can be gained from bubble oscillations and the conditions for acoustic-induced bubble release from yield stress fluids.

Progress
We explored different regimes of microbubble oscillation and translation driven by an ultrasound field in a model yield-stress fluid, a Carbopol microgel. We used acoustic pressure gradients to induce bubble translation and examine the elastic part of the response of the material below yielding. At moderate pressure amplitude, the additional stresses applied by volumetric oscillations and acoustic radiation forces did not lead to irreversible bubble motion. At high pressure amplitude, we observed non-spherical shape oscillations that result in erratic bubble motion. The lack of bubble release in Carbopol is rationalized in terms of its high elasticity.

2020 Publications
1. B. Saint-Michel and V. Garbin, Bubble dynamics for broadband microrheology of complex fluids, Current Opinion in Colloid and Interface Science 50, 101392 (2020).
2. B. Saint-Michel and V. Garbin, Acoustic bubble dynamics in a yield-stress fluid, Soft Matter 16, 10405 (2020).
3. S. Saha, B. Saint-Michel, V. Laynes, B.P. Binks, V. Garbin, Stability of bubbles in wax-based oleofoams: decoupling the effects of bulk oleogel rheology and interfacial rheology, Rheologica Acta 59, 255 (2020).



TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof. dr. ir. C. R. Kleijn
Prof.dr. S. Kenjeres, Dipl.-Ing

Research theme
Complex dynamics of fluids

Participants
Elin Vesper

Funded by
FOM and Tata Steel

Funded %
NWO 50 %
Industry 50 %

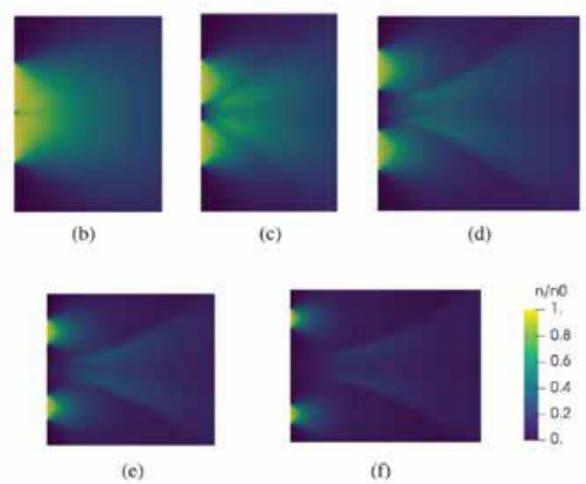
Start of the project
2017

Information
C.R. Kleijn
c.r.kleijn@tudelft.nl

Project title
Modeling of vapour flows for design optimization of a novel physical vapor deposition apparatus

Project aim
Physical Vapor Deposition (PVD) processes are traditionally used at a small scale and for batch processes for zinc coating. For the up-scaling to a continuous process it is necessary to use multiple vapor jets with a high mass flow rate. The Project aims at understanding the physics of the metal vapor transport towards the surface. The three major aims in this project are: First, the development of a hybrid solver coupling kinetic models and Direct Simulation Monte Carlo (DSMC). Second, modeling the flow inside the VDB and optimizing to achieve a high mass flow rate. Third, understanding the physics in a continuous PVD line, especially interacting jets.

Progress
We studied pairs of interacting planar plumes, with varying the degree of rarefaction, nozzle-separation-distance and mutual inclination. To consider the extremes of rarefaction, we derived an analytic solution for free molecular flow, and simulated the inviscid continuum solution using an approximate Riemann solver. In the transitional flow regime, direct simulation Monte Carlo was applied. To detect the shock location, we made use of the Method of Characteristics. We concluded that, although the rarefied flow regime physically lies in between the free molecular and the inviscid continuum flow regimes, the peak value of mass flux in the transitional flow regime exceeds both the one of free molecular flows and the one of inviscid continuum flows. Rarefied flow exhibits a broader, but weaker secondary expansion after the shock than continuum flow. The occurrence of the shock is rather insensitive to nozzle separation distance. Inclining the plumes toward each other leads to a stronger shock, but also to a stronger expansion, thus producing a more uniform mass flux with less stray mass fluxes.



Influence of nozzle separation on plume interaction. Number density contours as a function of nozzle separation L over nozzle diameter D. (b) D/L =30; (c) D/L=3; (d) D/L=1; (e) D/L=0.5; (f) D/L=0.25

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Prof. Rob Mudde, Dr. Ir. Luis Portela

Research theme
Complex structures of fluids

Participants
Ir. Manas Mandalahalli (PhD/TUD)

Cooperation
Prof. Hans Kuipers (TUE)
Prof. Detlef Lohse (UT)
Prof. Martin van Sint Annaland (TUE)
Dr. Ivo Roghair (TUE)
Dr. Bert Vreman (Akzo Nobel)
Dr. Peter Veenstra (Shell)
Dr. Patrick Wenmakers (DSM)
Dr. Christoph Dittrich (SABIC)
Ir. Dirk van der Plas (Tata Steel)

Funded by
NWO-IPP i36

Funded %
NWO 50 %
Industry 50 %

Start of the project
2015

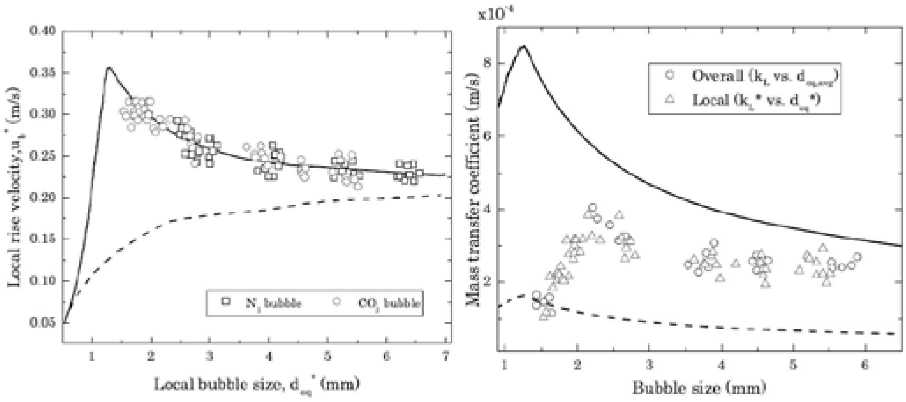
Information
Luis M. Portela
06 20046078
L.Portela@tudelft.nl

Project title
Mass transfer in bubbly flows

Project aim
Experimentally study of the local mass transfer phenomena and its implications on the global transport process in a dense bubbly flow. Local experimental techniques, such as optical probes, are slow and intrusive. Laserbased techniques for simultaneous flow and mass transfer are relatively new and need to be further developed at local bubble level, before its use in dense flows. A multiscale approach is used: from bubble-bubble interaction studies to swarm flows. This is done by a combination of intrusive probes and nonintrusive (X-ray) imaging techniques.

Progress
The focus was on studying the interplay between the hydrodynamics and mass transfer in a single bubble, using data from previous visualization experiments on the dynamics and size evolution of N2 bubbles (without mass transfer) and CO2 bubbles (with mass transfer), and from laser-induced fluorescence measurements of the CO2 concentration snapshots. The study showed that the mass-transfer process is a quasi-steady local process and that, in essence, a shrinking CO2 bubble simply behaves as a smaller N2 bubble.

2020 Publications
Mandalahalli, MM, Wagner, EC, Portela, LM, Mudde, RF. Electrolyte effects on recirculating dense bubbly flow: An experimental study using X-ray imaging. AIChE J. 2020; 66: e16696.



Left: Local rise velocity of the N2 and CO2 bubbles (continuous line: pure water; dotted line: contaminated water). Right: Comparison between the overall and local mass transfer coefficients (continuous line: Higbie correlation; dotted line: Frossling correlation).

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Dr. Luis Portela

Research theme
Complex structures of fluids

Participants
Prof. Ruud van Ommen (TUD)
Ing, Evert Wagner (TUD)

Funded by
TU Delft

Funded %
University 100 %

Start of the project
2007

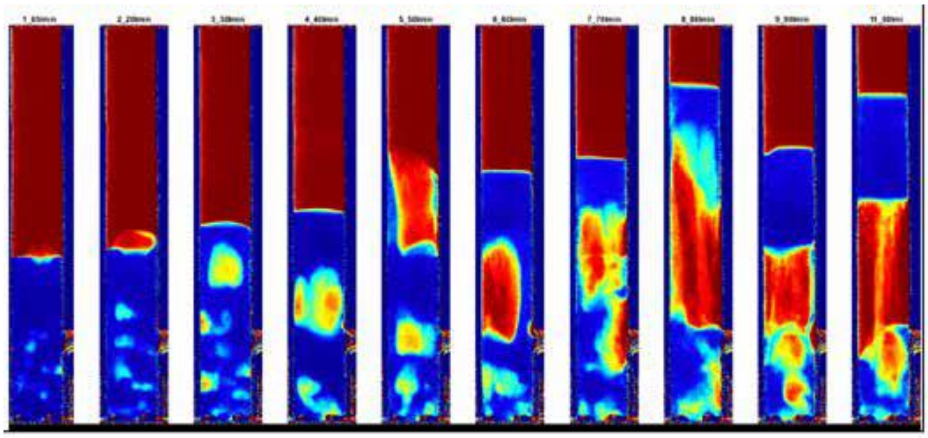
Information
Evert Wagner
e.c.wagner@tudelft.nl

Project title
X-ray tomography and SPECT imaging of dynamic structures in opaque multiphase flow

Project aim
To develop a fast X-ray tomography setup combined with SPECT particle tracking to measure velocities and structures inside opaque multiphase flows. The X-ray is sensitive to the phase distribution, whereas SPECT tracks radioactive particles in a multiphase system. We collaborate with various groups within and outside of TUD, who use this unique measurement technique in their research.

Progress
Together with CWI, we are further developing the algorithmic reconstruction techniques, with the goal being time resolved 3D reconstructions of bubbling fluidized beds.

2020 Publications
1. Electrolyte effects on recirculating dense bubbly flow: An experimental study using X-ray imaging. MM Mandalahalli, EC Wagner, LM Portela, RF Mudde AIChE Journal 66 (1), e16696.
2. Fluidization of spherical versus elongated particles-experimental investigation using X-ray tomography. I Mema, EC Wagner, JR van Ommen, JT Padding Chemical Engineering Journal 397, 125203.



TUD – Chemical Engineering – Transport Phenomena

Project leaders
L. M. Portela

Research theme
Complex dynamics of fluids

Participants
Matheus. M. Garcia

Cooperation
Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Institut National Polytechnique de Toulouse (INPT)
Lodz University of Technology (TU Lodz), Teletronic Rossendorf GmbH
CERG Fluides S.A.S., Frames Group B.V., Linde AG, Total S.A.
Shell Global Solutions International BV

Funded by
Marie Skłodowska-Curie actions (TOMOCON)

Funded %
EU 100 %

Start of the project
2019

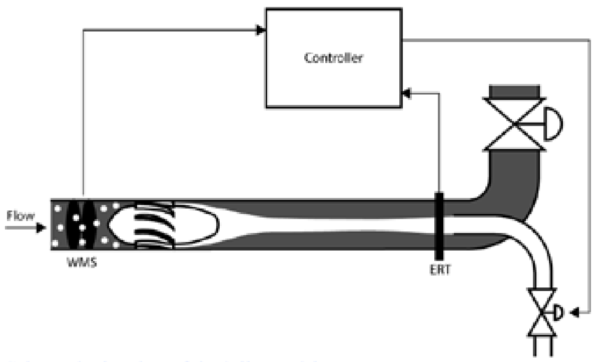
Information
Matheus Martinez Garcia
06 22753223
m.martinezgarcia@tudelft.nl

Project title
Control of an inline swirl fluid separator using tomography

Project aim
This project is part of a large EU consortium on Tomography and Control, TOMOCON (website: www.tomocon.eu), which targets new applications of smart control in the process industry. The main goal of this research is a proof of concept of Electrical Resistance Tomography (ERT) in the real-time control of an inline swirl fluid separator. Experiments are performed to investigate the physics behind the separation, to study the flow dynamics and to propose a simplified model of the process. The model is finally applied in the development of model-based controllers. The project has contributions of the following institutes: INTP, HZDR, and TU Lodz.

Progress
An industrial scale flow facility was built for the project, and the inline swirl separator has been operational since September 2020. Work was performed in the Electrical Resistance Tomography (ERT) system to make it faster and more reliable for the application, and the flow patterns taking place inside the separator were identified together with a MSc student. The current works involve the identification of the separation dynamics and the implementation of the proof of concept controller based on the ERT signal. A drawing of the controlled swirl separator is shown in the figure.

Scientific Publications
1. B. Sahovic, H. Atmani, M.A. Sattar, M.M. Garcia, E. Schleicher, D. Legendre, E. Climent, R. Zamansky, A. Pedrono, L. Babout, R. Banasiak, L.M. Portela,U. Hampel. Controlled Inline Fluid Separation Based on Smart Process Tomography Sensors. Chemie Ingenieur Technik, 2020. https://doi.org/10.1002/ cite.201900172.
2. M.M. Garcia, B. Sahovic, M.A. Sattar, H. Atmani, E. Schleicher, U. Hampel, L. Babout, D. Legendre, L.M. Portela. Control of a Gas-Liquid Inline Swirl Separator Based on Tomographic Measurements. IFAC-PapersOnLine, 2020. https://doi.org/10.1016/j.ifacol.2020.12.588.
3. M.A. Sattar, M.M. Garcia, R. Banasiak, L.M. Portela, L. Babout. Electrical Resistance Tomography for Control Applications: Quantitative Study of the Gas-Liquid Distribution inside A Cyclone. Sensors, 2020. https://doi. org/10.3390/s20216069.



Schematic drawing of the inline swirl separator.

TUD – Chemical Engineering – Transport Phenomena

Project leaders
L. M. Portela

Research theme
Complex dynamics of fluids

Participants
Siddartha Mukherjee, Frank Groen

Funded by
TU Delft

Funded %
University 100%

Start of the project
2018

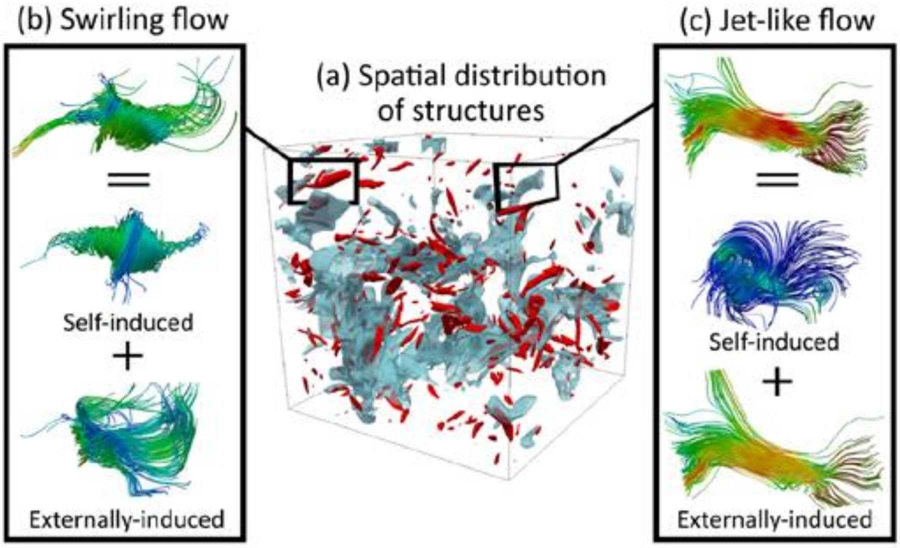
Information
Luis M. Portela
06 20046078
l.portela@tudelft.nl

Project title
Flow structures and organization in turbulence: a structure perspective on turbulence dynamics

Project aim
Study the dynamics of turbulent flows from a structure perspective, using generalized correlation and Helmholtz decomposition concepts. The usual governing equations provide a point-wise description of the flow dynamics, involving local derivatives. A structural perspective can describe turbulence (and dynamical vector fields in general) at a higher level of abstraction, allowing us to grasp large-scale features and emergent behaviour at a higher level of organization.

Progress
The general framework that had been developed was applied to study the force-density vector fields in isotropic turbulence. The interplay between these fields and the key structures of the velocity and vorticity fields (shown in the figure) will provide insights into the dynamics of these structures, and to what extent this dynamics is self-induced or externally-induced.

2020 Publications
S. Mukherjee, M. Mascini and L.M. Portela. Correlation and decomposition framework for identifying and disentangling flow structures: canonical examples and application to isotropic turbulence. arXiv preprint [physics.fludyn], arXiv:2011.07455v1 (2020).



Left: High enstrophy swirl structure. Right: High kinetic energy jet-like structure

TUD – Chemical Engineering – Transport Phenomena

Project leaders
L.M. Portela

Research theme
Complex dynamics of fluids

Participants
Galileu Oliveira (Petrobras engineer)

Cooperation
Petrobras

Funded by
Petrobras

Funded %
Industry 100 %

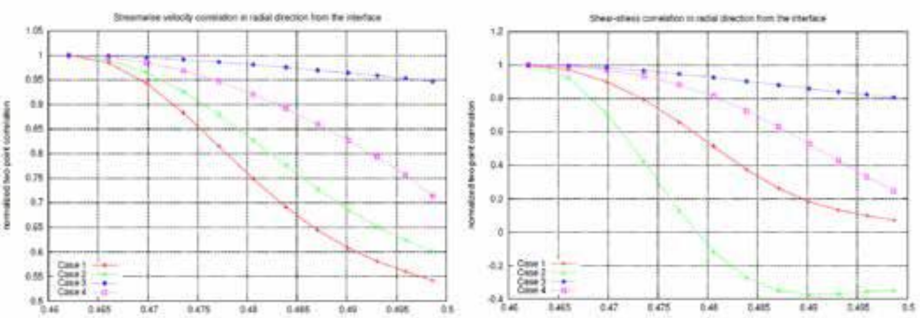
Start of the project
2015

Information
L.M. Portela
06 20046078
l.portela@tudelft.nl

Project title
Modeling and simulation of gas-liquid annular flow

Project aim
Use advanced numerical simulation techniques, like DNS and LES, together with some modelling, to isolate the different mechanisms in gas liquid annular flow, in order to get a better understanding of its dynamics,. Use this understanding to develop simple engineering models. The project is a continuation of the Ph.D. work of Galileu Oliveira, who had to resume his activities as an engineer at Petrobras; he is now working part-time in the project.

Progress
A correlation between the velocities and stresses at the interface and the velocities and stresses in the liquid film was performed, using DNS. The results show a high correlation, particularly for high viscosity liquid films. This suggests that it is possible to model in a simple way the dynamics in the liquid film and, in particular, to have a simple model for the stresses and velocities in the liquid film.



Correlation between the velocity (left) and shear-stress (right) at the interface (r==0.46) and in the liquid film (0.46<r<0.5).

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Luis M. Portela

Research theme
Complex dynamics of fluids

Participants
Debasish Sarker

Cooperation
Nouryon, DSM, Sabic, Shell, Tata, NWO

Funded by
NWO

Funded %
NWO 100 %

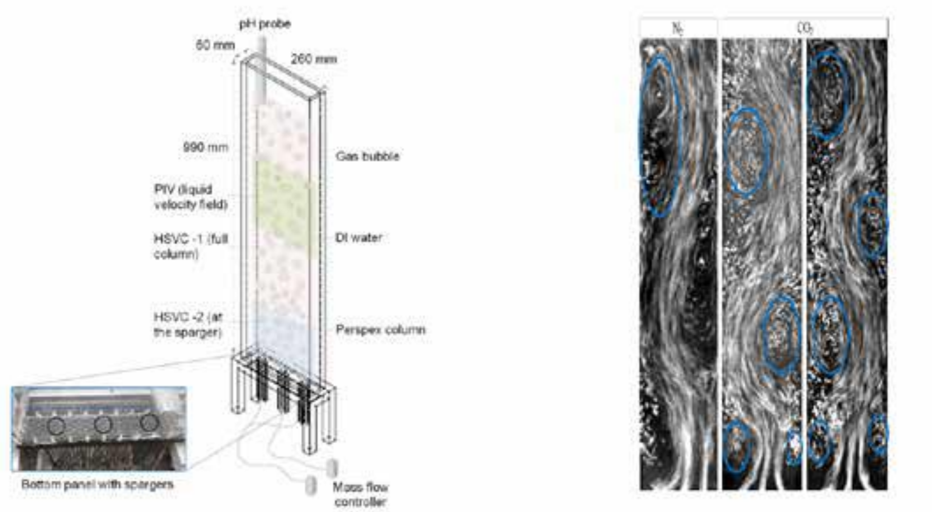
Start of the project
2019

Information
Luis M. Portela
06 20046078
l.portela@tudelft.nl

Project title
Interplay between hydrodynamics and mass transfer in bubbly flows: From dilute to dense

Project aim
Understand and quantify the effect of the gas fraction distribution on the overall mass transfer, through its effects on the interfacial area and hydrodynamics. Untangling their effects will provide a better understanding of the physics of dense bubbly flow. Experiments are performed in a pseudo-2D bubble column filled with water, with different patterns of N2 and CO2 bubbles injected at the bottom. Optical-based and X-ray imaging techniques, along with a pH probe to measure the dissolved CO2, are used.

Progress
The focus was on the analysis of the dynamics of the gas-patterns using feature extraction techniques from the 2D grey contours provided by the shadowgraphy. The gas injection pattern was fixed with a combination of three groups of needles. An example of the difference in the patterns obtained with N2 (without mass transfer) and with CO2 (with mass transfer) is shown in the figure.



A rectangular 2D bubble column. Bubble shadowgraphy. Left: N2 with Vsg=10.5 mm/s. Centre: CO2 with Vsg=10.5 mm/s. Right: CO2 with Vsg=21 mm/s.

TUD – Chemical Engineering – Transport Phenomena

Project leaders
L.M. Portela, Christian Schaerer, Norberto Mangiavachi

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Hyun Shin (Ph.D. student)

Cooperation
State University of Rio de Janeiro
National University of Asuncion

Funded %
Scholarships 100 %

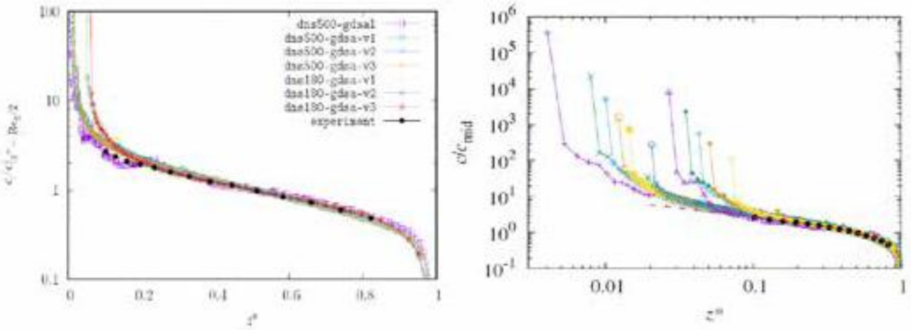
Start of the project
2015

Information
L.M. Portela
06 20046078
l.portela@tudelft.nl

Project title
Modeling and simulation of sediment transport

Project aim
Use sophisticated numerical simulation techniques, like DNS and LES, to improve the modelling and simulation of sediment transport using simpler engineering simulation tools (like two-fluid models and simple quasi-1D models).

Progress
The effect of different wall-resuspension models was careful evaluated. The results show that, even though the near-wall concentration is quite sensitive to the details of the model, further away from the wall the concentration is almost insensitive to the wall-resuspension model, as shown in the figure. From a pragmatic perspective, this indicates that simple wall resuspension models can be used in the simulation of suspended sediment transport. A paper was submitted to International Journal of Multiphase Flow.



Left: Profiles of the concentration for different particle-force and wall resuspension models
Right: Effect of different wall-resuspension models on the near-wall concentration

TUD – Chemical Engineering – Transport Phenomena

Project leaders
L. M. Portela

Research theme
Complex dynamics of fluids

Participants
Peng Wei

Cooperation
DSM, TUD Biotech Dept., University of Stuttgart, University of Liege, Syngulon, Centrient Pharmaceuticals

Funded by
ERA CoBiotech

Funded %
EU 100 %

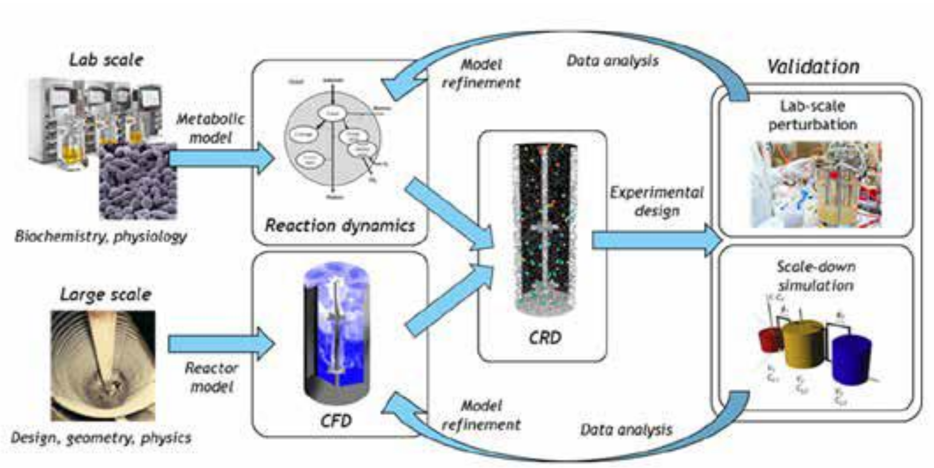
Start of the project
2018

Information
Peng Wei
06 17767196
P.Wei@tudelft.nl

Project title
Computational scale-up/scale-down simulator for industrial fermenters

Project aim
The project is embedded in a large EU consortium, ComRaDes (Computation for Rational Design: From Lab to Production with Success). The overall goal of the consortium is to develop a model-based platform to ramp up the scale-up/scale-down techniques currently used in bio-fermenters, which are based on empirical criteria. The focus of this project is on the systematic development of an integrated computational fluid/reactor dynamics (CFD+CRD) simulator, which will be built using ANSYS Fluent and Matlab.

Progress
The existing simulator, for a bubble-gassed stirred reactor with a dualimpeller, developed in the predecessor of this project, was expanded, in order to compute the dissolved-oxygen and the strain-rate lifelines (time-series "seen" by the micro-organisms), which can play an important role in the metabolism. Oxygen transfer and uptake processes were integrated in the CFD-CRD model; substrate/oxygen gradients, metabolic regime distributions, and lifelines data were obtained; preliminary scale-down design was implemented.



Schematic representation of the goals and approach used in ComRaDes

TUD – Chemical Engineering – Transport Phenomena

Project leaders
L.M. Portela

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Marisa Bazzi, Mohsen Barazandeh

Cooperation
Petrobras

Funded %
Scholarships 100 %

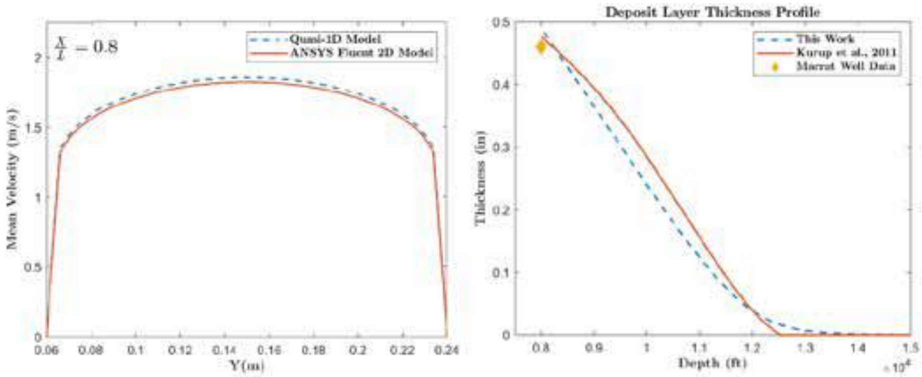
Start of the project
2015

Information
L.M. Portela
06 20046078
l.portela@tudelft.nl

Project title
Modeling of asphaltene deposition in pipelines

Project aim
Develop physically-based simple engineering models for the deposition of asphaltenes in pipelines.

Progress
A previously developed simplified population balance model, which was solved using a quasi-1D approach and the Direct Quadrature Method of the Moments, and had been compared with ANSYS Fluent and a benchmark test case for an actual oil well (see the figure), is being compared with the results of point-particle DNS and LES in pipe flow. This comparison will provide subsidies for physically-based parameter selection and for model improvements.



Left: Comparison between the turbulent fluid velocity profiles for the quasi-model and ANSYS Fluent, in a 2D flow contraction, from 0.3 m to 0.15 m over a distance L=100 m. Right: Comparison with a benchmark test case, in a vertical oil well, of the thickness of the deposition layer after two months of oil production, showing a good agreement both with the experimental data and an empirical model fitted to the well data (Kurup et al. Energy & Fuels 25, pp. 4506-4516, 2011).

TUD – Chemical Engineering – Transport Phenomena

Project leaders
Chris Kleijn, David Vermaas

Research theme
Complex dynamics of fluids

Participants
Lorenz Baumgartner

Funded by
EU

Funded %
EU 100 %

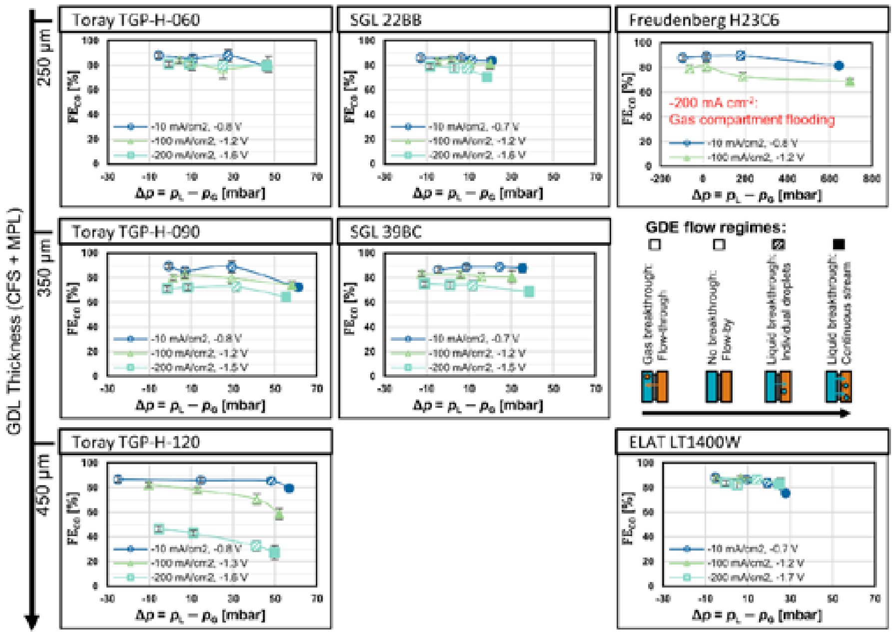
Start of the project
2018

Information
Lorenz Baumgartner
+49 15254915177
l.m.baumgartner@tudelft.nl

Project title
Scale-up limitation of gas diffusion electrodes (GDE) for electrochemical CO 2 reduction

Project aim
- Improve understanding of transport phenomena in GDEs to facilitate up-scaling
- Study the GDE structure to improve mass transfer and resistance against electrolyte flooding.

Progress
• Completed experimental phase (2 cm scale) and preparing publication
• Overview of results in picture below.



The flooding / liquid breakthrough due to high liquid overpressure does not reduce the faradaic efficiency (FECO) for GDEs with a thin carbon fibre substrate (CFS) and a wide pore size distribution. Preferential drainage paths in the porous GDE keep the overall liquid saturation low and maintain gas diffusion paths to the catalyst layer. Implications: GDE flooding can be tolerated as long as the gas compartment is sufficiently drained. This could make scale-up to the meter-scale feasible with commercial GDE materials.

TUD – Chemical Engineering – Product and Process Engineering

Prof.dr.ir. R van Ommen

The Product and Process Engineering (PPE) of the ChemE department of TU Delft views chemical engineering as an expanding field full of opportunities to create devices, processes and products. With expertise in reaction engineering, fluid mechanics and transport phenomena, we create solutions for soft-matter, nanotechnology, energy and lab-on-chip applications, often together with chemistry, physics and life-science groups. For us, engineering implies out-of-the-box thinking and design, from a sound basis in natural sciences with mathematics rigor. We are interested both in computational approaches and experimental work.

An important part of our work is related to the flow of droplets and bubbles in microchannels, with the aim of doing fun chemistry inside or outside those drops and bubbles. Each droplet can be seen as a miniaturized reactor that moves through a network on a chip as would a test tube through a chemistry lab. These flows are laminar, but the free interfaces bring in nonlinearities and instabilities, often driven by surface tension. Much of this work is done in cooperation with the group of prof. Chris Kleijn. There are lots of interesting chemistries that we deal with, ranging from catalysis to immiscible polymers.

Volkert van Steijn explores the use of aqueous polymer solutions to form droplets in an environment free of organic solvents and surfactants. Such droplets offer great potential for biomedical applications as they are fully biocompatible. In addition, he investigates the possibilities of using microfluidic devices for cell cultures.

Pouyan Boukany uses nanofluidics-based devices for providing quantitative insights into the fundamental mechanism of drug delivery, disease treatment, gene therapy and response of individual cells to therapeutic/ biomolecular reagents. In addition, he aims to understand the molecular dynamics of complex fluids using DNA as a model and advanced visualization techniques.

Ruud van Ommen is devoting an important part of his research efforts to dense gas-solid flows, where the solid phase consists of nanoparticles. In these systems, the nanoparticles cluster to form large, high-porosity agglomerates with fascinating interactions and flow properties. The aim is to chemically coat all individual nanoparticles in these agglomerates. He also studies gas-solid fluidized beds and three-phase systems with micron-sized particles, especially monitoring and structuring of these systems.

Gabrie Meesters is focusing his research on particle technology and product development. Complex multiphase flows involving particles and/or droplets are investigated, often in collaboration with industry. The relation between processing conditions and final products properties is a crucial part of this work.

TUD - Chemical Engineering - Product & Process Engineering

Project leaders
Dr. Ir. Volkert van Steijn
Dr. Ir. Gabrie Meesters
Prof. Dr. Ir. Ruud van Ommen

Research theme
□ Complex structures of fluids

Participants
E.J.G. Sewalt (PhD candidate)

Cooperation
FPE-WU, P&E-TUD, STW, Nutricia Research, DSM, Tetra Pak, General Electric

Funded by
STW, Nutricia Research, DSM, Tetra Pak.

Funding %
NWO 75%
Industry 25%

Start of the project
2017

Information
Volkert van Steijn
V.vanSteijn@tudelft.nl

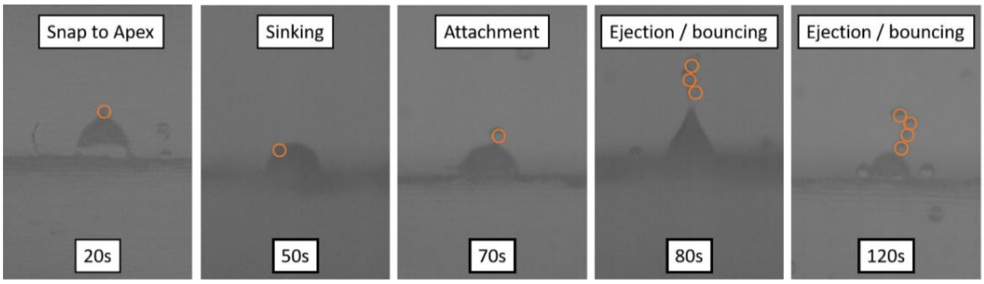
Project title
Multi-scale understanding and control of spray drying processes

Project aim

- To develop a better understanding of agglomeration behaviour in spray drying processes, by studying the drying and collision behaviour of drying droplets.
- Develop relations of drying temperature, dry matter content, and initial droplet size to the spatio-temporal concentration gradient and the resulting morphology.
- Develop relations of drying temperature, dry matter content, and initial droplet size to the collision behaviour of partially-dried droplets against solid objects.
- Additionally, the aim is to reduce the droplet size of testing to small droplets (< 200 μm) and higher temperatures to better mimic spray drying.

Progress
Two collision setups are being designed for testing the collisions.

1. Falling droplet dryer: A monodisperse stream of droplets falls through a glass column. A countercurrent nitrogen flow is present that can be heated before entry. Objects can be inserted into the side of the glass column to collide with the drying droplets.
2. Particle ejector: A particle ejector has been made and is used to eject solid particles against a drying droplet on a substrate. An example is shown in the figure.



Collisions of glass spheres (125-150 μm) against droplets with maltodextrin DE12 15 wt% (d_i = 400 μm) on a superhydrophobic substrate. A description of the collision event is shown as well as the drying time at room temperature.

Sticking conditions were investigated by comparing the theory for liquid bridge formation from sintering with the sticking conditions found for various stickiness measuring methods. The concepts of contact time between two sticking particles and the temperature elevation above the glass transition temperature are used to assess stickiness.

For studying the droplet evaporation of small droplets at higher temperatures we use droplet microfluidics. The used configuration is liquid-in-gas microfluidics. Water droplets were successfully made in a flow-focusing junction. Solute-containing droplets had problems with sticking to the channels of the chip. Wall modification through chemical coating and structuring has been used to decrease liquid-wall adhesion. Successful modification led to weakened binding of the two chip layers, which causes leakages. Hence, this approach has been unsuccessful.

A processing method using Optical Coherence Tomography scans is used to determine the diffusion coefficient gradient in an evaporating droplet with several micron-resolution. When the relation of concentration to viscosity is known a concentration mapping can be done using this relation and the Stokes-Einstein equation.

TUD - Chemical Engineering - Product & Process Engineering

Project leaders
Dr Pouyan E. Boukany

Research theme
Complex structures of fluids

Funded by
ERC

Funding %
EU 100%

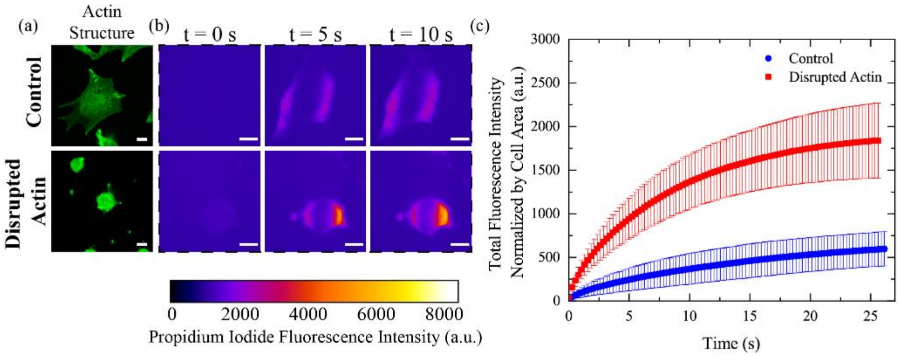
Start of the project
2017

Information
Dr. Pouyan E. Boukany
P.E.Boukany@tudelft.nl

Project title
Understanding the role of actin networks on uptake of small molecules to mammalian cells during electroporation.

Project aim
This Project aims at studying the role of actin networks, one of the cytoskeletal proteins present beneath the cell membrane on the uptake of small molecules to living cells during electroporation. Current theoretical frameworks for studying electroporation have been developed for simplified membrane systems like lipid bilayers and the predictions are inconsistent with the experiments done on living cells. We aim to provide experimental data, for living cells and cells with disrupted actin networks that will provide information for developing more accurate theoretical frameworks for predicting cellular responses to external electric fields.

Progress
To study the role of actin, we compare the fluorescence intensity emitted by propidium iodide (a marker molecule which emits a strong fluorescent signal when it enters the cell) during electroporation from Chinese Hamster Ovary (CHO) cells with intact actin network and chemically disrupted actin network. Our experiments show that disrupting actin networks prior to electroporation can result in increased uptake of propidium iodide to CHO cells during electroporation. We also perform the same experiments at different temperatures to correlate our data to the classical nucleation theory of electroporation.



(a) Structure of actin networks (in green) from control CHO cells and cells with disrupted actin. (b-c) Uptake of propidium iodide after applying an electric pulse of 800V/cm for 500 µs.

2020 Publications

S. Sachdev, S.F. Moreira, Y. Keehnen, L Rems, M.T. Kreutzer, P.E. Boukany, "DNA-membrane complex formation during electroporation is DNA size-dependent", Biochimica et Biophysica Acta (BBA)-Biomembranes 1862, 183089, (2020).

S. Sachdev, A. Muralidharan, D.K. Choudhary, D.L. Perrier, L. Rems M.T. Kreutzer, P.E. Boukany, "DNA translocation to giant unilamellar vesicles during electroporation is independent of DNA size", Soft matter 15 (45), 9187-9194 (2020).

TUD - Chemical Engineering - Product & Process Engineering

Project leaders
Dr. Ir. Volkert van Steijn
Prof. Dr. Ir. Michiel T. Kreutzer
Dr. Walter van Gulik

Research theme
Complex structures of fluids

Participants
Kartik Arun Totlani (PhD Student)

Funded by
NWO

Funding %
NWO 100%

Start of the project
2016

Information
Volkert van Steijn
V.vansteijn@tudelft.nl

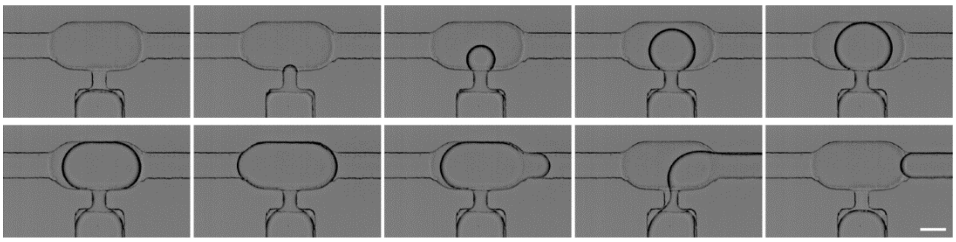
Project title
Development of droplet micro reactor for industrial relevant microbe screening

Project aim
This Project aims at development of a Lab on a Chip platform for performing high throughput screening of microbes under industrially relevant conditions. In current biotechnology practice screening of microbes is done in *batch* mode, while most industrial reactors operate in *fed-batch* mode. It is well established that the performance of microbes in nutrient rich environments differs from the performance in nutrient poor environments, explaining the limited success in microbe development. We address the lack of equipment to screen at fed-batch conditions by developing a microfluidic device where mutants grow in droplets, which can be periodically fed with nutrients to establish fed-batch conditions. By doing so, we wish to provide the same physiological environment to microbes during screening as it would have during the growth in the industrial fermentor.

Progress
This part of the project is dedicated to addressing the feeding part. Once the microbes are trapped inside a droplet it is important to develop a robust method to feed this cell containing droplet with nutrient droplets. In order to add defined volumes and concentration of nutrients, a droplet on demand method is established. This method ensures on demand generation and transport of monodisperse droplets which would form these nutrient droplets.

The droplet on demand generator consists of a T shaped junction. This junction is modified with a nozzle of a smaller size and height. The junction also includes a trap which confines the droplet. The height difference of the trap and main channel ensures the droplet is contained in the trap due to minimization of the surface energy.

The figure shows the droplet on demand formation process. Initially both oil /water pressure are balanced such that the interface rests at the nozzle. When the water pressure increases slowly the interface starts moving in the trap and fills the trap. In case the droplet phase exceeds the trap the curved edges and higher curvature in the trapping area enables relaxation of the droplet phase. Once the trap volume is completely filled, the oil pressure is increased to pushed the droplet out.



Grey scale images showing formation of Droplet on demand.

To test its robustness, we measured the resulting droplet volume for different pulse pressures, working fluids and waiting time between the generated droplets. To test its scalability, we fabricated a chip with 64 DoD-junctions in parallel, showing that all generators produce droplets with a volume between 91% and 105% of the predesigned volume.

We integrated DoD generators on a chip in which we trap a droplet with yeast cells, and supply nutrients to the cells by generating nutrient droplets and inducing coalescence with the cell-containing droplet. We show that the growth rate of the cells can be controlled through supply of the nutrients, enabling studies under fed-batch conditions. (Totlani et al, Advanced Materials Technologies, DOI: https://doi.org/10.1002/admt.202100083).

2020 Publications
Totlani, K., Hurkmans, J. W., Van Gulik, W. M., Kreutzer, M. T., & Van Steijn, V. (2020). Scalable microfluidic droplet on-demand generator for non-steady operation of droplet-based assays. Lab on a Chip, 20(8), 1398-1409.

TUD - Chemical Engineering - Product & Process Engineering

Project leaders
Ruud van Ommen
Volkert van Steijn

Research theme
Complex dynamics of fluids

Funded by
NWO (Project: Electrons to chemical bonds)

Funding %
NWO 70%
Industry 30%

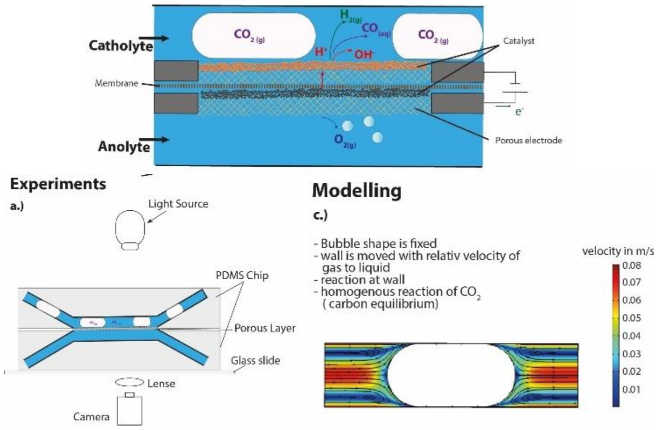
Start of the project
2019

Information
Isabell Bagemihl
i.bagemihl@tudelft.nl

Project title
Large Scale Modelling of Electrochemical Reduction of Carbon Dioxide to Ethylene

Project aim
Investigate gas-liquid reactors for the use of electrochemical reduction of carbon monoxide. Study the effect of gas evolution on the reactor performance and access how the reactor design and operating conditions dictate the reaction environment. Translate this knowledge to a large scale approach.

Progress
Experimental set-up and numerical model to study the effect of gas-liquid Taylor flow in electrochemical cells.



TUD - Chemical Engineering - Product & Process Engineering

Project leaders
Prof. Ruud van Ommen

Research theme
Complex dynamics of fluids

Participants
Fuweng Zhang, Damiano La Zara, Feilong Sun

Cooperation
AstraZeneca

Funded by
AstraZeneca, Health Holland

Funding %
University 20 %
Industry 60 %
TKI 20 %

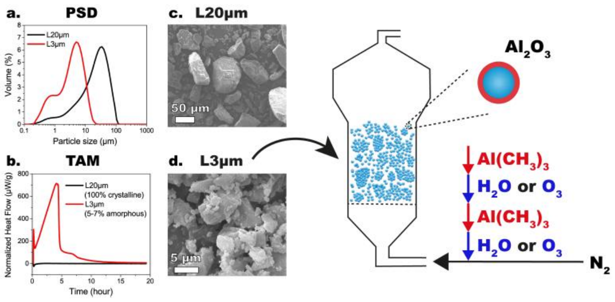
Start of the project
2016

Information
J.R. van Ommen
j.r.vanommen@tudelft.nl

Project title
Improving Fluidization and Conformal Coating of Ultrafine Pharmaceutical Powders in Fluidized Beds by ALD (atomic layer deposition).

Project aim
- Developing novel approaches to enhance the fluidization of cohesive pharmaceutical powders.
- Optimizing operating conditions (vibration/stirring/air jet) to get proper fluidization, so as to improve coating quality of ultra-fine powders.
- Minimizing the formation of agglomerates.
- Improving bulk powder properties (e.g., flowability and dispersibility).
- Unraveling the dynamic mechanism of the formation of agglomerates in the fluidized bed reactor.

Progress
- Set-up and modification of the experimental device – an ALD (atomic layer deposition) coating within a vibrated fluidized bed reactor.
- Explored the fluidization behaviour of serval kinds of ultra-fine powders under the coupled effects of vibration and gas (Nitrogen) flow.
- Completed the characterization of particles' physics properties before and after coating (particle
- size distribution/morphology analysis).



2020 Publications

La Zara, D., Bailey, M. R., Hagedoorn, P. L., Benz, D., Quayle, M. J., Folestad, S., & Van Ommen, J. R. (2020). Sub-nanoscale Surface Engineering of TiO2 Nanoparticles by Molecular Layer Deposition of Poly (ethylene terephthalate) for Suppressing Photoactivity and Enhancing Dispersibility. ACS Applied Nano Materials, 3(7), 6737-6748.

Guo, J., Benz, D., Nguyen, T. T. D., Nguyen, P. H., Le, T. L. T., Nguyen, H. H., La Zara, D., Liang, B., Hintzen, H. T., van Ommen, J. R., & Van Bui, H. (2020). Tuning the photocatalytic activity of TiO2 nanoparticles by ultrathin SiO2 films grown by low-temperature atmospheric pressure atomic layer deposition. Applied Surface Science, 530, 147244.

Navarrete, N., La Zara, D., Goulas, A., Valdesueiro, D., Hernández, L., van Ommen, J. R., & Mondragón, R. (2020). Improved thermal energy storage of nanoencapsulated phase change materials by atomic layer deposition. Solar Energy Materials and Solar Cells, 206, 110322.

TUD – Applied Mathematics – Numerical Analysis

Prof.dr.ir. C Vuik
Prof.dr.ir. CW Oosterlee

The research program of the Numerical Analysis group of TUD belongs to the field of computational science and engineering. We concentrate on the development and application of computing methods to the applied sciences. The focus is on mathematical models relying heavily on partial differential equations, such as occur in fluid dynamics. But we also consider similar mathematical models arising in other fields, for instance materials science and reservoir engineering, usually in cooperation with domain experts. A specialty is problems involving partial differential equations with moving internal boundaries, such as occur in bubbly flows and in phase transition problems. To diminish computing time in large-scale applications, iterative methods for solving large systems of algebraic equations are developed further, using deflation, multi-block, preconditioning and multigrid techniques.

Finally, these methods are implemented on modern hardware, clusters of PC's, GPU's and FPGA's. In order to achieve good results also HPC research is done in our group.

TUD – Applied Mathematics – Numerical Analysis	
Project leaders D. Toshniwal Research theme Mathematical and computational methods for fluid flow analysis Participants D. Toshniwal Cooperation University of Texas at Austin (R. Hiemstra,T.J.R. Hughes), University of Rome Tor Vergata (H. Speleers, C. Manni) Funded by UT Austin, TU Delft Funded % University 100 % Start of the project 2019 Information Deepesh Toshniwal d.toshniwal@tudelft.nl	Project title Non-uniform degree univariate and multivariate spline spaces Project aim Investigation of the theoretical and practical aspects of non-uniform degree splines, i.e., splines composed of polynomial pieces having different degrees. Progress A practical algorithm for computing multi-degree B-splines was previously presented; in the two publications below the algorithm is generalized to Tchebycheffian splines and a theoretical basis for it is presented. In particular, it is shown that the functions constructed by the algorithm are linearly independent, have the minimal support property, form a convex partition of unity, and span the entire non-uniform degree spline space. 2020 Publications 1. Toshniwal, D., Speleers, H., Hiemstra, R.R., Manni, C. and Hughes, T.J., 2020. Multi-degree B-splines: Algorithmic computation and properties. Computer Aided Geometric Design, 76, p.101792. 2. Hiemstra, R.R., Hughes, T.J., Manni, C., Speleers, H. and Toshniwal, D., 2020. A Tchebycheffian Extension of Multidegree B-Splines: Algorithmic Computation and Properties. SIAM Journal on Numerical Analysis, 58(2), pp.1138-1163.

<table>
<tr><th colspan="2">TUD – Applied Mathematics – Numerical Analysis</th></tr>
<tr><td>
Project leaders
C. Vuik

Research theme
Complex dynamics of fluids

Participants
C. Vuik, S. Maclachlan, Geenen, A. Segal, P. van Slingerland

Cooperation
TNO-Science and Industry
Utrecht University, Sepra Tufts University USA, Marin

Funded by
STW, TUD, TNO-Science and Industry, Nuffic-HEC, Marin

Funded %
University 25 %
Industry 75 %

Start of the project
1992

Information
C. Vuik
c.vuik@tudelft.nl
</td><td>
Project title
Solution methods for Navier-Stokes problems

Project aim
New preconditioners for the discretized Navier-Stokes equations will be developed. Parallel deflation methods will be included.

Progress
The discrete Navier-Stokes equations are solved by the SIMPLE(R) iteration method. To decrease the very large number of iterations, we have proposed multigrid and Krylov accelerated versions: GCR-SIMPLE(R). The properties of these methods are being investigated for simple two-dimensional flows and three-dimensional flows in industrial glass melting furnaces. These methods are generalised to a collocated discretization and combined with the deflated multiblock approach and parallel computing. Now we try to generalize these solvers to our FEM discretization (SEPRAN) and compare our methods with the recently developed methods given by Elman, Wathen, Sylvester, Benzi, Reusken and Schilders. It appears that MSIMPLER, a new variant of SIMPLER, leads to the fastest results. We also develop a solver based on the Schur complement and multigrid. This method is scalable and leads to very good results for geophysical applications. The GCR-simple solver is also implemented and tested in MARIN software. This led to a speed-up with a factor 5.

Dissertations
Baljinnyam Sereeter. Mathematical formulations and algorithms for fast and robust power system simulations Prof.dr.ir. C. Vuik and Prof.dr. C. Witteveen.

2020 Publications
1. AS. Markensteijn and J.E. Romate and C. Vuik (pdf, bibtex). A graph-based model framework for steady-state load flow problems of general multi-carrier energy systems Applied Energy, 280, 115286, 2020.
2. R. Tielen and M. Moller and D. Goddeke and C. Vuik (pdf, bibtex) p-multigrid methods and their comparison to h-multigrid methods within 2 Isogeometric Analysis. Computer Methods in Applied Mechanics and Engineering, 372, 113347, 2020.
3. Yue Qiu and Martin B. van Gijzen and Jan-Willem van Wingerden and Michel Verhaegen and Cornelis Vuik (pdf, bibtex). Preconditioning Navier-Stokes control using multilevel sequentially semiseparable matrix computations. Numerical Linear Algebra with Applications, 21, e2349, 2020.
4. X. He and C. Vuik (pdf, bibtex). Efficient and robust Schur complement approximations in the augmented Lagrangian preconditioner for the incompressible laminar flows. Journal of Computational Physics Volume 408, 1 May 2020, 109286.
5. M.E. Kootte, J.E. Romate, and C. Vuik (pdf, bibtex). Load flow computations for (integrated) Transmission and Distribution systems. A literature review Delft University of Technology. Delft Institute of Applied Mathematics. Report 20-01, 2020. ISSN 1389-6520.
</td></tr>
</table>

TUD – Applied Mathematics – Numerical Analysis

Project leaders
Matthias Möller

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Hugo Verhelst, Matthias Möller

Cooperation
TU Delft Department of Maritime and Transport Technology, Inria Sophia Antipolis – Mediterranée, Johannes Kepler University (JKU) Linz

Funded by
TU Delft

Funded %
University 100 %

Start of the project
2019

Information
Matthias Möller
m.moller@tudelft.nl

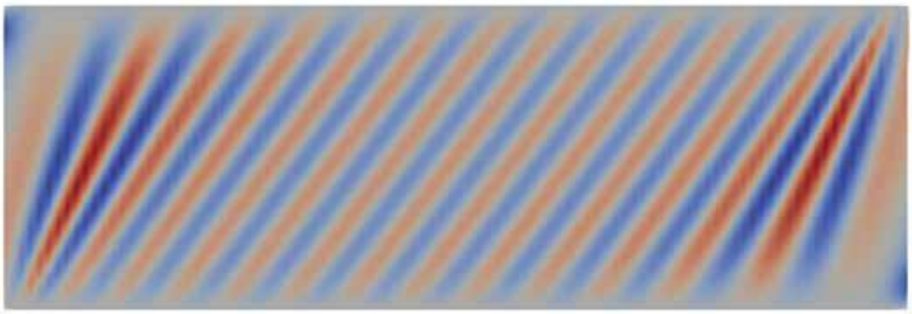
Information
M. Möller
m.moller@tudelft.nl

Project title
Numerical Analysis of Flexible Structures in the Ocean Environment

Project aim
Develop computational methods for the modelling of very thin, flexible, floating membrane structures within the framework of Isogeometric Analysis (IgA). The focus of the project is primarily on the development of models for structural analysis of complex membrane structures (e.g. with holes or reinforcement frames) and
their wrinkling response.

Progress
The isogeometric shell model has been implemented and the hyperelastic material module was finalized. Wrinkling simulations are performed and improved for tension and shear wrinkling cases with arc-length methods (ALMs). A novel parallel multi-grid ALM is currently being investigated. Mesh adaptivity methods have been developed; a paper will be submitted later this year. Strong patch coupling techniques have been implemented together with people from JKU and Inria and weak coupling methods will be considered soon.

2020 Publications
1. Verhelst, H. M., Möller, M., Besten, J. H. den, Kaminski, M. L., & Vermolen, F. J. (2020). Equilibrium Path Analysis Including Bifurcations with a Fully- Integrated Arc-Length Method Avoiding A Priori Perturbations. Proceedings of ENUMATH2019 Conference, 1109–1117. https://doi.org/10.1007/978-3-030-55874-1_110.



Out-of-plane solution for a wrinkling simulation for a thin sheet with constrained bottom side, free left and right sides, and a constrained top side with horizontal motion to the right. [simulation result by Hugo Verhelst].

TUD – Applied Mathematics – Numerical Analysis

Project leaders
M Möller

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Jochen Hinz, M. Möller

Cooperation
Johannes Kepler Universität Linz, Austria; Technische Universität Dortmund, Germany; RWTH Aachen, Germany

Funded by
EU

Funded %
EU 100%

Start of the project
2016

Information
M. Möller
m.moller@tudelft.nl

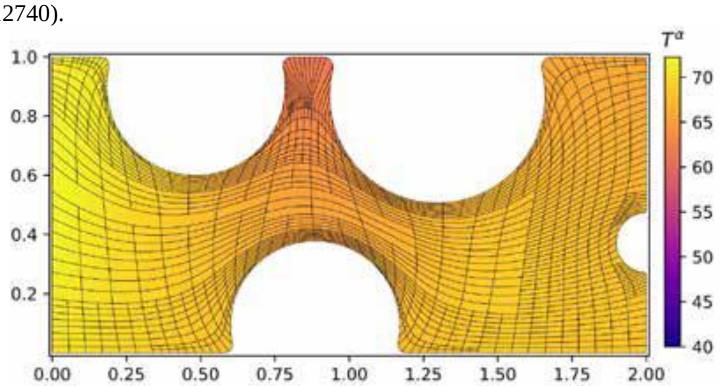
Project title
Isogeometric Grid Generation

Project aim
Develop efficient algorithms for the generation of analysis-suitable multi-patch volumetric B-spline parameterisations within the framework of isogeometric analysis. The focus is placed on complex geometries as they arise in industrial applications and, in particular, on twin-screw compressors (project partner TU Dortmund) and similar rotary devices (collaboration with RWTH Aachen on extruder geometries).

Progress
In the fourth year of his PhD, the PhD-candidate has finalized the development of his PDE-based framework for the fully automated generation of analysis-suitable volumetric multi-patch parametrizations for rotary twin-screw compressors and extruders. A particular highlight is the capability to refine parametrizations locally using THB-splines based on a goal-oriented error estimator of DWR-type. The PhD-candidate has furthermore embedded his framework into a gradient-based IgA shape optimization procedure in which both the parametrization and the solution are computed on-the-fly for each new shape under consideration.

Dissertations
Jochen Hinz, PDE-Based Parameterisation Techniques for Isogeometric Analysis Applications, TU Delft (https://doi.org/10.4233/uuid:6ae552a0-7425-4d7c-afd5- 5b7a42a10f27).

2020 Publications
1. Jochen Hinz, Michael Abdelmalik, and Matthias Möller. Goal-oriented adaptive THB-spline schemes for PDE-based planar parameterization, 2020. (https://arxiv.org/abs/2001.08874).
2. Jochen Hinz, Andrzej Jaeschke, Matthias Möller, and Cornelis Vuik. The role of PDE-based parameterization techniques in gradient-based IGA shape optimization applications. Computer Methods in Applied Mechanics and Engineering, 378:113685, 2021. (https://doi.org/10.1016/j.cma.2021.113685).
3. Jochen Hinz, Jan Helmig, Matthias Möller, and Stefanie Elgeti. Boundaryconforming finite element methods for twin-screw extruders using splinebased parameterization techniques. Computer Methods in Applied Mechanics and Engineering, 361:112740, 2020. (https://doi.org/10.1016/j. cma.2019.112740).



Cooling element designed with the gradient-based IgA shape optimization framework [source: Figure 7 from https://doi.org/10.1016/j.cma.2021.113685]

TUD – Applied Mathematics – Numerical Analysis	
Project leaders C. Vuik, C.W. Oosterlee Research theme Complex dynamics of fluids Participants C. Oosterlee, C. Vuik, D. Lahaye, A. Sheikh, V. Dwarka Cooperation TUD Chem.Tech, Philips, Shell, NLR, TU Berlin Funded by SenterNovem, NLR, Nuffic Funded % University 25% Industry 75 % Start of the project 2001 Information C. Vuik c.vuik@tudelft.nl	Project title Rigorous modeling of 3D wave propagation Project aim The aim is to develop efficient parallel iterative solvers for the Helmholtz problem. In order to estimate the layered structure of the earth crust seismic methods are used. The layer structure is used as input for porous media flow simulations. Progress A special preconditioner has been developed, which in a special combination of Krylov subspace and multigrid methods has resulted in a hundredfold increase in computing speed for the Helmholtz equation, describing wave propagation. Application in seismics has been very successful, and has generated much interest from the oil exploration industry, especially after a comparison with an industrial code in an application to a practical problem posed by industry. For the first time, realistic three-dimensional applications become feasible. This has already been realized on a single processor machine for medium-sized problems. The 3D code for the seismic simulation package has been parallelized. Furthermore, a comparison with analytic solutions will be made. The fast solver technique will be generalized to a finite element discretization of the Maxwell equations, for radar simulations. 2020 Publications V. Dwarka and C. Vuik (pdf, bibtex). Scalable Convergence Using Two-Level Deflation Preconditioning for the Helmholtz Equation. SIAM Journal of Scientific Computing, 42, pp. A901-A928, 2020.

TUD – Applied Mathematics – Numerical Analysis

<table>
<tr>
<td>
Project leaders
C. Vuik, F.J. Vermolen

Research theme
Complex dynamics of fluids

Participants
M.el Abbassi

Funded by
Deltares

Funded %
University 25 %
Industry 75 %

Start of the project
2007

Information
FJ Vermolen
</td>
<td>
Project title
Numerical methods for industrial flow problems

Project aim
Develop numerical methods for industrial flow problems.

Progress
A numerical method to cope with pressure boundary conditions on an extraction well has been constructed and analysed. The method is based on a finite-element discretization over the well where the integral boundary condition is evaluated as a boundary value problem. Oscillations are suppressed by the use of SUPG-techniques. Furthermore, a theorem has been formulated and proved about the shift of the spectrum of eigenvalues to the finite-element operator resulting from a standard and SUPG discretization of the boundary value problem. Besides this result, a model for the placement of bacteria has been coupled to the actual fortification model for soils. Next to this work, an experimental validation for the two-dimensional case of the model has been performed and a paper has been submitted about this issue to Transport in Porous Media. Furthermore, Menel Rahrah has started her PhD-project on DSI financed by STW. She works on poro-elasticity models to model the interaction between mechanical vibrations and fast flow of water through the soil at building sites.

Dissertations
Mathematical modelling of Fast, High Volume Infiltration in poroelastic media using finite elements
Prof.dr.ir. F.J. Vermolen and Prof.dr.ir. C. Vuik.

2020 Publications
D. Lahaye and M. el Abbassi and C. Vuik and M. Talice and F. Juretic (pdf, bibtex). Mitigating Thermal NOX by Changing the Secondary Air Injection Channel: A Case Study in the Cement Industry Fluids, 5, 220, 2020.
</td>
</tr>
</table>

TUD – Applied Mathematics – Numerical Analysis

Project leaders
S. Turek, M. Möller

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Babak S Hosseini, M. Möller

Start of the project
2011

Information
M. Möller
m.moller@tudelft.nl

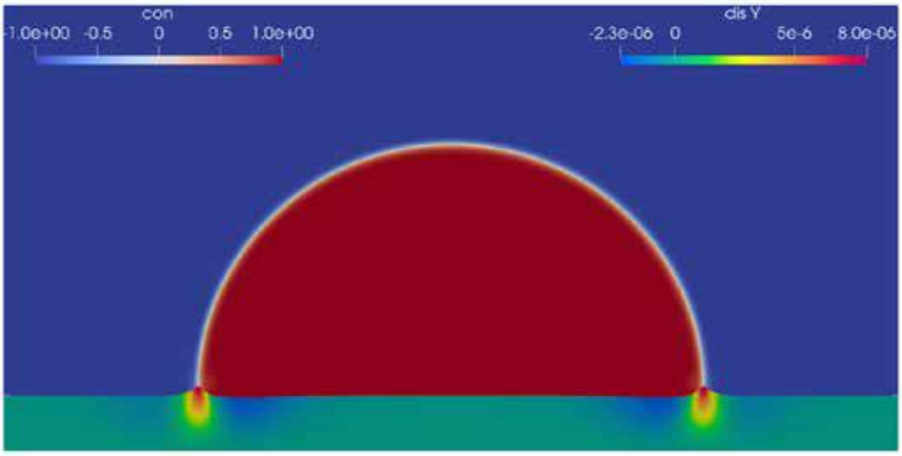
Project title
Isogeometric analysis of Cahn-Hilliard phase field-based Binary-Fluid-Structure Interaction based on an ALE variational formulation

Project aim
Development of a computational model and simulation technique capable of capturing the complex physics behind the intriguing phenomena of Elastocapillarity. Elastocapillarity refers to the ability of capillary forces or surface tensions to deform elastic solids through a complex interplay between the energy of the surfaces (interfaces) and the elastic strain energy in the solid bulk. The described configuration gives rise to a three-phase system featuring a fluid-fluid interface (for instance the interface of a liquid and an ambient fluid such as air) and two additional interfaces separating the elastic solid from the first and second fluids, respectively.

Progress
The PhD-candidate has extended his isogeometric analysis framework for two-phase flows to a multi-physics simulation toolbox for solving Binary-Fluid- Structure Interaction (BFSI) problems. The framework combines a Cahn-Hilliard phase field-based two-phase flow model with a hyperelastic material model for the solid. The FSI problem is implemented in a monolithic and fully coupled Arbitrary Lagrangian- Eulerian (ALE) variational formulation.

Dissertations
Hosseini B.S. (2020) Isogeometric analysis of Cahn-Hilliard phase field based Binary-Fluid-Structure Interaction based on an ALE variational formulation, submitted in August 2020, date of defence 24/02/2021 (https://eldorado.tu-dortmund.de/handle/2003/40171).

Scientific Publications
Hosseini, B.; Möller, M., Phase Field-Based Incompressible Two-Component Liquid Flow Simulation, van Brummelen, E., Corsini, A., Perotto, S., Rozza, G., 132, 165-176, Numerical Methods for Flows. Lecture Notes in Computational Science and Engineering, Springer, doi:10.1007/978-3-030-30705-9_15, 2020.



Simulation of a sessile glycerol droplet on a silicone deposit [Figure 6.5 from dissertation]

TUD – Applied Mathematics – Numerical Analysis

<table>
<tr>
<td>

Project leaders
M. Möller, C. Vuik

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Roel Tielen, M. Möller

Cooperation
Deltares

Funded by
University

Funded %
University 100 %

Start of the project
2017

Information
M. Möller
m.moller@tudelft.nl

</td>
<td>

Project title
High-Order MPM with application in fluid flow simulation

Project aim
Development of an improved Material Point Method (MPM) that overcomes the main shortcomings of this hybrid particle-mesh method in its standard variant, namely, the poor spatial and temporal accuracy due to grid crossing errors, inaccurate numerical integration, low-order finite element basis functions and low-order time integration schemes. These limitations shall be overcome by using high-order B-Spline basis functions inspired by the Isogeometric Analysis (IgA) approach, conservative function reconstruction techniques combined with 'exact' numerical quadrature and more accurate grid transfer as well as time integration schemes.

Progress
In the fourth year of his PhD, the PhD-candidate extended his research on efficient solvers for IgA to multipatch geometries. That is, the p-multigrid method developed in 2019 was applied to multipatch geometries and a special smoother was designed that makes use of the resulting block structure to increase the parallelizability of the p-multigrid method. Finally, the PhD-candidate collaborated with two fellow researchers to (1) extend MPM to Powell-Sabin splines and (2) compare/unify it with Optimal Transportation Meshfree (OTM) methods.

2020 Publications
1. R. Tielen, M. Möller, D. Göddeke and C. Vuik: p-multigrid methods and their comparison to h-multigrid methods within Isogeometric Analysis, Journal of Computer Methods in Applied Mechanics and Engineering, 2020.
2. P. de Koster, R. Tielen, E.D. Wobbes and M. Möller: Towards a Material Point Method with C1-continuous Powell-Sabin spline basis functions on unstructured triangulations , Journal of Computational Particle Mechanics, 2020.
3. E.D. Wobbes, R. Tielen, M. Möller and C. Vuik: Comparison and unification of material-point and optimal transportation meshfree methods, Journal of Computational Particle Mechanics, 2020.

</td>
</tr>
</table>

TUD – Applied Mathematics – Numerical Analysis	
Project leaders C Vuik Research theme Mathematical and computational methods for fluid flow analysis Participants G. Lipari, PhD Cooperation Shell Technology Centre, Bangalore, India Funded by Shell Funded % Industry 100% Start of the project 2018 Information C. Vuik, c.vuik@tudelft.nl	Project title GPU acceleration and numerical optimization for SPH codes Project aim GPUs stands for graphic processing units and SPH for smoothed particle hydrodynamics. SPH is a meshless method for Lagrangian particles suited for modelling fluids with highly-deformable, even fragmented interfaces. Advanced simulations of violent-impact flows like slamming gravity waves on structures require numbers of particles of O(108) and are effectively accelerated by GPU parallelism. To strike the optimum between algorithmic efficiency, simulation quality and compute efficiency, physics, mathematics and computer science are interwoven horizontally. The aim is to identify state-of-the-art SPH solvers best equipped for defined classes of problems. Progress All project milestones are disseminated on Twitter (https://twitter.com/ sph_delft), on a dedicated YouTube channel (http://bit.ly/sph_tube) and on the webpage the Numerical Analysis group at the Delft Institute of Applied Mathematics (https://www.tudelft.nl/ewi/over-defaculteit/afdelingen/appliedmathematics/ numerical-analysis/). 2020 Publications 1. AA. Lukyanov and C. Vuik (pdf, bibtex). A stable SPH discretization of the elliptic operator with heterogeneous coefficients. Journal of Computational and Applied Mathematics, 374, 2020. 2. E. Wobbes and R. Tielen and M. Moller and C. Vuik (pdf, bibtex). Comparison and unification of material-point and optimal transportation meshfree methods Computational Particle Mechanics, 2020. 3. E. Wobbes and M. Moller and V. Galavi and C. Vuik (pdf, bibtex). Taylor least squares reconstruction technique for material point methods Proceedings of the 6th European Conference on Computational Mechanics: Solids, Structures and Coupled Problems, ECCM 2018 and 7th European Conference on Computational Fluid Dynamics, ECFD 2018. Editors: R, Owen and R. de Borst and J. Reese and C, Pearce. International Centre for Numerical Methods in Engineering, CIMNE, Barcelona, pp. 806-817, 2020.

TUD – Applied Mathematics – Numerical Analysis	
Project leaders M. Möller Research theme Mathematical and computational methods for fluid flow analysis Participants Jochen Hinz, Matthias Möller Cooperation Johannes Kepler University Linz, Austria; Technische Universität Dortmund, Germany; RWTH Aachen, Germany Funded by EU Funded % EU 100 % Start of the project 2016 Information M. Möller m.moller@tudelft.nl	Project title PDE-based parameterization techniques for isogeometric analysis applications Project aim Develop efficient algorithms for the generation of analysis-suitable multi-patch volumetric B spline parameterisations within the framework of isogeometric analysis. The focus is placed on complex geometries as they arise in industrial applications and, in particular, on twin-screw compressors (project partner TU Dortmund) and similar rotary devices (collaboration with RWTH Aachen on extruder geometries. Progress The PhD-candidate has developed a fully functional framework for the (semi-)automatic generation of analysis-suitable multi-patch parameterizations for isogeometric analysis (IgA) simulations. The IgA-framework makes use of an elliptic grid generation approach and utilizes advanced reparameterisation and solution techniques to produce high-quality planar parameterisations for stationary and rotating screw machine geometries. The PhD-candidate has extended the framework to volumetric geometries of twin screw machines and devised a mixed formulation approach that enables the straightforward handling of multi-patch topologies. The developed techniques were successfully applied in the numerical simulation of co-rotating twin screw machine extruders. Dissertations Jochen Hinz (Cum Laude) PDE-Based Parameterization Techniques for Isogeometric Analysis Applications Prof.dr.ir. C. Vuik and Dr. M. Möller.

TUD – Applied Mathematics – Numerical Analysis

Project leaders Fred Vermolen Research theme Mathematical and computational methods for fluid flow analysis Participants Qiyao Peng (TUD), Ginger Egberts (TUD), Daphne Weihs (Technion, Israel), Paul van Zuijlen (VUAmsterdam), Fred Vermolen (UHasselt, Belgium) Cooperation Technion Israel, Amsterdam Free University, UHasselt Funded by China Scholarship Council (CSC), Dutch Burns Foundation (Nederlandse Brandwondenstichting) Funded % Scholarships 100 % Start of the project 2015 Information FJ Vermolen	Project title Mathematics of burn injuries Project aim The Project aims at modeling the contraction and formation of hypertrophic scars in burn injuries in order to improve treatments so that the formation of hypertrophic scar tissue, as well as contractures are prevented. Progress The research is devoted to partial differential equations and agent-based formalisms to model the contraction reaction of scarred skin after a serious burn injury. Furthermore, the momentum balance with morphoelasticity is considered. Numerical methods are based on the finite-element method and isogeometric analysis. Furthermore, mathematical analysis has been done to assess stability and the treatment of the immerse boundary method. The models were also subject to uncertainty quantification. 2020 Publications 1. F.J. Vermolen. Mathematical models of healing of burns. In: Innovations and emerging technologies in wound care. Chapter 1, Edited by A. Gefen, Academic Press, 2020, pages 1—20. 2. Q. Peng, F.J. Vermolen. Point forces and their alternatives in cell-based models for skin contraction in two dimensions. Proceedings (reviewed) of the International Conference on Mathematics and Computers in Science, IEEE, 2020, 250-259. 3. Q. Peng, F.J. Vermolen. Numerical methods to compute stresses and displacements from cellular forces: application to the contraction of tissue. Arxiv, 2009.08777, 2020. 4. Q. Peng, F.J. Vermolen. Point forces in elasticity equation and their alternatives in multi dimensions. Arxiv, 2009.08795, 2020. 5. G. Egberts, F.J. Vermolen, P.P.M. van Zuijlen. A one-dimensional morphoelastic model for burn injuries: sensitivity analysis and a feasibility study. Arxiv, 2010.12902, 2020. 6. G. Egberts, F.J. Vermolen, P.P.M. van Zuijlen. A one-dimensional morphoelastic model for burn injuries: stability analysis, numerical validation and biological interpretation, Arxiv, 2012.12897, 2020. 7. Q. Peng, F.J. Vermolen. Agent-based modelling and parameter sensitivity analysis with a finite element method for skin contraction. Biomechanics and Modelling in Mechanobiology. 19 (6), 2020, 2525—2551.

TUD – Applied Mathematics – Numerical Analysis	
Project leaders Fred Vermolen Research theme Mathematical and computational methods for fluid flow analysis Participants Menel Rahrah (TUD), Bernard Meulenbroek (TUD), Luiz Lopez-Pena (TUD), Fred Vermolen (UHasselt) Cooperation UHasselt Funded by STW Funded % NWO 100 % Start of the project 2015 Information FJ Vermolen	Project title Mathematical models for flow in porous media Project aim Develop models and computational methods for flow in porous media. Progress The research consists of using finite element methods for the solution of Biot-type problems. The solution is done by the use of several finite element methods. We have accomplished a moving finite element in order to deal with large deformations. Furthermore, we have proved monotonicity of numerical solution under certain conditions and under the use of stabilization with applications to morpho-poroelasticity. Dissertations Menel Rahrah. Mathematical modelling of Fast, High Volume Infiltration in poroelastic media using finite elements, June 8, 2020, TU-Delft. 2020 Publications 1. M. Rahrah, F.J. Vermolen. A moving finite element method for fast infiltration in nonlinear poroelastic media. Journal of Computational Geosciences, 25(2), 2020, 793—805. 2. M. Rahrah, L.A. Lopez-Pena, F.J. Vermolen, B. Meulenbroek. Network-inspired versus Kozeny-Carman based permeability-porosity relations applied to Biot's poroelasticity model. Journal of Mathematics in Industry, 10(1), 2020, 1—21.

TUD – Applied Mathematics – Numerical Analysis	
Project leaders Fred Vermolen Research theme Mathematical and computational methods for fluid flow analysis Participants Jiao Chen (TUD), Ilkka Polonen (Jyvaskyla), Daphne Weihs (Technion) Cooperation University of Jyvaskyla, Technion, UHasselt Funded by China Scholarship Council Funded % Scholarships 100 % Start of the project 2015 Information FJ Vermolen	Project title Agent-based modeling of cancer Project aim Develop agent-based models for cancer. Progress Agent-based and cellular automata models have been developed to simulate cancer. Uncertainty quantification has been used to estimate the probability of various pathologies. Applications are in pancreatic cancer and skin cancer. Dissertations Jiao Chen Agent-based mathematical modeling of pancreatic cancer growth and several therapies Prof.dr.ir. C. Vuik and Dr.ir. F.J. Vermolen. 2020 Publications 1. J. Chen, D. Weihs, F.J. Vermolen. A cellular automata model of oncolytic virotherapy in pancreatic cancer. Bulletin of Mathematical Biology. 82(8), 2020, 1—25. 2. F.J. Vermolen, I. Polonen. Uncertainty quantification on a spatial Markov-Chain model for the progression of skin cancer. Journal of Mathematical Biology, 80(3), 2020, 545—573.

TUD – Applied Mathematics – Mathematical physics

Prof.dr.ir. AW Heemink
Prof.dr.ir. ELC Deleersnijder
Prof.dr.ir M. Verlaan

The central research direction of the group is the mathematical modeling of physical phenomena using (partial) differential equations. The research is application driven and includes the modeling phase, analysis of the model and the numerical implementation of the model. The focus is now more and more on the research themes:

Inverse modeling and data assimilation
Data assimilation methods are used to combine the results of a large scale numerical model with the measurement information available in order to obtain an optimal reconstruction of the dynamic behaviour of the model state. Many data assimilation schemes are based on solving the Euler-Lagrange equations. A recursive algorithm to solve this two-point boundary value problem can be derived and results in the well-known Kalman filtering algorithm. Variational data assimilation is also a powerful method, but requires the implementation of the adjoint (of the tangent linear approximation) of the numerical model. In a series of externally funded PhD projects the mathematical algorithms have been developed and applied in a number of real-life applications:

- Tidal flow models (funding: Rijkswaterstaat)
- Atmospheric-chemistry modeling (funding: NWO, TNO, RIVM).
- Reservoir modeling (Funding: Shell, TNO).

Perturbation methods for partial differential equations
The main focus within this theme is to develop perturbation methods to analyse initial value problems and initial-boundary value problems for partial differential equations. The applications are in a variety of fields, such as: the wind flow (or rain-wind) induced oscillations of bridges, high-rise buildings, or of overhead power transmission lines; the vibrations of conveyor belts; and the morphodynamics in tidal embayments.

High performance computing and parallel algorithms
This research theme aims to design efficient and scalable parallel algorithms and apply high performance computing technology to applications, such as the storm surge forecasting and pollutant transport in North Sea or rivers. Domain decomposition and grid partitioning is an effective approach for parallel simulation of models described by partial differential equations. Sparse matrices typically occur in numerical simulation of problems described by partial differential equations. One of our research focus is on designing parallel algorithms for solving sparse matrix systems. Lagrangian models, often also called particle models, for transport problems in coastal waters, can deal with steep gradients of concentration. Because the movements of the particles are largely independent from each other, so particle models are very suited for parallel and distributed computing. We have developed parallel models for transport problems of the Dutch coastal water (e.g., Wadden sea). Currently, a particle model with adaptive time steps is being developed, besides the derivation of the numerical scheme the additional challenge is to maintain a good load balance in an adaptive scheme. Grid computing is the next step of development in high performance computing.

<table>
<tr><th colspan="2">TUD – Applied Mathematics – Mathematical Physics</th></tr>
<tr><td>
<u>Project leaders</u>
A.W. Heemink

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
C. van Velzen, M. Verlaan, A.W. Heemink, G.Y. El-Serafy, Cong Xiao, Jianbing Jin, A. Ziemba, Aad Vijn

<u>Cooperation</u>
Deltares, Statoil, TNO, Vortech

<u>Funded by</u>
Deltares, Shell, TNO, NWO

<u>Funded %</u>
NWO 40%
TNO 30%
GTI 20%
Scholarships 10%

<u>Start of the project</u>
2001

<u>Information</u>
A.W. Heemink
a.w.heemink@tudelft.nl
</td><td>
<u>Project title</u>
Data assimilation in CFD

<u>Project aim</u>
Large scale numerical models are often used for prediction problems. These models however are however far from perfect. The model predictions can be improved by assimilating measurements into the model. Model Order Reduction is a corner stone in developing new efficient sub-optimal data assimilation schemes.

<u>Progress</u>
We have developed a model reduction methodology for large scale numerical models in corporation with TNO, Shell and Deltares. New PhD projects around the theme “Inverse modeling of atmospheric transport” have started. In these PhD projects we will develop and apply model reduction and filtering techniques for assimilating data into the EUROS-LOTOS atmospheric transport models. Other application areas are ecological coastal sea models and reservoir models.

<u>2020 Publications</u>
1. Forecasting PM10 and PM2.5 in the Aburrá Valley (Medellín, Colombia) via EnKF based data assimilation, S Lopez-Restrepo, A Yarce, N Pinel, OL Quintero, A Segers, AW Heemink, Atmospheric Environment 232, 117507.
2. Source backtracking for dust storm emission inversion using an adjoint method: case study of Northeast China, J Jin, A Segers, H Liao, A Heemink, R Kranenburg, HX Lin, Atmospheric Chemistry and Physics 20 (23), 15207.
3. Precipitation regime classification based on cloud-top temperature time series for spatially-varied parameterization of precipitation models, S Lu, M ten Veldhuis, N van de Giesen, A Heemink, M Verlaan, Remote Sensing 12 (2), 289.
</td></tr>
</table>

TUD – Applied Mathematics – Mathematical Physics

Project leaders
A.W. Heemink

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
W.T. van Horssen, H.M. Schuttelaars, J.L.A. Dubbeldam, A. Geyer, Y. van Gennip, T. Akkaya, M. Kumar, X. Wei, K. Xi, Y. Dijkstra, Jin Jianbing, Xiao Deng, Wang Jing, Jie Liu

Funded %
University 60%
NWO 10%
Industry 10%
Scholarships 20%

Start of the project
2003

Information
W.T. van Horssen
W.T.vanHorssen@tudelft.nl

Project title
Perturbation methods for partial differential equations

Project aim
The main focus within this project is to develop and to apply perturbation methods to analyze initial value problems and (initial)boundary value problems for partial differential equations. The applications are in a variety of fields, such as: the wind or rain-wind induced oscillations of elastics structures (such as bridges, high-rise buildings, and overhead power transmission lines); the vibrations of conveyor belts and elevator cables; the morphodynamic evolution of coastal systems (such as beaches, and estuaries); nonlinear water waves and the dynamics of polymers in shear flow.

Progress
In 2020 the applicability of different types of perturbation and bifurcation methods was investigated. For problems with boundary damping, for nonselfadjoint problems, for weakly nonlinear problems, and for problems with variable coefficients all kinds of computational aspects have been studied by using perturbation methods, methods from dynamical system theory, numerical methods, and stochastic methods.

2020 Publications
1. Classification of link-breaking and link-creation updating rules in susceptibleinfected susceptible epidemics on adaptive networks Achterberg, M. A., Dubbeldam, J. L. A., Van Mieghem, P. & Stam, C. J., 2020, In: Physical Review E. 101, 5, p. 052302-1 - 052302-19 19 p., 052302.
2. Dynamics and buckling loads for a vibrating damped Euler–Bernoulli beam connected to an inhomogeneous foundation Abramian, A. K., Vakulenko, S. A., Lukichev, D. V. & van Horssen, W. T., 2020, In: Archive of Applied Mechanics. 91, 4, p. 1291-1308 18 p.
3. Graph Merriman-Bence-Osher as a semidiscrete implicit Euler scheme for graph Allen-Cahn flow Budd, J. & van Gennip, Y., 2020, In: SIAM Journal on Mathematical Analysis. 52, 5, p. 4101–4139 39 p
4. Historical and future development of the tidally averaged transport of sandy sediments in the Scheldt estuary: a 2D exploratory model Boelens, T., Schuttelaars, H., Plancke, Y. & De Mulder, T., 2020, In: Ocean Dynamics. 70, 4, p. 481-504 24 p.
5. Influence of Back-Barrier Basin Geometry on Multiple Tidal Inlet Systems: The Roles of Resonance and Bottom Friction Reef, K. R. G., Roos, P. C., Schuttelaars, H. M. & Hulscher, S. J. M. H., 2020, In: Journal of Geophysical Research: Earth Surface. 125, 3, p. 1-20 20 p., e2019JF005261.
6. Intratidal and spatial variability over a slope in the Ems estuary: Robust alongchannel SPM transport versus episodic events Schulz, K., Burchard, H., Mohrholz, V., Holtermann, P., Schuttelaars, H. M., Becker, M., Maushake, C. & Gerkema, T., 2020, In: Estuarine, Coastal and Shelf Science. 243, p. 1-12 12
p., 106902.
7. Kemeny’s constant for several families of graphs and real-world networks Kooij, R. E. & Dubbeldam, J. L. A., 2020, In: Discrete Applied Mathematics. 285, p. 96-107 12 p.
8. Oblique sand ridges in confined tidal channels due to Coriolis and frictional torques Hepkema, T. M., de Swart, H. E., Nnafie, A., Schramkowski, G. P. & Schuttelaars, H. M., 2020, In: Ocean Dynamics. 70, 12, p. 1505-1513 9 p.
9. On the Bolotin’s reduced beam model versus various boundary conditions Andrianov, I. I., Awrejcewicz, J. & van Horssen, W. T., 2020, In: Mechanics Research Communications. 105, 10 p., 103505.
10. Spectral instability of the peaked periodic wave in the reduced Ostrovsky equations Geyer, A. & Pelinovsky, D. E., 2020, In: Proceedings of the American Mathematical Society. 148, 12, p. 5109-5125 17 p
11. The Stability of Non-linear Power Systems Xi, K., Dubbeldam, J. L. A., Gao, F., Lin, H. X. & van Schuppen, J. H., 3 Nov 2020, Nonlinear Dynamics of Discrete and Continuous Systems. Abramian, A., Andrianov, I. & Gaiko, V. (eds.). Cham, Switzerland: Springer, Vol. STRUCTMAT 139. p. 217-238 22 p. (Advanced Structured Materials).
12. Unraveling the essential effects of flocculation on large-scale sediment transport patterns in a tide-dominated estuary Horemans, D. M. L., Dijkstra, Y. M., Schuttelaars, H. M., Meire, P. & Cox, T. J. S., 2020, In: Journal of Physical Oceanography. 50, 7, p. 1957-1981 25 p.

TUD – Applied Mathematics – Mathematical Physics	
Project leaders A.W. Heemink, H.X. Lin Research theme Mathematical and computational methods for fluid flow analysis Participants H.X. Lin, J.B. Jin, T. Deng Cooperation TNO, CMA China, Nanjing University of Information Science and Technology, China Funded % University 50% Scholarships 50% Start of the project 2015 Information H.X.Lin H.X.Lin@tudelft.nl	Project title Modelling and forecasting of air pollution Project aim The research aims at improving the forecast accuracy of (mainly) PM10 concentrations caused by dust storms. Progress The research focuses on modeling the emission and transport process of dust in heavy dessert storms. Measurements data are assimilated to improve model forecast, in particular the unknown spatial varying emission parameters are estimated. We want to predict the dust concentrations in populated cities often far from the dessert. The PM10 observations around the city actually represent a sum of dust and non-dust aerosols, the latter are anthropogenic emissions from factories and vehicles etc. It is impossible to separately measure the two. So, for the dust emission inversion we use machine learning to estimate the PM10 concentrations as a result of anthropogenic emissions and then compute the PM10 concentrations related to the dust storms. Simulation using the chemical transport model (CTM) Lotus-Euros has shown that the combination of data assimilation and machine learning produces far more accurate forecast. 2020 Publications 1. J. Jin, A. Segers, H. Liao, A.W. Heemink, R. Kranenburg, H.X. Lin, Source backtracking for dust storm emission inversion using adjoint method: case study of northeast China, Atmospheric Chemistry and Physics, 2020. 2. J. Jin, A.J. Segers. H.X. Lin, B. Henzing, X. Wang, A.W. Heemink, H. Liao, Position correction in dust storm forecast using LOTUS-EUROS v2.1: grid distorted data assimilation v1.0, Geoscientific Model Development.

TUD – Applied Mathematics – Mathematical Physics	
Project leaders D. Lahaye Research theme Mathematical and computational methods for fluid flow analysis Participants Mohamed el Abbassi (PhD student) C. Vuik Funded by Almatis B.V. Rotterdam Funded % Industry 100 % Start of the project 2015 Information Domenico Lahaye d.j.p.lahaye@tudelft.nl	Project title Turbulent combustion in a rotary kiln Project aim The aim of this project is to predict the temperature and the radiative heat flux in a rotary kiln used for the production of cement by our industrial partner. The kiln is fired by the non-premixed combustion of gaseous fuel. Information on the heat release will allow to further optimize the production process while at the same time meeting ever more stringent conditions on the environmental footprint. Progress Progress was made in the modeling of the turbulent non-premixed combustion in three dimensions, including the post-processing for the NOx concentrations. 2020 Publications 1. D. Lahaye, M. el Abbassi, C. Vuik, M. Talice and F. Juretic, “Thermal NOx Mitigation by Changing the Secondary Air Injection Channel: A Case Study in the Cement Industry”, Fluids, 5(4), 220, 2020 (https://doi.org/10.3390/fluids5040220). 2. D. Lahaye, “Reducing Emissions of Industrial Furnaces: Two Study Cases in the Port of Rotterdam”, talk accepted for the WE-Heraeus-Seminar Fuels, Processes, and Combustion Physics in the Energy Transformation in March 2020.

<table>
<tr><th colspan="2">TUD – Applied Mathematics – Mathematical Physics</th></tr>
<tr><td>
<u>Project leaders</u>
D. Lahaye

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
Prajakta Nakate (PhD student)
C.Vuik

<u>Funded by</u>
AluChemie B.V. Rotterdam

<u>Funded %</u>
Industry 100 %

<u>Start of the project</u>
2017

<u>Information</u>
Domenico Lahaye
d.j.p.lahaye@tudelft.nl
</td><td>
<u>Project title</u>
Turbulent combustion in an anode baking furnace

<u>Project aim</u>
Our industrial partner employs furnaces for the baking of anodes for the production of aluminum. These furnaces are fired by the non-premixed combustion of gaseous fuel. The aim of this project is to predict the heat distribution and the pollutant formation in these furnaces. The outcome of the project is expected to allow to further optimize the production process while at the same time meeting ever more stringent conditions on the environmental footprint.

<u>Progress</u>
Progress was made in the modeling of non-premixed turbulent combustion in two dimensions and the modeling of non-reactive flow in three dimensions.

<u>2020 Publications</u>
1. D. Lahaye, P. Nakate, C. Vuik, M. Talice and F. Juretic, “Turbulent Non-Premixed Combustion in Single Heating Section of an Anode Baking Ring Furnace”, talk, 16th OpenFoam Workshop, June 8th - 11th 2021, Dublin, Ireland.
2. P. Nakate, D. Lahaye, C. Vuik and M. Talice, “Analysis of the Aerodynamics in the Heating Section of an Anode Baking Furnace using Non-Linear Finite Element Simulations, Fluids , 2021 (https://doi.org/10.3390/fluids6010046).
</td></tr>
</table>

TUD – Aerospace Engineering– Aerodynamics

Prof.dr. F Scarano
Prof.dr. S Hickel

The research in the Aerodynamics Group involves fundamentals of Fluid Dynamics and its applications to aerodynamic problems of relevance in Aerospace Engineering systems.

The activities cover boundary layer research in low speed and high-speed flows, including re-entry aero-thermodynamics, complex unsteady flows, fluid structure interaction problems and aeroacoustics.

The group works in close connection with the Wind Energy section for the investigation of rotor blade aerodynamics. Specific flow control strategies by passive (e.g. vortex generators) and active means (suction, plasma actuators) are explored for their application in flow transition and separation delay.

The experimental research is supported by the Aerodynamics Laboratories, which cover flow simulation range from incompressible to hypersonic regime. Emphasis is given to the development and application of image based advanced flow diagnostic techniques like Tomographic PIV, Background Oriented Schlieren, InfraRed Thermography. The study of Fluid-Structure interactions and of unsteady flow simulation drive the development of efficient simulation tools for 3D-unsteady viscous flows (e.g. adaptive meshing, mimetic methods, multiscale computation of turbulence). Applications range from flapping wings and micro aerial vehicle aerodynamics to aircraft flutter and unsteady loads on wind turbines. The research on CFD also covers quantification of uncertainties in aerodynamics problems simulation.

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Prof. Stefan Hickel
Dr. Steven Hulshoff

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Xiaodong Li
Nils Barfknecht

Funded by
CSC

Funding %
Scholarships 100%

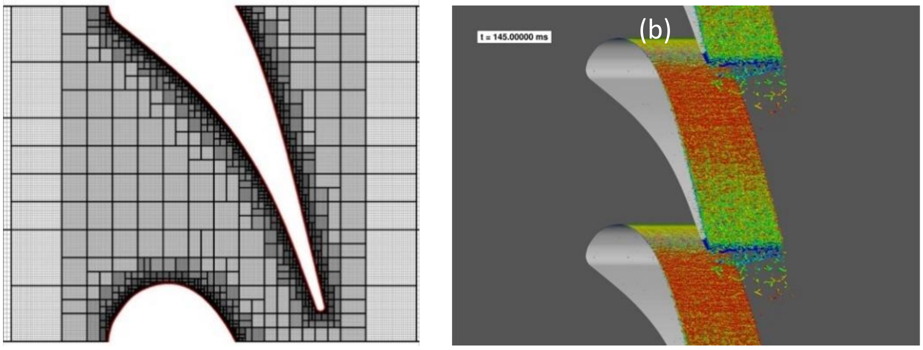
Start of the project
2017

Information
Stefan Hickel
S.Hickel@tudelft.nl

Project title
Efficient Adjoint Approach to Automatic Mesh Optimization for Predictive Large Eddy Simulation

Project aim
LES is highly successful in situations where the computational mesh is fine enough to resolve an appropriate range of scales. Currently, only highly experienced engineers are able to generate meshes that ensure the required prediction accuracy at tractable computational cost. The need for manual input is so far preventing the application of LES in automatic design optimization and limiting the practical application of LES for the validation and verification of design decisions. This research project will address the above-mentioned challenges and lead to intervention-free high-fidelity simulation methods for rigorous design assessment and optimization.

Progress
Over the last year we have focused on the representation of the unsteady primal solution within the solution of the adjoint equations. Effective online reduced-order representations have been developed which can be scaled to large problems. Our current test simplified test problems show large increases in efficiency are possible with little effect on adaptive mesh optimization patterns.



Adaptive-grid LES of a transonic nozzle cascade: (a) Cartesian AMR grid; (b) Iso-surfaces of lambda 2 criterion visualizing instantaneous coherent structures (Hickel 2017, unpublished).

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. A. Sciacchitano
Prof. F. Scarano

Research theme
Complex structures of fluids

Participants
D. Engler Faleiros
C. Jux
E. Saredi

Cooperation
Dutch Aerospace Center NLR
LaVision GmbH

Funded by
NLR, LaVision GmbH, University

Funding %

University	25%
NLR	25%
Industry	50%

Start of the project
2013

Information
Andrea Sciacchitano
a.sciacchitano@tudelft.nl

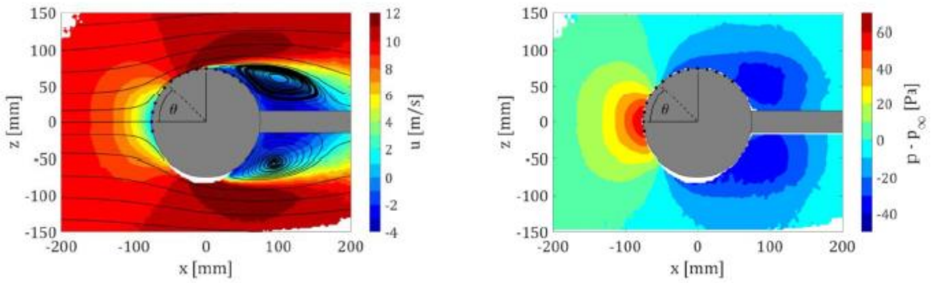
Project title
Large-scale Particle Image Velocimetry

Project aim
The Project aims at developing large-scale Particle Image Velocimetry (PIV) for three-dimensional measurements over large scales (several liters, up to the size of the human body). Sub-millimetre helium-filled soap bubbles (HFSB) are used as flow tracers due to their high light scattering efficiency. The project investigates the properties and production of the HFSB, as well as dedicated approaches for the flow measurements based on co-axial arrangement between imaging and illumination, and robotic manipulation of the measurement system. The technique is applied for flow measurements in the field of automotive and aeronautics.

Progress
A novel approach has been developed for the estimation of the surface pressure on solid objects of generic shapes. The approach is based on the evaluation of the pressure field by Bernoulli equation in regions of irrotational flow, and the integration of the pressure gradient from the rotational flow external boundary towards the object's surface.
A multi-Δt approach has been introduced which aims at increasing the dynamic velocity range of 3D-PTV measurements in high-speed flows, where the limited temporal resolution of the measurement system hinders tracking the tracer particles for more than two time instants. The approach is based on a predictor-corrector scheme, where measurements are conducted at a small time separation to produce a robust yet imprecise predictor; then, measurements with a larger time separation yield a precise corrector which enhances the measurement dynamic velocity range.
The large-scale robotic PTV technique has been applied for the investigation of the flow around automotive side-mirror models.

2020 Dissertations
Faleiros DE. Soap bubbles for large-scale PIV: Generation, control and tracing accuracy, PhD Dissertation

2020 Publications
Jux C, Sciacchitano A, Scarano F. Flow pressure evaluation on generic surfaces by robotic volumetric PTV. Measurement Science and Technology. 2020 Jul 8;31(10):104001.
Kim D, Kim M, Saredi E, Scarano F, Kim KC. Robotic PTV study of the flow around automotive side-view mirror models. Experimental Thermal and Fluid Science. 2020 Nov 1;119:110202.
Saredi E, Sciacchitano A, Scarano F. Multi-Δt 3D-PTV based on Reynolds decomposition. Measurement Science and Technology. 2020 May 26;31(8):084005.



TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. BW van Oudheusden

Research theme
Complex dynamics of fluids

Participants
M Percin

Cooperation
Middle Eastern Technical University

Funding %
University 100%

Start of the project
2010

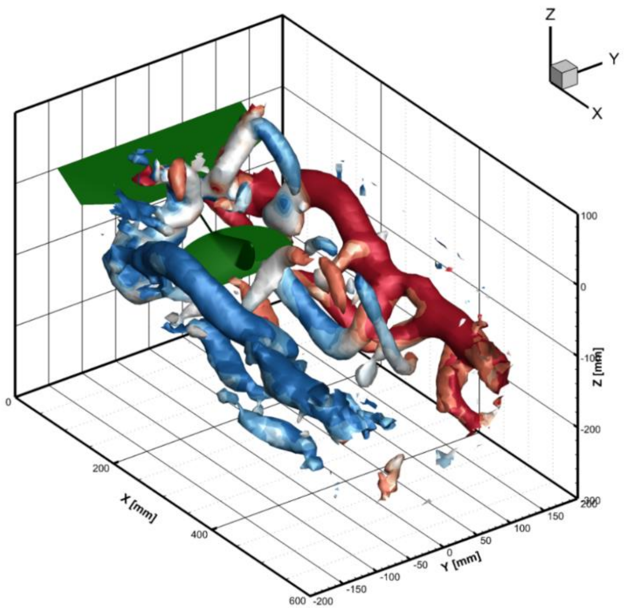
Information
BW van Oudheusden
B.W.vanOudheusden@tudelft.nl.

Project title
Flapping-flight propulsion

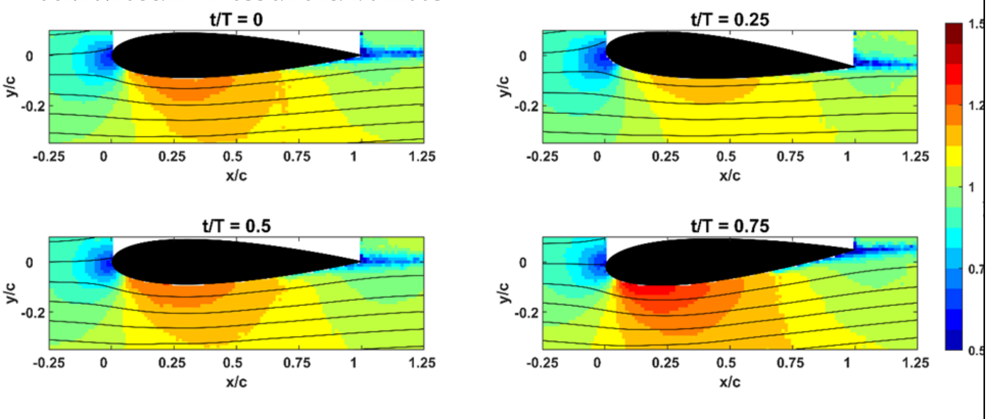
Project aim
The project considers fundamental and applied research on aerodynamic characterization of flapping-wing propulsion for the flight regime of small Micro Aerial Vehicles (Re ~ 10,000). Specific challenges are the occurrence of highly unsteady flow features resulting from massive separation, wing-wing interaction and the high amount of wing flexibility.

Progress
The work has two major fields of attention: 1) the study of generic aeroelastic phenomena relevant to MAV propulsion and 2) a more detailed characterization of the aerodynamic behaviour of the flapping-wing DelFly MAV itself. PIV wind tunnel studies were directed towards the characterization of the DelFly in both near-hover and forward flight configurations. Large-scale and free-flight visualization studies were performed using helium-filled soap bubbles.

2020 Publications
B Martínez Gallar, BW van Oudheusden, A Sciacchitano, M Karásek: Large-scale volumetric flow visualization of the unsteady wake of a flapping-wing micro air vehicle, Experiments in Fluids, 61:16 (2020).
DNWM Heitzig, BW van Oudheusden, D Olejnik, M Karasek: Effects of asymmetrical inflow in forward flight on the deformation of interacting flapping-wings, International Journal of Micro Air Vehicles, Volume 12: 1–11 (2020).



Large-scale flow visualization of a flapping-wing Micro Air Vehicle (Blanca Martinez Gallar, Experiments in Fluids, 2020)

TUD – Aerospace Engineering– Aerodynamics	
Project leaders Dr. BW van Oudheusden Dr. A Sciacchitano Prof. F Scarano Dr. FFJ Schrijer Researchtheme Complex dynamics of fluids Participants G Gonzalez C Mertens A D‘Aguanno Cooperation DLR, CNRS, Uni BW Munich, ONERA, Univ. Southampton, LaVision GmbH Funded by EU (H2020-HOMER) Funding % University 20 % EU 80 % Start of the project 2018 Information BW van Oudheusden B.W.vanOudheusden@tudelft.nl.	Project title Non-Intrusive Optical Metrology for Aeroelastic Analysis Project aim Novel non-intrusive experimental procedures are developed and applied in the analysis of aeroelastic problems (flutter, buffet). An integrated measurement approach is followed that characterizes fluid-dynamic and structural aspects simultaneously, with a single measurement system. Extensive use is made of Robotic PIV that allows both flow tracers and surface markers to be tracked. This data is subsequently used to establish all forces involved (aerodynamic, elastic and inertial). Progress 1) Assessment of different experimental configurations (flow over an actuated surface panel, wing-flap in pitching excitation and gust response of a flexible wing). Classic techniques are used for validation. 2) Advancement of data assimilation to improve measurement spatial resolution. 3) Investigation of the instability mechanism and control of transonic buffet. 2020 Publications C Mertens, A Sciacchitano, BW van Oudheusden, J Sodja: Non-intrusive determination of the unsteady surface pressure and aerodynamic loads on a pitching airfoil, 10th EASN International Conference on “Innovation in Aviation & Space to the Satisfaction of the European Citizens”, 2-4 Sept 2020, online conference. A Schröder, B Leclaire, BW van Oudheusden, L Chatellier, CJ Kähler: HOMER - Holistic optical metrology for aeroelastic research, 10th EASN International Conference on “Innovation in Aviation & Space to the Satisfaction of the European Citizens”, 2-4 Sept 2020, online conference. J. Fernández Barrio, C. Mertens, D. Ragni, A. Schiacchitano and B. van Oudheusden: Pressure based active load control of a blade in dynamic stall conditions, TORQUE 2020 (online), 28 Sept. – 2 Oct. 2020; Journal of Physics: Conference Series ,Vol.1618, 022003, doi:10.1088/1742-6596/1618/2/022003  *Non-intrusive determination of the unsteady aerodynamic loads on a pitching airfoil: flow field and airfoil position measurement.*

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. BW van Oudheusden
Prof. F Scarano
Dr. A Sciacchitano
Dr. FFJ Schrijer

Research theme
Complex dynamics of fluids

Participants
G Gonzalez
C Mertens
C Jux

Cooperation
DNW, CNRS, DLR, Uni BW Munich, ONERA, Univ. Southampton, LaVision GmbH

Funded by
EU (H2020-HOMER), LaVision

Funding %
University 20 %
Industry 20 %
EU 60 %

Start of the project
2006

Information
BW van Oudheusden
B.W.vanOudheusden@tudelft.nl.

Project title
PIV-based Non-Intrusive Determination of Pressure and Aerodynamic Loads

Project aim
Novel non-intrusive experimental approaches are developed and applied to determine the flow-field pressure, as well as the integral aerodynamic loads on objects, from flow velocity data measured with particle image velocimetry (PIV). This is achieved by combining the flow field information with basic momentum and control-volume principles. Apart from looking at the fundamental principles (notably 3D flow effects) the project also addresses the development of practical procedures.

Progress
1) Pressure field determination for large-scale flows based on the helium-filled soap bubble PTV technique and its application in aeroelasticity.
2) Surface pressure evaluation on objects of complex shape by means of robotic volumetric PIV.

2020 Publications
C Mertens, A Sciacchitano, BW van Oudheusden, J Sodja: Non-intrusive determination of the unsteady surface pressure and aerodynamic loads on a pitching airfoil, 10th EASN International Conference on “Innovation in Aviation & Space to the Satisfaction of the European Citizens”, 2-4 Sept 2020, online conference.
FFJ Schrijer and BW van Oudheusden: Application of pressure from PIV for compressible base-flows (invited), Final colloquium of the collaborative research center TRR40, 16 & 17 Nov 2020, online conference

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. BW van Oudheusden
Prof. F Scarano
Dr. A Sciacchitano
Dr. FFJ Schrijer

Researchtheme
Complex dynamics of fluids

Participants
G Gonzalez
C Mertens
C Jux

Cooperation
DNW, CNRS, DLR, Uni BW Munich, ONERA, Univ. Southampton, LaVision GmbH

Funded by
EU (H2020-HOMER), LaVision

Funding %

University	20 %
Industry	20 %
EU	60 %

Start of the project
2006

Information
BW van Oudheusden
B.W.vanOudheusden@tudelft.nl.

Project title
PIV-based Non-Intrusive Determination of Pressure and Aerodynamic Loads

Project aim
Novel non-intrusive experimental approaches are developed and applied to determine the flow-field pressure, as well as the integral aerodynamic loads on objects, from flow velocity data measured with particle image velocimetry (PIV). This is achieved by combining the flow field information with basic momentum and control-volume principles. Apart from looking at the fundamental principles (notably 3D flow effects) the project also addresses the development of practical procedures.

Progress
1) Pressure field determination for large-scale flows based on the helium-filled soap bubble PTV technique and its application in aeroelasticity.
2) Surface pressure evaluation on objects of complex shape by means of robotic volumetric PIV.

2020 Publications
C Mertens, A Sciacchitano, BW van Oudheusden, J Sodja: Non-intrusive determination of the unsteady surface pressure and aerodynamic loads on a pitching airfoil, 10th EASN International Conference on “Innovation in Aviation & Space to the Satisfaction of the European Citizens”, 2-4 Sept 2020, online conference.
FFJ Schrijer and BW van Oudheusden: Application of pressure from PIV for compressible base-flows (invited), Final colloquium of the collaborative research centre TRR40, 16 & 17 Nov 2020, online conference

TUD – Aerospace Engineering– Aerodynamics

<u>Project leaders</u>
Dr. M. Kotsonis

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Dr. Theo Michelis (promoted 2017)
John Kurelek (defence 2020)
Srikar Yadala Venkata (defence 2020)
Alberto Vidales (defence 2021)
Kaisheng Peng (defence 2023)
Giulia Zoppini (defence 2023)
Jordi Casacuberta (defence 2023)
Sven Westerbeek (defence 2024)
Alessandro Della Pia (defence 2023)

<u>Cooperation</u>
Dr. Serhiy Yarusevych (U.Waterloo)
Dr. Nicolas Benard (U. Poitiers)
Dr. Markus Kloker (U. Stuttgart)
Prof. Kozo Fujii (Tokyo Univ. Science)
Prof. Luigi de Luca (U. Napoli)

<u>Funded by</u>
NWO, Embraer, ERC

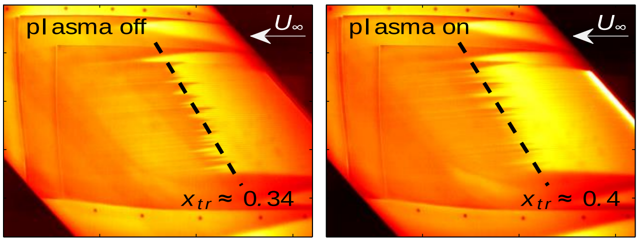
<u>Funding %</u>
University 10%
Industry 20%
EU 70%

<u>Start of the project</u>
2013

<u>Information</u>
Name: Marios Kotsonis
E-mail: m.kotsonis@tudelft.nl

<u>Project title</u>
Advanced flow diagnostics and flow control for swept wing transition

<u>Project aim</u>
This is a long running Project aimed at the investigation of crossflow (CF) instabilities, widely acknowledged as the leading mechanism behind swept wing transition. While extremely important for the efficiency of modern aircraft, several aspects of CF-dominated transition are currently unknown. The Project aims at closing this gap through advanced spatio-temporal measurements and BiLocal stability analysis of primary and secondary crossflow instabilities. A second aim is the development of viable active control methods to mitigate the growth of CF instabilities, thus leading to transition delay and drag reduction.



Thermal imaging of swept wing transition controlled by plasma actuators (Re_c = 2.2 million)

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. Davide Modesti
Prof. Stefan Hickel

Research theme
Complex dynamics of fluids

Participants
Haris Shahzad

Funded by
TU Delft

Funding %
University 100%

Start of the project
2020

Information
Davide Modesti
D.Modesti@tudelft.nl

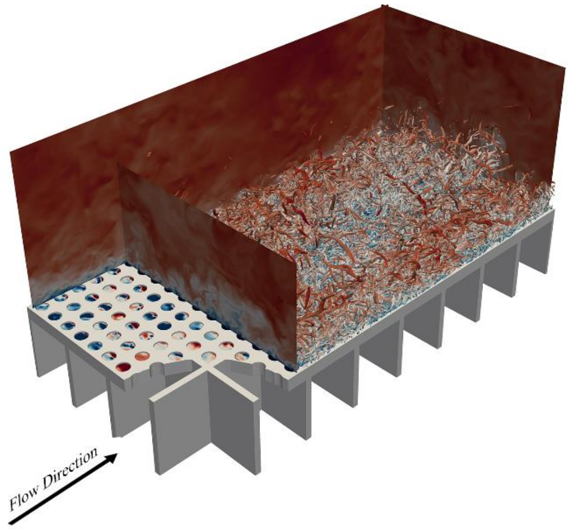
Project title
Turbulent Flows over Acoustic Liners

Project aim
The Project aims at understanding the physics of turbulent flows over acoustic liners for aircraft engines. Acoustic liners constitute the state-of-the-art technology for acoustic noise abatment over turbofan engines, but they have the drawback of increasing aerodynamic drag compared to a smooth wall. During the project, we will carry out unprecedented direct numerical simulations (DNS) of turbulent flows over fully resolved acoustic liners and focus on the flow physics of these surfaces. The project is divided into two main phases. In the first phase we generated a DNS dataset of turbulent channel flow for different surface porosities and Reynolds numbers to quantify the added drag and parametrize this flow. In the second phase we will perform DNS turbulent boundary layer flow over acoustic liners with and without acoustic waves, to help understand all facets of acoustic liner behaviour.

Progress
We have generated a DNS dataset of turbulent channel flow over fully resolved acoustic liners for different surface porosities σ=0.0357–0.322 and friction Reynolds numbers Re_τ=500–2000, at bulk Mach number M_b= 0.3.. The dataset comprises five liner flow cases in which the ratio between the orifice diameter and channel half width is constant d/δ=0.8 and the porosity and Reynolds number are progressively increased. Simulations have been carried out using our compressible flow solver STREAmS and the complexity of the geometry is handled using an immersed boundary method. As customary for flows over rough walls, we measure the drag increase using the Hama roughness function ΔU^+ because it is fairly independent from the Reynolds number, unlike the friction coefficient. Preliminary results show evidence of a fully rough regime for these surfaces, whose onset is determined by the value of the viscous-scaled square root of the permeability. Having identified as a promising length scale for acoustic liner behaviour, we aim to further understand its role by performing turbulent channel flow simulations of additioanal liner geometries, for instance changing the facesheet thickness and further expanding the range of surface porosity.

2020 Publications

H. Shahzad, S. Hickel, D. Modesti Turbulence over acoustic liner roughness, Proceedings of the 13th International Ercoftac Symposium on Engineering Turbulence Modelling and Measurements , Rhode 2021.



Instantaneous flow field from DNS of turbulent channel flow at Re_τ= 2000 and bulk Mach number M_b= 0.3. The streamwise velocity is shown in a streamwise and cross-stream plane, whereas the pressure is shown inside the orifices. Vortical structures are visualised using the Q-Criterion.

TUD – Aerospace Engineering– Aerodynamics	
Project leaders Dr. F.F.J Schrijer Dr. B.W. van Oudheusden Prof. S. Hickel Research theme Complex dynamics of fluids Participants A D'Aguanno L Laguarda Sanchez Cooperation ESA, VKI, DLR, NLR, Airbus, ArianeGroup, TNO, ONERA, UNINA Funded by EU (HOMER, CS2) Funding % University 20% EU 80% Start of the project 2003 Information F. Schrijer f.f.j.schrijer@tudelft.nl	Project title Experimental high-speed flows and compressible turbulence Project aim Development and implementation of non-intrusive measurement techniques for high-speed compressible flows with special focus on particle image velocimetry. The measurement techniques are subsequently used to investigate flow phenomena that are pertinent to the high-speed aerospace technology domain, such as including launchers, high-speed air transport systems and reentry vehicles. Specific topics that are studied are shockwave boundary layer interaction (SWBLI), transonic buffet control, (compressible) roughness-induced boundary layer transition, supersonic fluid structure interaction and compressible baseflows. Progress 1) Study and control of transonic and supersonic shock-wave boundary layer interactions 2) Investigation of supersonic jet in cross flow 3) Launcher base flows, using (tomographic) PIV and PIV-based pressure integration 4) Unsteady transonic airfoil aerodynamics and buffet control 5) Investigation of supersonic shock-shock interactions 6) Supersonic fluid structure interaction 2020 Publications Dynamics of unsteady asymmetric shock interactions, Laguarda, L., Hickel, S., Schrijer, F. F. J. & Van Oudheusden, B. W., 2020, In: Journal of Fluid Mechanics. 888, 22 p., A18. Theoretical study on regular reflection of shock wave-boundary layer interactions, Xue, L., Schrijer, F. F. J., Van Oudheusden, B. W., Wang, C., Shi, Z. & Cheng, K., 2020, In: Journal of Fluid Mechanics. 899, 20 p., A30.

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Prof. Stefan Hickel

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Mohamad Fathi
Dr. Maria Chertova
Dr. Nicolas Renaud
Prof. Dirk Roekaerts

Cooperation
eScience Centre Amsterdam

Funded by
JCER eScience for Energy Research

Funding %
NWO 100%

Start of the project
2012

Information
Stefan Hickel
S.Hickel@tudelft.nl

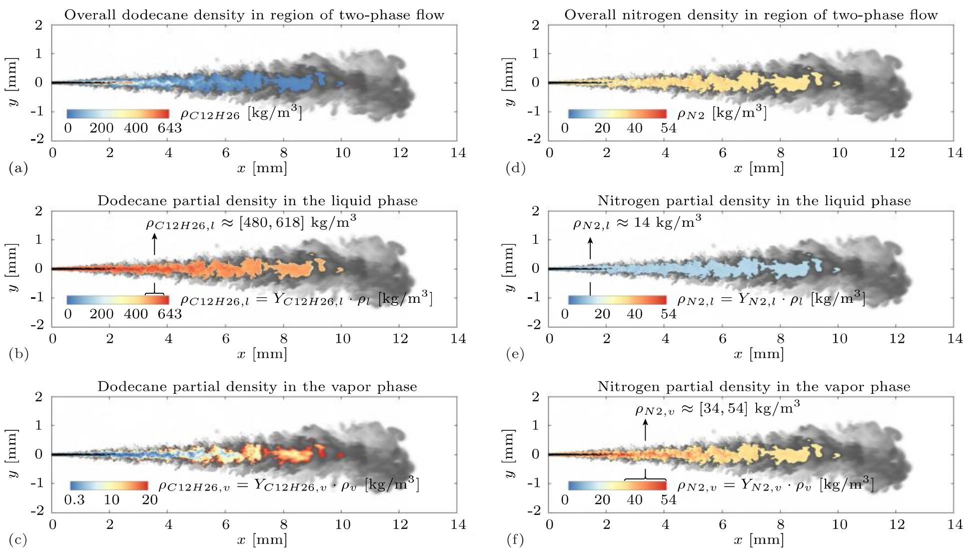
Project title
Scalable high-fidelity simulations of reacting multiphase flows at transcritical pressure

Project aim
We develop methods for scalable multi-physics massively parallel high-performance computing. The new methods will be applied for performing unprecedented large-eddy simulations (LES) of high-pressure liquid-fuel injection and reacting transcritical multiphase flows in modern energy conversion systems, such as rocket engines, gas turbines and diesel engines, to provide detailed insight into high-pressure-injection phenomena and contribute to the solid physical understanding necessary to further improve the efficiency of these technical systems.

Progress
Fathi and Hickel (2020) have significantly improved the computational efficiency and accuracy of a multispecies two-phase model for mixtures with very large numbers of species. The model is based on cubic equations of state and vapor-liquid equilibrium calculations (Matheis & Hickel, 2018) and can accurately represent supercritical states and multi-component subcritical two-phase states, ignition, and the transition between deflagration and detonation modes of combustion. We are also in final steps of publishing our work on dynamic multi-level parallelization for improving the efficiency and scalability of high-performance multi-physics simulations.

2020 Publications

Fathi, M., Hickel, S. (2020) Rapid multi-component phase-split calculations using volume functions and reduction methods. AIChE Journal, 67(6), e17174



Full phase information for VLE-LES of transcritical n-dodecane jet at 60bar. Left and right column show contours of dodecane and nitrogen partial densities, respectively, from blue to red shades. All cells with 0.1% > vapor volume fraction > 99.9% are blanked out and the background contour shows the temperature field from dark to light shades (Matheis & Hickel, 2018)

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Prof. Stefan Hickel
Dr. Bas van Oudheusden
Dr. Ferry Schrijer

Research theme
Complex dynamics of fluids

Participants
Luis Laguarda
Weibo Hu
Jaime Santiago Petterson

Funded by
Self, CSC

Funding %
University 50%
Scholarships 50%

Start of the project
2017

Information
Stefan Hickel
S.Hickel@tudelft.nl

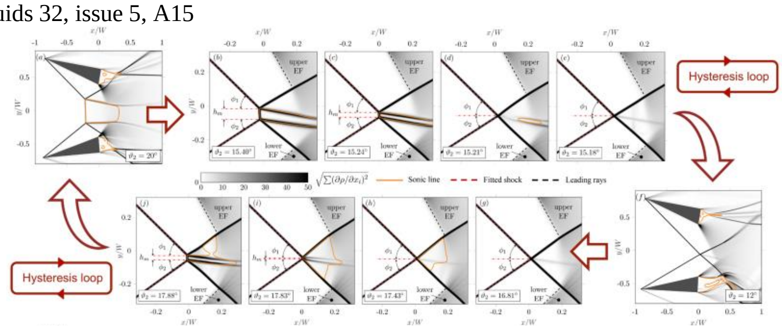
Project title
Computational and experimental investigation of unsteady asymmetric interactions of shock waves and between shock waves and boundary layers over a flexible plate

Project aim
Supersonic inlet diffusors and over-expanded rocket nozzles are characterized by complex shock interaction pattern, shock turbulence interactions and boundary layer separation, which result in unsteady and asymmetric loads that bend and deform thin-walled structure. As the deformed nozzle or inlet contour directly affects its internal flow field and thus the mean and dynamic flow-separation and side-load behaviour, these fluid-structure interactions can self-amplify and quickly cause structural failure. Our goal is to improve the understanding of the dynamics of these processes and the prediction capabilities of numerical tools.

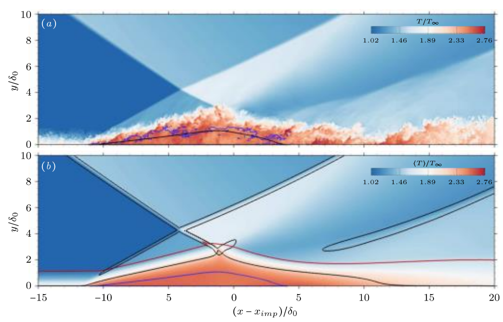
Progress
Laguarda et al (2020) found that finite time scales of the bi-directional transition process between two-shock and three-shock interaction pattern can determine the topology of the mean flow field of shock-wave/boundary-layer interactions. We have performed a theoretical, numerical, and experimental analysis of quasi 2-D shock-shock interactions. We found that and present theory based on steady equilibrium flow assumptions fails in the practically relevant case of unsteady or non-quiescent internal flow. Our preliminary results further indicated additional geometry effects, and strong effects of free stream noise in the wind-tunnel experiments.

2020 Dissertations
Weibo Hu: Dynamics of a supersonic flow over a backward/forward-facing step

2020 Publications
Laguarda, L., Hickel, S., Schrijer, F.F.J., Van Oudheusden, B.W. (2020) Dynamics of unsteady asymmetric shock interactions. Journal of Fluid Mechanics, 888, A18
Hu, W., Hickel, S., Van Oudheusden, B. (2020) Influence of upstream disturbances on the primary and secondary instabilities in a supersonic separated flow over a backward-facing step. Physics of Fluids 32, issue 5, A15



Asymmetric shock interaction hysteresis loop at M=3 (Laguarda & Hickel, 2017)


LES of shock/boundary-layer interaction at Mach 3 and Re_δ=200 000: (a) Instantaneous temperature contours; (b) Mean temperature contours (Pasquariello et al. 2017).

TUD – Aerospace Engineering– Aerodynamics

Project leaders
Dr. A. Sciacchitano
Prof. F. Scarano

Research theme
Complex structures of fluids

Participants
W. Terra
A. Spoelstra

Cooperation
Team Sunweb
TeamNL / NOC*NSF
TU Delft Sports Engineering Institute

Funded by
ERC Proof of Concept Grant 665477
NWO-TTW OTP Grant 15583

Funding %
TTW 50%
EU 50%

Start of the project
2015

Information
Andrea Sciacchitano
a.sciacchitano@tudelft.nl

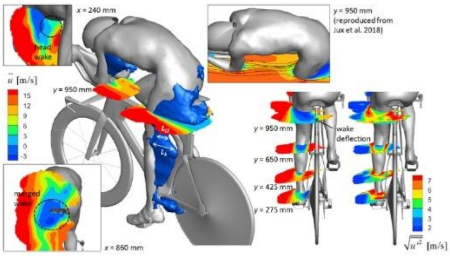
Project title
Sport Aerodynamics Investigation by Quantitative Flow Visualization

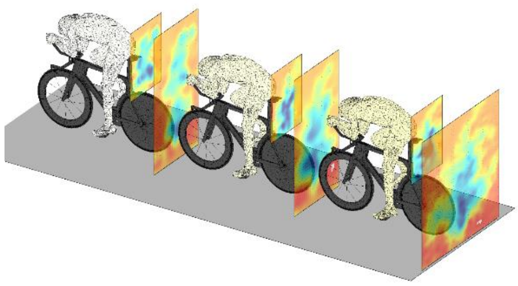
Project aim
This Project aims at investigating speed sports aerodynamics by quantitative flow visualization. Velocity measurements are conducted via large-scale Particle Image Velocimetry (PIV) both in wind tunnels and on-site during the athletes' training, using the innovative "ring of fire" concept. From the velocity measurements, the aerodynamic loads and their distribution over the athlete's body are evaluated via application of the governing equations of fluid motion, in particular the conservation of momentum. The flow measurements provide information on the areas that are most critical for the generation of drag, thus enabling design optimizations that enhance the athlete's performance.

Progress
The Reynolds number effects on body limbs of a cyclist model, namely leg and arm, have been investigated via robotic volumetric Particle Image Velocimetry measurements in the velocity range from 5 m/s to 25 m/s. Variations of the wake width with the Reynolds number have been observed and related to the variation of the local drag coefficient, indicating a drag crisis behaviour on both leg and arm.
The ring of fire system has been deployed for the on-site evaluation of the drag of cyclists during drafting. A mathematical model has been introduced which predicts the aerodynamic drag of the trailing rider based on his/her relative position (longitudinal and lateral offset) with respect to the leading rider.

2020 Dissertations
Terra W, Cyclist Aerodynamic Drag Analysis through Large-Scale PIV, PhD Dissertation

2020 Publications
Spoelstra A, Mahalingesh N, Sciacchitano A. Drafting Effect in Cycling: On-Site Aerodynamic Investigation by the 'Ring of Fire'. In Multidisciplinary Digital Publishing Institute Proceedings 2020 (Vol. 49, No. 1, p. 113).
Terra W, Sciacchitano A, Scarano F. Cyclist Reynolds number effects and drag crisis distribution. Journal of Wind Engineering and Industrial Aerodynamics. 2020 May 1;200:104143.





TUD – Aerospace Engineering– Aeroacoustics

Prof.dr.ir. D Casalino

The group's research includes the study of noise generated by turbulent flows passing over the trailing edge of an airfoil, aero-engine fan noise, noise mitigation through acoustic liners, noise generation by axial/vertical wind turbines, and noise generation by ducted wind turbines.

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Daniele Ragni, Dr. Francesco Avallone, Dr. Damiano Casalino, Dr. Mirjam Snellen, Dr. Sybrand van der Zwaag

Research theme
Complex dynamics of fluids

Participants
Leandro Rego, Salil Luesutthiviboon, Alejandro Rubio Carpio

Cooperation
VKI, KU Leuven, DNW, GE\LM Wind Power

Funded by
NWO

Funded %
NWO 100 %

Start of the project
2018

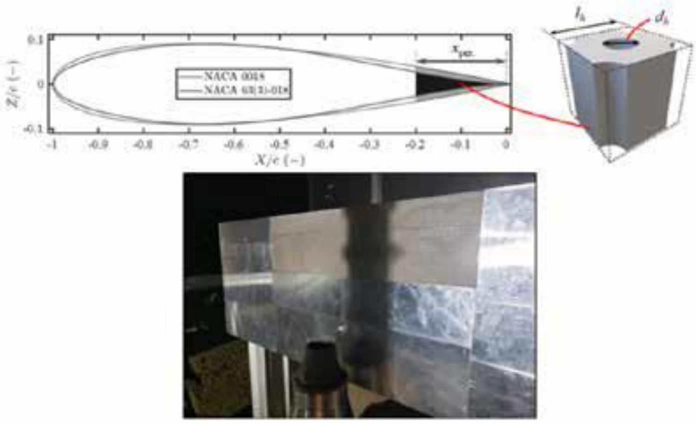
Information
L.Rego@tudelft.nl
A.RubioCarpio@tudelft.nl
S.Luesutthiviboon@tudelft.nl

Project title
Innovative permeable materials design for wind turbine and aircraft applications

Project aim
In the framework of IPER-MAN Innovative PERmeable Materials for Airfoil Noise reduction, the goal of this project is the reduction of airfoil self-noise emissions (e.g. turbulent boundary layer trailing edge noise) and turbulence impingement noise using 3D printed open cell porous materials with customized shapes and inner topologies. One of the applications of this research is to reduce aerodynamic noise emitted by wind turbine blades at high Reynolds numbers (Re > 106). Another application is the noise due to interaction between the exhaust flow of an aircraft turbofan engine and a deployed flap, which is known as jet installation noise.

Progress
For both airfoil self noise and jet-installation noise applications, experiments and CFD simulations with metal foams and 3D printed perforated trailing edges have successfully shown that significant noise reduction can be achieved. These materials can also be combined with Kevlar sheets to avoid the occurrence of tonal noise and reduce performance degradation. The physical mechanisms by which those materials provide noise reduction are currently being studied and focus has been given to the creation of fast aerodynamic and aeroacoustic models that can accurately predict the effect of porous materials on trailing edge noise.

2020 Publications
1. A. Rubio Carpio, F. Avallone, D. Ragni, M. Snellen, S. Van der Zwaag; Quantitative Criteria to Design Optimal Permeable Trailing Edges for Noise Abatement; Journal of Sound & Vibration 485 (2020) 115596; https://doi.org/10.1016/j.jsv.2020.115596.
2. L. Rego, F. Avallone, D. Ragni, D. Casalino, Jet-installation noise and nearfield characteristics of jet-surface interaction, Journal of Fluid Mechanics 895, 2020. doi:10.1017/jfm.2020.294.



Airfoils with perforated materials and application to jet-installation noise reduction of porous foams

TUD – Aerospace Engineering– Aeroacoustics

<u>Project leaders</u>
Dr. Damiano Casalino, Dr. Daniele Ragni, Dr. Francesco Avallone, Dr. Christophe Schram

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Christopher Teruna
Lourenço Tércio Lima Pereira

<u>Cooperation</u>
TUD, Siemens Gamesa, VKI, ECL, KTH, University of Southampton, KUL, NTUA, CETIAT, IMP-PAM, EPFL, CNRS/LAUM

<u>Funded by</u>
EU : Horizon

<u>Funded %</u>
EU 100 %

<u>Start of the project</u>
2017

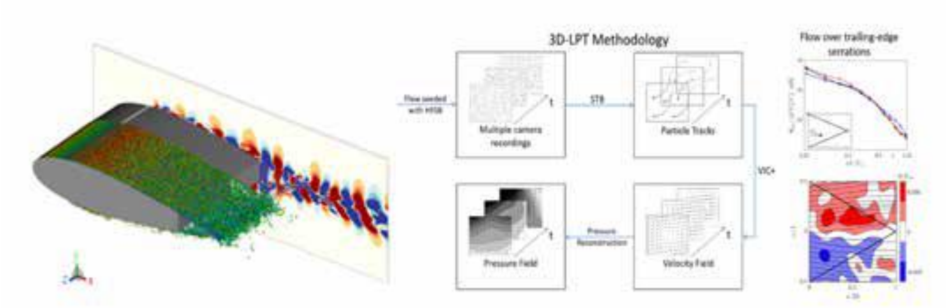
<u>Information</u>
L.T.LimaPereira@tudelft.nl
C.Teruna@tudelft.nl

<u>Project title</u>
Industrial applications of smart mitigation technologies for flow induced acoustic radiation

<u>Project aim</u>
In the framework of SMARTAnswer: Smart Mitigation of flow-induced Acoustic Radiation and Transmission for reduced Aircraft, surface traNSport, Workplaces and wind enERgy noise, new technologies have been emerging to reduce noise emissions, based on new concepts for flow and acoustic control. Beside the phenomenological understanding, the Project aims at modelling and optimizing the novel noise mitigation strategies for full scale industrial application. Additionally, the project fosters a young researchers network with industrial stakeholders from the aeronautical, automotive, wind turbine and cooling/ventilation sectors.

<u>Progress</u>
The work has already explained many noise reduction mechanisms for the application of porous materials and serrations for airfoil self-noise attenuation. Numerical and experimental techniques are employed for studying the flow surrounding porous/serrated surfaces. In the work of Teruna et al. 2020b, numerical methods are employed to study the flow field surrounding an airfoil equipped with a porous extension at the trailing-edge. In the work of Lima Pereira et al. 2020, experimental techniques are explored, based on the volumetric particle image velocimetry technique (3D-PIV), to study the velocity and pressure fluctuations at the vicinity of a serrated trailing-edge.

<u>2020 Publications</u>
1. C. Teruna, F. Manegar, F. Avallone, D. Ragni, D. Casalino, T. Carolus, Noise reduction mechanisms of an open-cell metal-foam trailing edge, Journal of Fluid Mechanics, 898, DOI:10.1017/jfm.2020.363.
2. C. Teruna, F. Avallone, D. Casalino, D. Ragni, Numerical Investigation of Leading Edge Noise Reduction on a Rod-Airfoil Configuration Using Porous Materials and Serrations, Journal of Sound and Vibration, 494, DOI: 10.1016/j. jsv.2020.115880.
3. L. T. Lima Pereira, D. Ragni, F. Avallone, F. Scarano, Pressure fluctuations from large-scale PIV over a serrated trailing edge, Experiments in Fluids, 61/3, DOI:10.1007/s00348-020-2888-x



Left: Instantaneous flow and pressure field on the top of an airfoil model equipped with a porous insert (Teruna et al. 2020b); right: schematic flow chart of the 3D PIV methodology employed and the results obtained (Lima Pereira et al. 2020).

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Daniele Ragni, Dr. Francesco Avallone, Dr. Tomas Sinnige, Dr. Woutijn Baars, Dr. Christophe Schram

Research theme
Complex dynamics of fluids

Participants
Fernanda Monteiro, Hasse Dekker

Cooperation
VKI, ECL, NLR, UBristol, UTwente

Funded by
EU: Horizon 2020

Funded %
EU 100 %

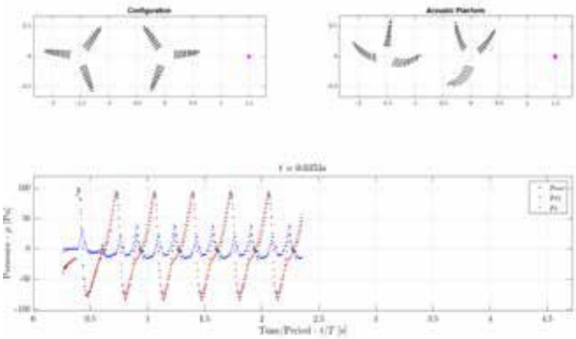
Start of the project
2020

Information
f.donascimentomonteiro@tudelft.nl
h.n.j.dekker-1@tudelft.nl

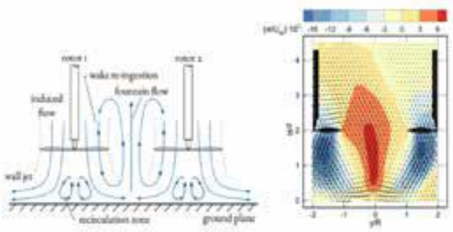
Project title
Aeroacoustics of multi rotors in cruise and vertical take off conditions

Project aim
In the framework of ENODISE Enabling Optimized DISruptivE Airframe- Propulsion Integration, the project is aimed at studying the integration of novel propulsion systems with the airframe. The complex aerodynamic and aeroacoustic engine airframe interactions are investigated at a low TRL to build a solid basis of knowledge and methods based on simplified but representative configurations, one of which being distributed propulsion. The goal of the current investigation is to study acoustic and aerodynamic installation effects of a distributed propeller wing system in a tractor configuration and to investigate aerodynamic interactions during the vertical take off and landing of multi rotor systems.

Progress
An analytical code to predict tonal noise of an array of propellers considering aerodynamic and acoustic interference effects is being implemented and it is under process of validation using in house experimental data. Radial wake expansion and the resulting formation of an upward moving flow known as the fountain flow, as illustrated in the figure on the left, are associated with the vertical take off and landing of multi rotors. These aerodynamic interactions have been investigated by means of volumetric velocimetry, using Helium Filled Soap Bubbles, see the figure on the right.



Analytical tool displaying the configuration simulated, its acoustic planform and the acoustic pressure in function of time, with its thickness and loading contributions. The magenta marker shows the observer position.



Conceptual flow topology of a side-by-side rotor in ground effect (left) and axial velocity contours (right).

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Daniele Ragni, Dr. Francesco Avallone, Dr. Christophe Schram

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Andrea Piccolo, Jose Manoel Freire Guimarães, Umberto Boatto

Cooperation
TUD, Siemens Gamesa, VKI, NTUA, NTU, SAMTECH, UPM, UTW, WU, IMP-PAM

Funded by
EU: Horizon 2020

Funded %
EU 100%

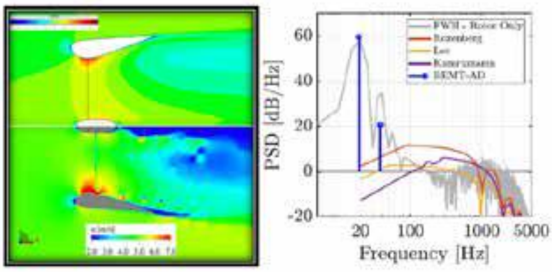
Start of the project
2020

Information
J.M.FreireGuimaraes@tudelft.nl
A.Piccolo@tudelft.nl
U.Boatto@tudelft.nl

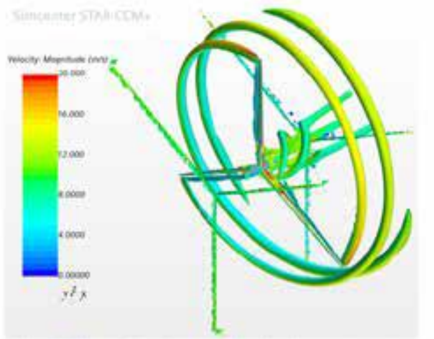
Project title
Aeroacoustics of on-shore wind-energy resources

Project aim
In the framework of zEPHYR: Towards a more efficient exploitation of on-shore and urban wind energy resources, the Project aims to investigate and further develop promising emerging technologies enabling a more efficient harvesting of wind energy resources in ‘conventional’ on shore as well as urban environments, through more accurate and robust simulation methodologies. The Project aims at: (i) the investigation of the aeroelastic behaviour of very onshore wind turbines (WT) considering the effect of atmospheric boundary layer, (ii) aeroacoustic optimization of ducted wind turbines (DWT) and (iii) effect of turbulent inflow conditions on the acoustic noise emission of WT.

Progress
The proposed low fidelity modeling of the DWT shows good agreement with high fidelity simulations, as shown in the figures. Additionally, a coupled CFD-FEA model was developed to analyze the aeroelastic behaviour of a 5MW wind turbine. Results were validated against the literature and the behaviour at a yaw flow is being investigated.



Left: Reynolds Averaged Navier Stokes (RANS) axial velocity contours for a ducted wind-turbine (top half) compared with Lattice Boltzmann Method results (bottom half); right: high-fidelity spectrum of the noise at 1 diameter distance and 90 degrees (gray) compared to BEMT (tonal)-approach with the RANS velocity in input, and to literature-based (broadband) noise contributions.



RANS visualization of the 5MW wind-turbine studied in the project, iso surface of the Q-criterion of the coupled model.

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Damiano Casalino, Dr. Daniele Ragni, Dr. Francesco Avallone

Research theme
Complex structures of fluids

Participants
Edoardo Grande

Cooperation
Parrot, UNICUSANO

Funded by
Airbus Defense and Space 100%

Funded %
Industry 100%

Start of the project
2018

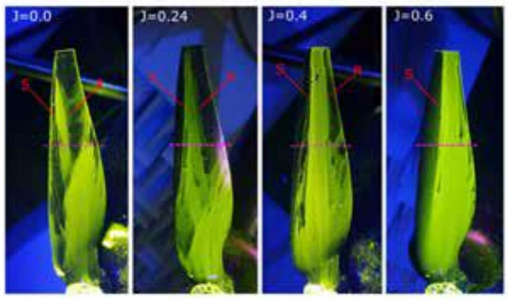
Information
E.Grande@tudelft.nl

Project title
Aerodynamic and aeroacoustic study of low-Reynolds number propellers

Project aim
Propellers operating at low Reynolds number are the most employed propulsive system for Unmanned Aerial Vehicles (UAVs). UAVs are developing incredibly fast, both for military and commercial operations, and their use can be severely limited by the noise generation levels. Therefore, the focus of the project is to identify the characteristic physical phenomena for a propeller operating at low Reynolds number and to understand how they influence the noise emitted, with the final goal of designing a quiet and efficient propeller. To this purpose, a combination of wind tunnel experiments, high fidelity simulations and analytical methods are used.

Progress
Experimental measurements and low-/high- fidelity numerical predictions on a propeller of 30 cm diameter operated at a rotational speed between 4000 and 6000 rpm and an axial velocity ranging from 0 to 24 m/s are available (advance ratio ranging from 0 to 0.8). A favorable agreement between measured and predicted loads and noise has been found. The experimental results reveal a complex flow field around the blade with the appearance of a laminar separation bubble on the suction side of the blade. The separation bubble causes vortex shedding and is responsible a high-frequency hump in the far field noise spectra.

2020 Publications
D. Casalino, E. Grande, G. Romani, D. Ragni, F. Avallone, "Definition of a benchmark for low Reynolds number propeller aeroacoustics", Aerospace Science and Technology.



Propeller setup mounted in the A-Tunnel at TU Delft.
Oil flow visualizations of the suction side of the blade at 4000 rpm and different advance ratios J

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Damiano Casalino, Dr. Daniele Ragni, Dr. Francesco Avallone

Research theme
Complex dynamics of fluids

Participants
Gianluca Romani, Qingqing Ye

Cooperation
DLR, ONERA, CIRA, ECL

Funded by
EU : Horizon 2020

Funded %
Industry 100%

Start of the project
2017

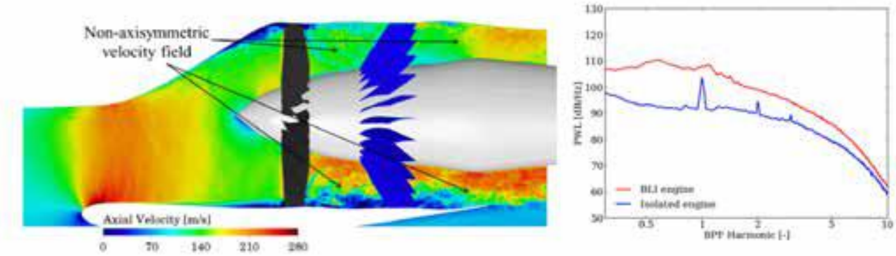
Information
g.romani@tudelft.nl

Project title
Aircraft Noise Reduction Technologies and Related Environmental Impact - ARTEM

Project aim
The goal of this project is to study the impact of Boundary-Layer Ingestion (BLI) on fan noise. BLI occurs when turbofan engines are semi buried in aft fuselage, with potential benefits in terms of propulsion efficiency/fuel savings. However, several drawbacks should be addressed before quantifying the actual benefits of BLI, such as the impact of the inlet flow distortion on engine efficiency and aeroacoustics. The acoustic assessment is performed using high-fidelity CFD/CAA simulations based on the lattice-Boltzmann method by predicting and comparing fan noise generation and propagation from a BLI and conventional engine on the ONERA NOVA aircraft.

Progress
The BLI fan-stage configuration is found to be characterized by strong azimuthal fan blade loading unsteadiness (left figure), less axisymmetric and coherent rotor wake tangential velocity variations and higher levels of in-plane velocity fluctuations compared to the isolated engine (left figure). This resulted in no distinct tonal components and higher broadband levels of the far field noise (right figure), as well as in an increment of cumulative noise levels up to 18 EPNdB.

2020 Publications
G. Romani, Q. Ye, F. Avallone, D. Ragni, D. Casalino, “Numerical analysis of fan noise for the NOVA boundary-layer ingestion configuration”, Aerospace Science and Technology, Volume 96, 2020.



Left: BLI fan stage simulation from the project ARTEM, right: far field noise comparison with respect to isolated engine.

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Dr. Mirjam Snellen, Dr. Daniele Ragni, Dr. Francesco Avallone

Research theme
Complex dynamics of fluids

Participants
Hugo Mourão Bento, Colin Van Dercreek

Cooperation
Siemens Gamesa, NLR, 3DS, Deutsche WindGuard

Funded by
NWO

Funded %
NWO 100%

Start of the project
2017

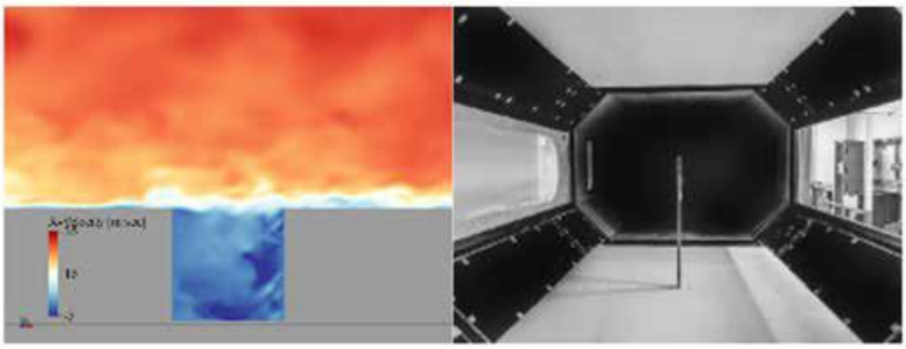
Information
h.f.mouraobento@tudelft.nl

Project title
Acoustic treatments for new aeroacoustic wind-tunnels

Project aim
Within the THAMES; Towards High-Reynolds Airfoil self-noise MEasurementS framework, aeroacoustic refurbishments of wind tunnels with closed test sections are investigated as the most suitable facilities for combined microphone/PIV measurements. The usefulness of these closed test section wind tunnels for acoustic measurements is, however, severely limited by their background noise. Still, combining aerodynamic and acoustic measurements in the same facility is highly preferable. To achieve this, new wind tunnel design solutions are proposed, with optimized cavities in which microphones are mounted, recessed from the TBL, and with new acoustic wall treatments.

Progress
a) Several microphone cavity design solutions have been studied, both experimentally and numerically. The main physical mechanisms which lead to the propagation of noise to the bottom of the cavity have been understood. New cavity designs have been proposed.
b) An acoustic modelling algorithm has been proposed, for predicting the improvement caused by applying new acoustic treatments to wind tunnel walls. Several wall treatments have been tested, in order to validate the algorithm.

2020 Publications
VanDercreek, C. P., et al. "Comparison of cavity geometries for a microphone array in an open–jet wind–tunnel experiment." 8th Berlin beamforming conference, March. 2020.



Left: cavity study for recessed microphone implementation via Lattice Boltzmann numerical simulations (3DS Simulia), Right: application of wind-tunnel lining in Low Speed Low Turbulence Tunnel (TU Delft).

TUD – Aerospace Engineering– Aeroacoustics

Project leaders
Francesco Avallone

Research theme
Complex dynamics of fluids

Participants
Damiano Casalino

Funded by
Aeroacoustic Research Consortium 100%

Funded %
Industry 100%

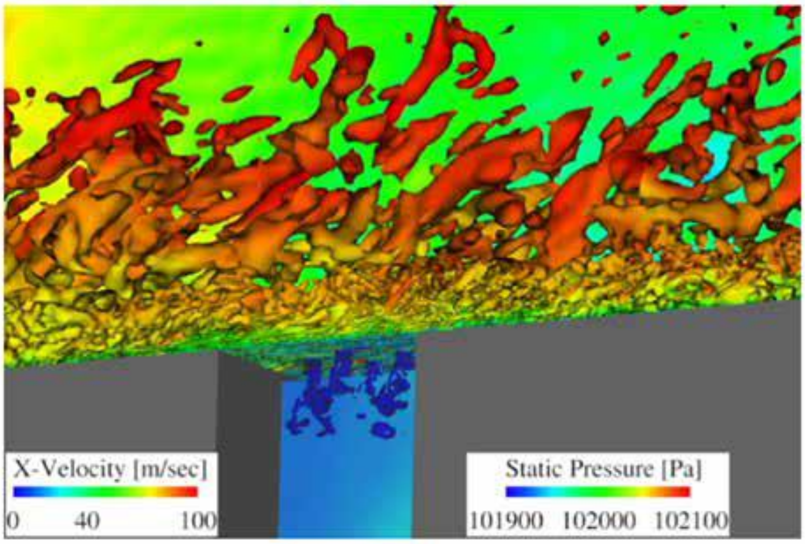
Start of the project
2020

Information
f.avallone@tudelft.nl

Project title
Acoustic Liners: Fundamental Understanding of the Physical Mechanisms and Physics-Based Low-Order Model

Project aim
The Project aims at performing high fidelity numerical simulations with the lattice-Boltzmann method of an acoustic liner grazed by a high Mach number turbulent flow and high amplitude acoustic wave in order to shed light on the physical description of the physical mechanisms and improve existing low-order models.

Progress
High fidelity numerical simulations have been carried out for acoustic wave propagating in the same direction of the mean flow and the opposite one. Numerical results have been assessed against experimental data. Results have shown that the acoustic impedance does not depend on the relative direction of the mean flow and the acoustic wave. From the results the acoustic-induced velocity has been extracted; it shows that the in-orifice velocity is very similar to the one obtained without grazing flow and that the main reason for the change in resistance between the two case is the different spatial distribution of the velocity within the orifice.



Instantaneous flow field from the simulations

TUD – Civil Engineering and Geosciences – Hydraulic Engineering – Environmental - Fluid Mechanics

Prof.dr.ir. AJHM Reniers
Prof.dr. JD Pietrzak
Prof.dr.ir. WSJ Uijttewaal

The Environmental Fluid Mechanics Group performs fundamental research with a focus on physical understanding of fluid flow problems ranging in scale from turbulence to geophysical fluid dynamics. We also help solve problems of practical relevance in water management, environmental engineering, hydraulic engineering and coastal engineering. We have a strong output of scientific papers and contribute to the numerical modelling community, with models such as SWAN (Simulating Waves Nearshore) and SWASH (Simulating Waves till Shore). Free surface flow models based upon unstructured grids are in preparation and will be released in the near future.

Within this philosophy the research program encompasses the following topics:

- Fluid dynamics our areas of research include shallow flows, turbulence and flow structures in rivers and their flood plains, stability and transport under waves and currents of rock elements in cover layers consisting of loose, granular material.
- Physical Oceanography our areas of research include mixing, internal waves, estuarine and coastal processes, large scale dynamics and climate, numerical modelling and field observations.
- Free surface waves our areas of research include the generation and prediction of squall oscillations and harbour seiches, dynamics of surf beat and the wave models SWAN and SWASH.
- Sediment dynamics our areas of research include advanced experimental and numerical work concerning particle-turbulence interaction, as well as flocculation and sedimentation processes.
- Numerical model development our areas of research include development of non-hydrostatic models for the investigation of dam breaks including inundations, short wave problems, tsunamis, near field plume discharges, stratified flows, and local scour near dams, unstructured grids via finite volume methods and finite element methods.

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics	
Project leaders Dr. C.A. Katsman Research theme Complex dynamics of fluids Participants Prof. J.D. Pietrzak, Dr. J-M Sayol (post-doc), S. Georgiou (PhD), S.L. Ypma (PhD) Funded by NWO Funded % NWO 100% Start of the project 2015 Information Sotiria Georgiou S.Georgiou@tudelft.nl	Project title The impact of the local eddy activity on deep convection and sinking processes in the Labrador and Irminger Seas Project aim The aim of this PhD is to investigate the impact of the local eddy activity on deep convection and sinking of dense waters in the Labrador and Irminger Seas. Using a highly idealized regional model (Massachusetts Institute of Technology (MIT) general circulation model –MITgcm) fundamental research will be conducted in order to study the impacts of the local eddy activity on deep convection and sinking of dense waters in the Labrador and Irminger Seas. Progress The pathways and the timescales of the water masses exiting the Labrador Sea (LS) via the boundary current have been investigated by Lagrangian particle tracking. This method is applied to the output of an idealized model that is capable of representing the physical processes involved in the cycle of convection in the LS. The trajectories reveal that prior to exiting the domain the water masses follow either a fast route within the boundary current or a slower route that involves boundary-interior exchanges. This study underlines the necessity of resolving the mesoscale features required to capture the interior-boundary exchange in order to correctly represent the export of the LSW. 2020 Publications • Ypma, S.L., Georgiou, S., Dugstad, J.S., Pietrzak, J.D., Katsman, C.A. Pathways and Water Mass Transformation Along and Across the Mohn-Knipovich Ridge in the Nordic Seas (2020) Journal of Geophysical Research: Oceans, 125 (9), art. no. e2020JC016075. • Georgiou, S., Ypma, S.L., Brüggemann, N., Sayol, J.-M., Pietrzak, J.D., Katsman, C.A. Pathways of the water masses exiting the Labrador Sea: The importance of boundary–interior exchanges (2020) Ocean Modelling, 150, art. no. 101623. • Ypma, S.L., Spall, M.A., Lambert, E., Georgiou, S., Pietrzak, J.D., Katsman, C.A. The contrasting dynamics of the buoyancy-forced lofoten and greenland basins (2020) Journal of Physical Oceanography, 50 (5), pp. 1227-1244.

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics

Project leaders
Robert Jan Labeur, Wim Uijttewaal

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Merel Verbeek

Cooperation
Deltares, Svasek Hydraulics BV, Witteveen + Bos BV, Tocardo Solutions BV, Dutch Marine Energy Centre, Rijkswaterstaat, European Regional Development Fund (ERDF) 2014-2020

Funded by
NWO

Funded %
NWO 90%
Industry 10%

Start of the project
2016

Information
Merel Verbeek
06 27498117
verbeekmc@gmail.com

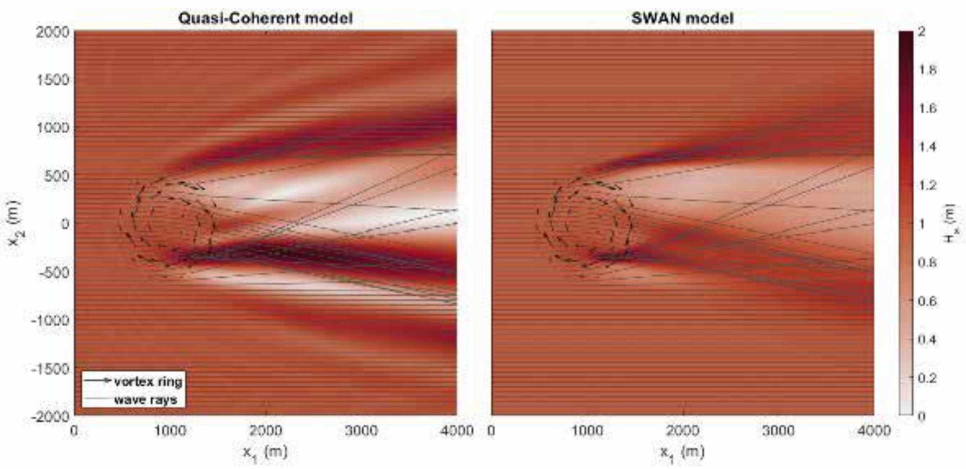
Project title
Experimental and numerical assessment of near /far -field flow interaction at storm surge barriers with hydro-turbines

Project aim
We develop a calculation tool to quantify the energy production and the hydraulic resistance of free-stream turbines in barriers. The Eastern Scheldt Storm Surge barrier in the Netherlands houses the world’s first array of tidal turbines. The hydrodynamics and performance of the location is investigated. The resulting theoretical model is validated using detailed experiments and implemented in a regional numerical model to investigate the far-field response of turbines. In this way, environmentally acceptable levels of tidal energy can be harvested.

Progress
We conducted experiments with a down-scaled tidal turbine in the flume (see the figure) and measured performance and wakes. The data prove how more power can be harvested by adjusting turbine position. Furthermore the data validate the theoretical model developed in this project and point at ways to reduce model redundancy.

2020 Publications

- Verbeek, M.C., Labeur, R.J., Uijttewaal, W.S.J. The performance of a weir-mounted tidal turbine: Field observations and theoretical modelling (2020) Renewable Energy, 153, pp. 601-614.
- Verbeek, M.C., Labeur, R.J., Uijttewaal, W.S.J. Estimating the stability of a bed protection of a weir-mounted tidal turbine (2020) International Marine Energy Journal, 3 (1), pp. 21-24.

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics	
Project leaders Prof.dr.ir. Reniers, A.J.H.M., Dr.ir. Zijlema, M. Research theme Mathematical and computational methods for fluid flow analysis Participants G. Akrish Cooperation Advisor: Dr. P.B. Smit (Sofar Ocean Technologies) Funded by NWO Funded % NWO 100% Start of the project 2016 Information Gal Akrish g.akrish@tudelft.nl	Project title Development of a stochastic wave model for coherent and nonlinear waves over variable medium Project aim This project aims to develop a generalized stochastic wave model that allows for statistically heterogeneous and non-Gaussian wave statistics when required, but otherwise reduces to a conventional action balance as used in existing spectral wave models. The proposed approach is to couple a generalized action-balance equation (which transports the full second-order statistics) with an evolution equation for the bi-spectrum. This requires not only further development of the transport equations for the cross-correlations and for the bi-spectrum, but also developing an advanced approximation for the statistical closure. Progress This part of the project generalizes the model proposed by Smit and Janssen (2015), which transports the complete second-order statistics of a wave field over variable bathymetry, to cases of wave-current interactions. Therefore, the generalized model, referred to as the "Quasi-Coherent model", is currently capable to account for the statistical contribution of wave interferences (usually neglected in operational model such as SWAN) generated in areas where currents induced focal zones (see the figure). 2020 Publications Akrish, G., Smit, P., Zijlema, M., Reniers, A. Modelling statistical wave interferences over shear currents (2020) Journal of Fluid Mechanics, 891, art. no. A2.  *The distribution of the significant wave height due to the interaction between waves and a vortex-ring.*

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics	
Project leaders C.A. Katsman, J.D Pietrzak, H.A. Dijkstra Research theme Mathematical and computational methods for fluid flow analysis Participants C.G. van der Boog Funded by Delft University of Technology Funded % University 100% Start of the project 2016 Information Carine van der Boog 06 33 878 152 C.G.vanderBoog@tudelft.nl	Project title The life cycle of eddies in the Caribbean Sea Project aim Satellite altimetry shows that the Caribbean Sea is a region rich in eddy activity. Eddies shed from the North Brazil Current (NBC) intermittently enter the region through straits between the chain of islands that separates the Atlantic from the Caribbean. However, not all NBC eddies reach the Caribbean and it is unclear why this is the case. Surprisingly, the altimetry shows that the eddies become stronger over time once they are in the Caribbean region. No satisfactory explanation exists for this phenomenon either. In this study, a regional model of the Caribbean Sea will be developed, in which the lifecycle of ocean eddies can be studied to address these questions. Progress • Development of a regional model of the Caribbean Sea and an analysis of the influence of wind stress on the development of Caribbean eddies. • Development of a high-resolution ocean model of the Caribbean Sea to study the eddy-island interaction. • Analysis of double-diffusive thermohaline staircase structures in the Caribbean Sea to study water mass transformation in this region. 2020 Publications • van Westen, R.M., Dijkstra, H.A., van der Boog, C.G., Katsman, C.A., James, R.K., Bouma, T.J., Kleptsova, O., Klees, R., Riva, R.E.M., Slobbe, D.C., Zijlema, M., Pietrzak, J.D. Ocean model resolution dependence of Caribbean sea-level projections (2020) Scientific Reports, 10 (1), art. no. 14599. • Keyzer, L.M., Herman, P.M.J., Smits, B.P., Pietrzak, J.D., James, R.K., Candy, A.S., Riva, R.E.M., Bouma, T.J., van der Boog, C.G., Katsman, C.A., Slobbe, D.C., Zijlema, M., van Westen, R.M., Dijkstra, H.A. The potential of coastal ecosystems to mitigate the impact of sea-level rise in shallow tropical bays (2020) Estuarine, Coastal and Shelf Science, 246, art. no. 107050.

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics

Project leaders
Wim Uijttewaal, Robert Jan Labeurl

Research theme
Complex dynamics of fluids

Participants
Said Alhaddad

Funded by
NWO and industry

Funded %
NWO 55%
Industry 45%

Start of the project
2016

Information
Said Alhaddad
S.M.S.Alhaddad@tudelft.nl

Project title
Understanding soil-water interaction as relevant to breaching flow slides

Project aim
Breaching is a gradual, retrogressive failure of a steep subaqueous slope, greater than the angle of repose. Breaching flow slides are accompanied by the generation of turbidity currents. This current is driven by excess density versus the ambient fluid; it may increase erosion of the sand surface, picking up more sediment into suspension, thereby increasing speed and erosion potential. The aim of this research is to understand the interaction between the turbidity current and the slope surface.

Progress
Novel large-scale experiments on breaching flow slides were conducted. We obtained direct measurements of breaching-generated turbidity currents illustrating their spatial development and visualizing the structure of their velocity and sediment concentration. The results reveal that breaching generated turbidity currents are self-accelerating; sediment entrainment and flow velocity enhance each other. Consequently, the erosion rate of the breach face increases in the downstream direction until a certain threshold, possibly
due to turbulence damping.

2020 Publications
- Alhaddad, S., Labeur, R.J., Uijttewaal, W. Large-Scale Experiments on Breaching Flow Slides and the Associated Turbidity Current (2020) Journal of Geophysical Research: Earth Surface, 125 (10), art. no. e2020JF005582.
- Alhaddad, S., De Wit, L., Labeur, R.J., Uijttewaal, W. Modeling of breaching-generated turbidity currents using large eddy simulation (2020) Journal of Marine Science and Engineering, 8 (9), art. no. 0728.
- Alhaddad, S., Labeur, R.J., Uijttewaal, W. Breaching flow slides and the associated turbidity current (2020) Journal of Marine Science and Engineering, 8 (2), art. no. 67.

The experimental setup

TUD – Civil Engineering and Geosciences – Environmental Fluid Mechanics	
Project leaders Prof.dr.ir. Z.B. Wang Research theme Mathematical and computational methods for fluid flow analysis Participants F.P. de Wit, M.F.S. Tissier, A.J.H.M. Reniers, 3 more PhD's, 7 supervisors and 4 promotors from Delft University of Technology, Utrecht University and University of Twente Funded by STW and industry Funded % NWO 67% Industry 33% Start of the project 2016 Information Floris de Wit f.p.dewit@tudelft.nl	Project title SEAWAD Project aim Improve understanding and model capabilities of intra-wave sediment transport in environments where waves and strong currents are encountered. The SEAWAD project focusses on the tidal inlet near Ameland, where tidal currents and waves combine and result in sediment transport. Firstly representation of wave nonlinearity (skewness and asymmetry) will be investigated and improved as this is an important driving force for sediment transport. Subsequently, steps will be made towards intra-wave sediment transport. Progress An additional paper on the project results has been published. 2020 Publications Van Prooijen, B.C., Tissier, M.F.S., De Wit, F.P., Pearson, S.G., Brakenhoff, L.B., Van Maarseveen, M.C.G., Van Der Vegt, M., Mol, J.-W., Kok, F., Holzhauer, H., Van Der Werf, J.J., Vermaas, T., Gawehn, M., Grasmeijer, B., Elias, E.P.L., Tonnon, P.K., Santinelli, G., Antolínez, J.A.A., De Vet, P.L.M., Reniers, A.J.H.M., Wang, Z.B., Den Heijer, C., Van Gelder-Maas, C., Wilmink, R.J.A., Schipper, C.A., De Looff, H. Measurements of hydrodynamics, sediment, morphology and benthos on Ameland ebb-tidal delta and lower shoreface (2020) Earth System Science Data, 12 (4), art. no. 2775, pp. 2775-2786.

REPORTS OF INDIVIDUAL RESEARCH GROUPS

TU/e Technische Universiteit **Eindhoven** University of Technology

Department of Applied Physics

Fluids & Flows
Transport in Permeable Media
Elementary Processes in Gas Discharges

Department of Mechanical Engineering

Energy Technology
Power & Flow
Microsystems

Department Chemical Engineering and Chemistry

Multi-scale Modelling of Multi-phase Flows
Chemical Process Intensification
Interfaces with Mass Transfer

Department Mathematics and Computer Science

Centre for Analysis, Scientific Computing and Applications (CASA)

Department of the Built Environment

Buiding Physics

TU/e – Applied Physics– Fluids and Flows

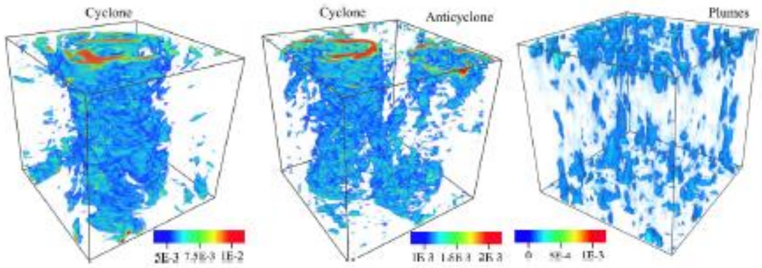
Prof.dr. HJH Clercx
Prof.dr.ir. GJF van Heijst
Prof.dr. F Toschi
Prof.dr.ir. AA Darhuber

The research in this section addresses fundamental questions in fluid dynamics through statistical fluid mechanics, large-scale computations and dedicated laboratory experiments with modern optical diagnostics for detailed flow measurements. Our motivation is the desire to unravel the fundamental properties of these phenomena and systems and use that knowledge to help solve challenges.

Our tools effectively expose the intricate fundamentals of turbulence to our curiosity, allowing us to answer many intriguing questions. How do buoyancy and rotation affect the statistical properties of turbulence? What can – and should – we do to shape turbulence to our needs? These questions are also extended to environmental applications. How is particulate matter such as sediment or droplets transported in turbulent flows?

Within the exciting field of multiphase and complex fluids, we study the role of mesoscale physics on the macroscopic dynamics of such systems. For example: how drastically do fluid-fluid interfaces, bubbles, particles or polymers affect macroscopic properties? We also study the fluid physics at even smaller scales where capillary and even molecular-length scales are relevant, and where surface and interfacial effects dominate the behaviour. Examples include Marangoni flows, interfacial instabilities and (de)wetting phenomena.

Our main research themes are turbulence, environmental fluid mechanics, multiphase and complex fluids, and nano- and microhydrodynamics.

TU/e – Applied Physics– Fluids and Flows	
Project leaders RPJ Kunnen, HJH Clercx Research theme Complex dynamics of fluids Participants AJ Aguirre Guzman, M Madonia, JS Cheng, HJH Clercx, RPJ Kunnen Funded by ERC Funded % EU 100 % Start of the project 2016 Information RPJ Kunnen r.p.j.kunnen@tue.nl	Project title TROCONVEX: Turbulent rotating convection to the extreme Project aim Many geophysical and astrophysical systems are driven by buoyancy and affected by rotation. The flow behaviour in these large-scale systems shows remarkable differences with the small-scale geometries usually studied in laboratory or simulation settings, making extrapolations from current scaling models impossible. In recent years evidence of a new scaling regime has been observed. This so-called geostrophic regime is expected to be relevant to these large-scale flows. Heat transfer models are an essential part of studying their energy balance, however strong thermal forcing and rapid rotation make difficult to replicate in numerical computations. In this project we aim to model the heat transfer by carrying out parallel numerical simulations capable to cover an unprecedented part of this new regime and to compare whenever possible with experimental results from the companion experimental investigations in this project. The outcome is crucial for the understanding of rotating convection in geo/astrophysics. Progress We study rotating Rayleigh-Bénard convection of different fluids at extreme parameter values relevant to geo-/astrophysical settings. We characterize the flow via heat transport efficiency, mean temperature distribution and flow statistics. We distinguish several flow regimes, in particular, one of large-scale vortices (LSVs; see the figure) for fluids with low- and moderate-viscosity at high rotation rates. These large-scale flows resemble those in the atmosphere, in the oceans and in the interior of planets as in the Earth's outer core. Multiple articles were submitted in 2020. 

TU/e – Applied Physics– Fluids and Flows

Project leaders
JCH Zeegers, JGM Kuerten, AA Darhuber

Research theme
Complex dynamics of fluids

Participants
R Dellaert, S Tajfirooz, J van der Veen and project leaders

Cooperation
TU Delft, UTwente, UU, Radboud University, and several companies, part of Perspectief programme STW

Funded by
STW, Umincorp, Sumitomo, Syngenta, Dimaen, FNLI, AEB,

Funded %
STW 70 %
Industry 30 %

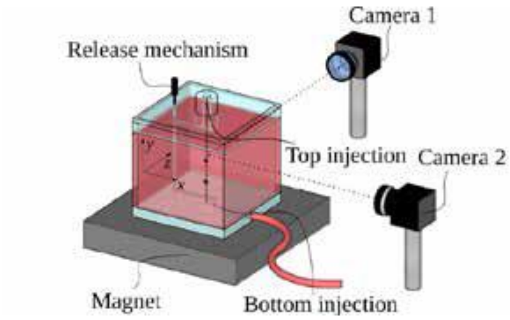
Start of the project
2016

Information
J Zeegers
j.c.h.zeegers@tue.nl

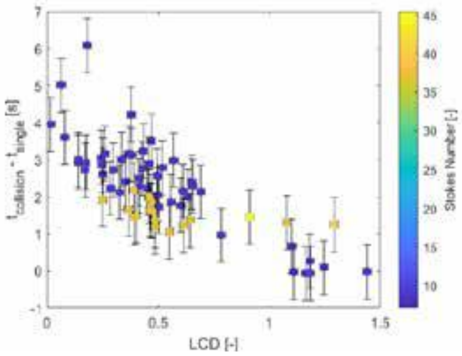
Project title
Particle fluid particle interaction project 2.2 of Perspectief MDS

Project aim
For Magnetic Density Separation (MDS) the behaviour of particles in a low turbulence channel flow is relevant. Particles are involved in a neutral buoyancy sedimentation field and it is important that particles with different densities can be separated. In this study the so-called particle-fluid-particle interaction is studied under various circumstances in order to know how a stable stratification of particles will take place in a neutral buoyancy field. The investigations are carried out experimentally as well as via numerical methods by two PhD students.

Progress
Inside the tank which has an effective mass density gradient a lot of experiments have been carried out with particle tracking velocimetry (PTV). This effective mass density gradient has been created with a MnCl2 solution together with a strong Halbach array magnet. The experiments both used single particles as multiple particles that could also collide. Where the particles could be spherical as ellipsoid in shape.



Tank used for PTV setup with magnet and MnCl2 solution.



Experimental results of PTV measurements.

TU/e – Applied Physics– Fluids and Flows

Project leaders
JCH Zeegers, JGM Kuerten, AA Darhuber

Research theme
Complex dynamics of fluids

Participants
R Dellaert, S Tajfirooz, J van der Veen and project leaders

Cooperation
TU Delft, UTwente, UU, Radboud University, and several companies, part of Perspectief programme STW

Funded by
STW, Umincorp, Sumitomo, Syngenta, Dimaen, FNLI, AEB,

Funded %
NWO 70%
Industry 30%

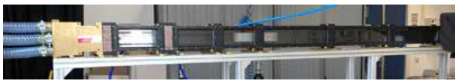
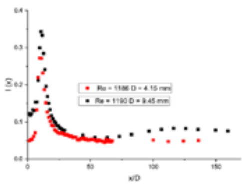
Start of the project
2016

Information
J Zeegers
j.c.h.zeegers@tue.nl

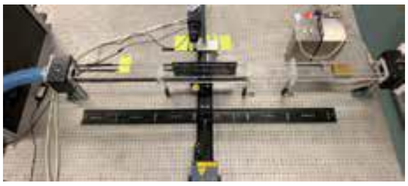
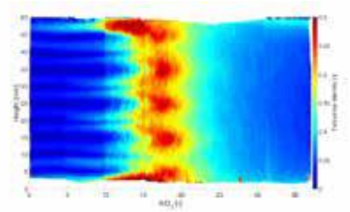
Project title
Ultra low turbulence ducts for magnetic density separation. Project 2.1 Perspectief MDS

Project aim
For Magnetic Density Separation (MDS) development of a low turbulence channel flow is important. This has to be achieved through honeycomb pipe bundles alone as filter screens cannot be used due to fouling. Experimental and numerical investigations will be carried out to study under which conditions the downstream flow field of a honeycomb system has lowest turbulence level. This is needed to achieve best separation quality downstream of the honeycomb. The study is carried out by two PhD students.

Progress
Currently a lot of measurement data have been gathered from the laser Doppler velocimetry (LDV) wind tunnel. The understanding of these results will be fit with a turbulence decay power law and put into a paper. Parallel to the LDV wind tunnel also a lot of experiments have been carried out in the particle image velocimetry (PIV) wind tunnel. These results are also being fit in a turbulence decay power law and are then compared with the LDV turbulence decay power law. The plan is to publish a paper about these results as well. Already some results have been presented at Physics@Veldhoven 2020.

Various honeycombs that can be installed in both wind tunnels.
Wind tunnel for LDV setup.
Experimental results of LDV setup.

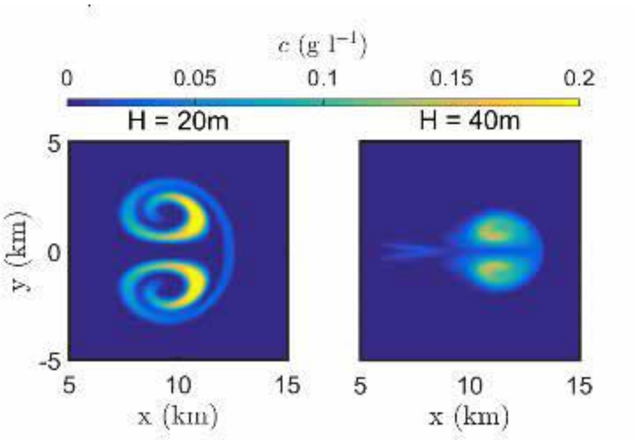
Wind tunnel for PIVsetup. *Experimental results of PIV setup.*

TU/e – Applied Physics– Fluids and Flows	
Project leaders AA Darhuber, PPAM van de Schoot, JDR Harting Research theme Complex dynamics of fluids Participants B He, TGW van der Heijden, AA Darhuber, PPAM van der Schoot, JDR Harting Cooperation ASML Funded by STW, ASML Funded % STW 70% Industry 30% Start of the project 2015 Information AA Darhuber a.a.darhuber@tue.nl	Project title Towards zero defectivity Project aim The goal of the project is to elucidate the origin of and physical mechanism behind so-called watermark defects in immersion lithography. When photoresist-covered wafers come into contact with water droplets, the photoresist structures can be adversely affected after development, pointing towards a lower solubility of the resist. The generic suspicion is that the leaching of photoresist chemicals by the water droplets is responsible for the defect formation. Progress Several manuscripts have been finalized and published containing experimental results and numerical/theoretical models. The main insight was that water droplets remove ionic material from the surface region of the photoresist, which induces deliquescence, i.e. a reduction in vapor pressure and a corresponding slow down of the droplet evaporation rate. Moreover, it was discovered that residues that were deposited after complete evaporation could not completely be removed by rinsing with water (which is standard procedure in the industrial process), despite that the residue is thought to be composed of small molecular species.

TU/e – Applied Physics– Fluids and Flows	
Project leaders M Duran Matute, HJH Clercx Research theme Complex dynamics of fluids Participants TJJM van Overveld, M Duran Matute, HJH Clercx Cooperation W-P Breugem (TUD) Funded by University Funded % University 100% Start of the project 2019 Information M Duran Matute m.duran.matute@tue.nl	Project title Towards zero defectivity Project aim The objective of this project is to characterize the non-linear interactions between sediment particles and an oscillating fluid, which result in the emergence of large-scale patterns in the form of particle chains or ripples. This project focuses on the dynamics of sediment under a symmetrically oscillating flow, for which non-linear effects are directly visible. Different flow conditions will be related to both the statistical response of individual sediment grains and the macroscopic patterns. Non-linear effects will be studied by changing the flow conditions in a complex manner. The problem will be studied using fully resolved simulations and verified using experiments. Progress Experimental and numerical results from literature (for particle chains and ripples) were collected and compared in order to define relevant regimes. Using these parameters, exploratory experiments on particle chains under a free surface with gravity waves have been done. Data analysis tools were developed to obtain particle positions, trajectories and 2D density correlation function from these experiments. For future experiments a setup without a free surface was designed. For the numerical simulations, a finite volume code with immersed boundary method has been adapted.

TU/e – Applied Physics– Fluids and Flows	
Project leaders HJH Clercx, F Toschi Research theme Complex dynamics of fluids Participants RRL Reinartz, HJH Clercx, F Toschi, RPJ Kunnen Cooperation DA Shestakov (VDL ETG) Funded by VDL ETG, Rijksdienst voor Ondernemend Nederland, TU/e Funded % University 10% Industry 60% Scholarships 30% Start of the project 2019 Information HJH Clercx h.j.h.clercx@tue.nl	Project title ACCESS – Active Contamination Control for Equipment and SubstrateS particle contamination transport Project aim Obtain fundamental understanding of the transportation of small contamination particles at finite Knudsen numbers. In this rarefied regime non-equilibrium effects take place for which continuum approach modelling is inaccurate. Direct simulation Monte Carlo (DSMC) will therefore be used to investigate the complicating effects like velocity slip and thermophoretic forcing. The numerical modelling will be verified with experiments. The obtained knowledge will be used to develop a simplified model for the behaviour of these particles in rarefied conditions which can be used in engineering focused simulation software. Progress Familiarize with the world of rarefied gases and the inhouse developed DSMC code. Implemented wall boundary condition based on the CLLscattering kernel and the adiabatic-scattering kernel. Working on the implementation of a particle inside the DSMC simulation procedure. Setting up the requirements for the validation experiment which will then be incorporated into the design of apparatus staged at VDL ETG.

TU/e – Applied Physics– Fluids and Flows	
Project leaders R Benzi, F Toschi Research theme Complex dynamics of fluids Participants G Guccione, R Benzi, F Toschi A Plummer D Nelson Cooperation University of Rome, Tor Vergata, HPC-LEAP Program Funded by HPC-LEAP (Marie Curie Fellowship) Funded % University 50% EU 50 % Start of the project 2017 Information F Toschi f.toschi@tue.nl	Project title Population dynamics in two-dimensional compressible turbulence Project aim This project focuses on the enhancement or suppression of selective advantage in population dynamics subject to advection of two-dimensional compressible turbulence. We implement a two-dimensional code particularly suited to investigating a large number of particles. Progress Many interesting studies can follow up on our work. One of those we are here moving into is to analyze the dynamic and the genetic of marine phytoplankton in the presence of a turbulent flow and we develop a new theory to understand how the eddy diffusivity is involved on the fixation time of beneficial allele.

TU/e – Applied Physics– Fluids and Flows	
Project leaders GJF van Heijst, M Duran Matute Research theme Complex dynamics of fluids Participants AS Gonzalez Vera, M Duran Matute Funded by CONACYT (Mexico) Funded % Scholarships 100% Start of the project 2015 Information M Duran Matute m.duran.matute@tue.nl	Project title The effects of barotropic vortices on sediment transport: an experimental and numerical study Project aim Understand the underlying physics and improve the modeling of how barotropic vortices and sediment interact. Progress Our research on the sediment transport capabilities of translating monopolar vortices has concluded with a publication currently under review. Additionally, we made an in-depth analysis of the photogrammetric technique used to measure the deformation of a submerged sediment bed. Currently, we are studying the transport of sediment by large-scale, tidal dipolar vortices by means of numerical simulations. The hydrodynamics are solved using a coastal three-dimensional model that solves the equations of motion with the use of a turbulence closure model. The interaction with the sediment bed is modelled using a zeroth order Partheniades-Krone resuspension formulation. Furthermore, the importance of the vertical velocities on the sediment transport capacity of the dipolar vortex is determined by comparing simulations with and without the vertical advection of sediment. 

TU/e – Applied Physics– Fluids and Flows

Project leaders
HJH Clercx, F Toschi

Research theme
Complex dynamics of fluids

Participants
B Goshayeshi, G Di Staso, HJH Clercx, F Toschi

Funded by
TU/e

Funded %
University 100 %

Start of the project
2017

Information
HJH Clercx
h.j.h.clercx@tue.nl
F Toschi
f.toschi@tue.nl

Project title
Study of non-isothermal rarefied gas flows with hybrid DSMC -LBM simulations

Project aim
This project will contribute to the development of a hybrid computational tool combining Direct Simulation Monte Carlo (DSMC) and Lattice Boltzmann Method (LBM) for the simulation of rarefied gasses. This hybrid DSMC-LBM algorithm allows to switch efficiently between DSMC and LBM depending on the local value of the Knudsen number quantifying the rarefaction of the flow. Within this project the hybrid DSMC-LBM algorithm will be extended to include thermal effects. The goal in the end is to measure, model and understand heat fluxes and fluid-wall interactions on surfaces over which a rarefied gas flow is forced.

Progress
Physics of thermal transfer problem in a 2D rarefied Rayleigh-Bénard system at the slip flow regime for $0.015<Kn<0.03$ is numerically investigated, using high-performance DSMC simulations. Heat transfer was evaluated (in the form of a Nusselt number) under different rarefactions and gravitational accelerations. At each rarefaction degree, three behaviours were observed as the gravitational acceleration increases: (i) onset of convection, (ii) maximum of convection, (iii) extinction of convection (presumably due to the stratification). It was observed that the onset of convection can be approximated at a constant modified Rayleigh number. At the maximum convection state, rarefaction affects the Nusselt number to have a linear behaviour with respect to the rarefaction degree (Kn). Finally, the extinction of convection at high Rayleigh numbers found to be under the effect of two factors of stratification and rarefaction.

<table>
<tr><th colspan="2">TU/e – Applied Physics– Fluids and Flows</th></tr>
<tr><td>Project leaders
F Toschi, PPAM van der Schoot

Research theme
Complex dynamics of fluids

Participants
A Goyal, F Toschi, PPAM van der Schoot, RAJ Janssen

Funded by
Shell-NWO/FOM

Funded %
NWO 100%

Start of the project
2016

Information
Prof. dr. F Toschi
f.toschi@tue.nl
PPAM van der Schoot
p.p.a.m.v.d.schoot@tue.nl</td><td>Project title
High-efficiency organic solar cells by controlling microstructure through processing

Project aim
The aim is to study the basic fluid dynamics physics of multi-component phase separation and solidification of suspensions under steady evaporation. This will contribute to a better understanding on the dynamics of the morphology formed during the processing of organic solar cells and may contribute in increasing the power conversion efficiency.

Progress
Demixing dynamics near the substrate: Comparison of the Lattice Boltzmann results and the diffusion-dominated phase field theory to understand the impact of hydrodynamics during the demixing process of a binary blend near a preferentially wetting substrate. Percolation characteristics of demixing morphologies: Study the evolution of the extend of percolating domains in demixing morphologies especially in contact with a preferentially wetting substrate, particularly important in the context of organic solar cells for effective charge transport.</td></tr>
</table>

TU/e – Applied Physics– Fluids and Flows	
Project leaders AA Darhuber, LPJ Kamp Research theme Complex dynamics of fluids Participants S Wang, AA Darhuber, LPJ Kamp Cooperation Canon Funded by NWO, Canon, TU/e, UT Funded % University 100% Start of the project 2019 Information AA Darhuber a.a.darhuber@tue.nl	Project title Capacitive tomography of water saturation distributions in thin porous media Project aim The goal of the project is to develop an experimental technique for measuring the distribution of water in paper with high spatial and temporal resolution based on capacitance tomography and to validate it using experiments and numerical simulations. Progress Shuo Wang has followed an introduction into micro-/nanofluidics and electrokinetics and has commenced experiments at Canon using an existing setup for 1D monitoring of the imbibition dynamics of water into paper based on impedance spectroscopy. The design of a setup providing 3D resolution is underway.

TU/e – Applied Physics– Fluids and Flows

Project leaders
F Toschi, HJH Clercx

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
C Livi, G Di Staso, F Toschi, HJH Clercx

Cooperation
HC van Brummelen, ASML

Funded by
STW

Funded %
NWO 100%

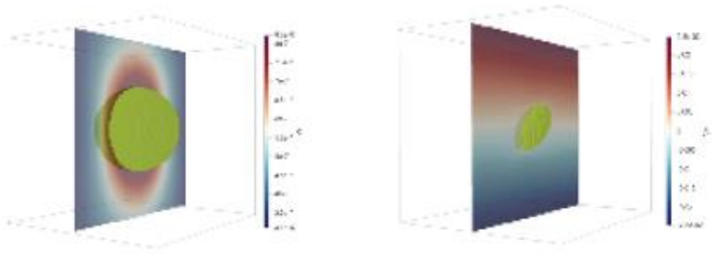
Start of the project
2018

Information
F Toschi
f.toschi@tue.nl

Project title
RARET RANS : Transport in rarefied gases in next generation photolitography machines

Project aim
The RareTrans project focuses on the development of computational techniques to predict heat and mass transfer in rarefied gas flows, as occurring in Extreme-Ultra-Violet (EUV) machines. Gas flows in EUV machines are extremely complicated, on account of the wide range of Knudsen numbers (viz, the ratio of the mean free path between gas molecules and the device scale) that occur in EUV machines. Conventional Navier-Stokes continuum models are invalid in the rarefied regime, so the RareTrans project addresses the development of computational techniques to overcome this barrier by solving the more fundamental Boltzmann equation.

Progress
We performed a systematic error analysis of finite-size particle dynamics using the Hermite regularized Lattice Boltzmann method, for different particles resolution, addressing translational and rotational systems separately. The motivation is the understanding of the degree of accuracy provided by different boundary condition models at the fluid-solid interface, with a focus on cases where particles are discretized using few lattice grid points. An improved version of the interpolation scheme for curved boundaries is developed in order to compute particle boundary position also when the particle is crossing a link between two fluid nodes, showing that the capability to resolve non-spherical particles is further enhanced.



Snapshots of LBM 3D simulations. On the left the settling of a spherical particle in the Stokes regime is used to investigate translational dynamics accuracy. On the right the rotation of an ellipsoid in a shear flow is used to address rotational dynamics accuracy.

TU/e – Applied Physics– Fluids and Flows

Project leaders
AA Darhuber, J Harting

Research theme
Complex dynamics of fluids

Participants
G Venditti, AA Darhuber, J Harting

Cooperation
S Luding (UT), Océ

Funded by
NWO/STW

Funded %
NWO 75 %
Industry 25%

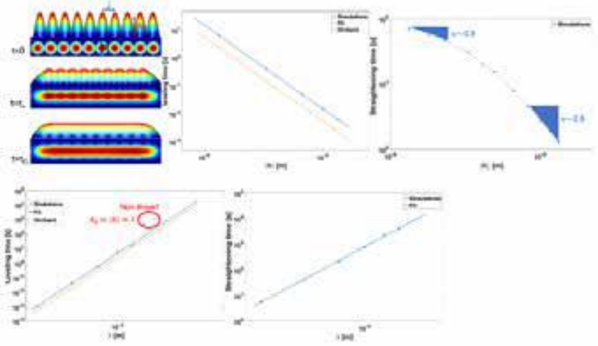
Start of the project
2016

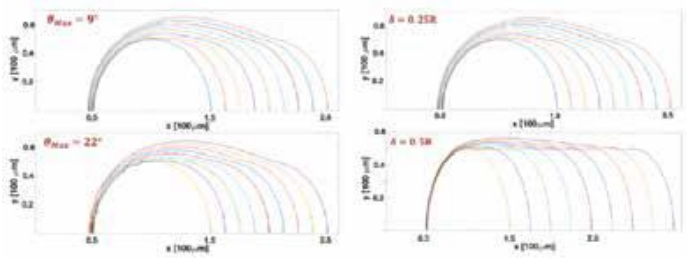
Information
AA Darhuber
a.a.darhuber@tue.nl

Project title
Spreading and imbibition of water-based ink in porous media

Project aim
The aim of the project is to understand and control the absorption and imbibition of solutions and suspensions (e.g. inkjet ink) in porous media, such as paper.

Progress
We studied the process of leveling1 and straightening2 for an infinite array of small overlapping droplets. We compare the leveling time with the Orchard time (leveling time for lines). We developed a model that include contact angle heterogeneities without phenomenological relationship, but based on thin film approach. We tested and proved that this model has more than one equilibrium contact angle and the advancing contact angle is different from the receding one. We modelized the deposition process as a smooth process, it can be however generalized to instantaneous/sharp deposition paying in computational work. We finally use all the features developed to study the formation of primary head on both the horizontal (contact line) plane and vertical (thickness) plane.





TU/e – Applied Physics– Fluids and Flows

Project leaders
HJH Clercx, RPJ Kunnen

Research theme
Complex dynamics of fluids

Participants
A Kozhevnikov, RPJ Kunnen, HJH Clercx

Cooperation
TNO

Funded by
TKI

Funded %
Industry 100 %

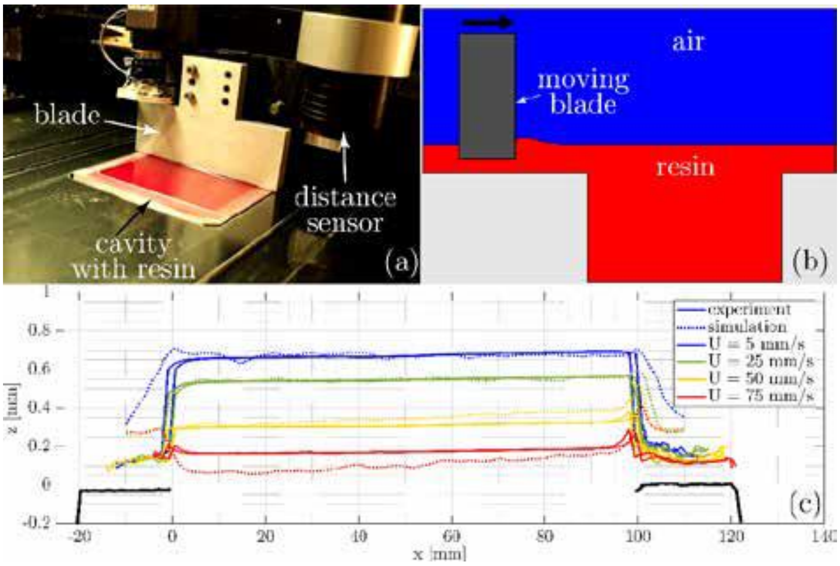
Start of the project
2017

Information
HJH Clercx
h.j.h.clercx@tue.nl

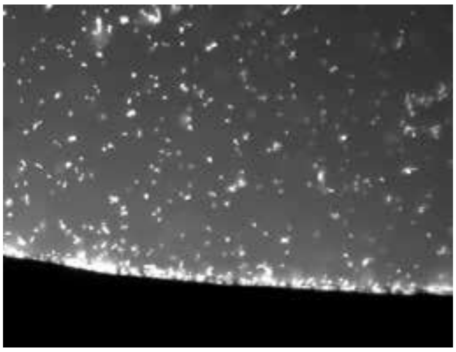
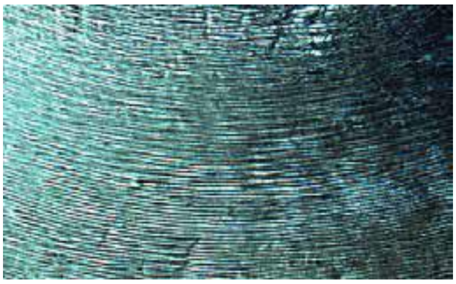
Project title
Sensing and controlling resin-layer thickness in additive manufacturing processes

Project aim
The project focuses on the improvement of technologies in industrial additive manufacturing and particularly in ceramic vat photopolymerization that aims in decreasing of the building time and increasing quality and accuracy of the final products. The project will contribute to the development of a tool for the resin layer thickness measurements after the recoating process. The study will also include an investigation of recoating parameters and their influence on the free-surface deformations with different geometries.

Progress
Experimental and numerical study of the recoating process in additive manufacturing have been continued. A liquid layer deposition over a rectangular-shaped topography was explored. It has been found that the resin surface level over the cavity can be successfully controlled by changing the recoater speed. Simple analytical expression has been obtained to qualitatively predict the liquid surface topography. Numerical and analytical results showed a good agreement with the experiment.



The experimental setup (a) and sketch of the 2D CFD model (b) (not to scale). The comparison of resin surface topography (c) after recoating at different speeds (solid lines are for the experiments, dashed lines are for the CFD simulations).

TU/e – Applied Physics– Fluids and Flows	
<u>Project leaders</u> H Gelderblom <u>Research theme</u> Complex dynamics of fluids <u>Participants</u> T Wilting, H Gelderblom <u>Cooperation</u> AA Darhuber (TU/e), J Foolen (TU/e), A Marin (UT), J Snoeijer (UT) <u>Funded by</u> Veni (NWO), TU/e <u>Funded %</u> University 50% NWO 50% <u>Start of the project</u> 2018 <u>Information</u> H Gelderblom h.gelderblom@tue.nl	<u>Project title</u> Evaporation of “living liquid” drops <u>Project aim</u> 1) Study on the formation of the coffee-ring effect for evaporation drops of living liquids. Identify the basic mechanisms that control cell-pattern formation and the differences in (collective) dynamics of E-coli bacteria inside the drop and near the contact line. (2) Study on the deposition patterns of collagen in evaporating collagen solution drops and the response of mammalian cells to these patterns. Construct a morphology map for different collagen pattern structures in function of collagen concentration of the solution and the evaporation rate. (3) Study on the peeling and re-peeling mechanism of an adhesive tape under tension. <u>Progress</u> (1) Construct and test setup; humidity control, side view recording and fluorescence microscopy of the bacteria. (2) Using the setup of (1) we record the contact line motion and contact angle. With polarized light microscopy (PLM) we measure the profile of the collagen stain, shown in the figure. Cells are found to strongly respond to the patterns formed. Collaboration with BioMedical Engineering (TU/e) (3) Formulating theoretical model to describe the (re-) peeling mechanisms of an adhesive tape loop. Model is in good comparison to experimental data. Collaboration with University of Twente (UT).   *Bacteria at the contact line of a drying drop* *PLM image of a collagen stain*

TU/e – Applied Physics– Fluids and Flows

Project leaders
F Toschi, HJH Clercx

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
H Li, F Toschi, HJH Clercx

Funded by
China Scholarship Council

Funded %
Scholarships 100 %

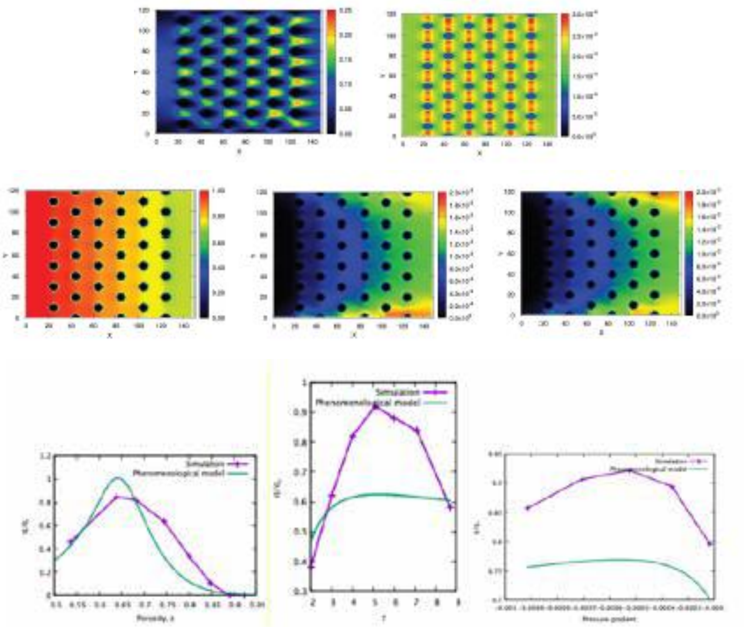
Start of the project
2017

Information
F Toschi
f.toschi@tue.nl

Project title
Plasma reactor simulation for CO 2-neutral methanol synthesis

Project aim
The conversion of CO2 into methanol using energy that is not produced from fossil fuels has been suggested to be one of the best ways of storing energy as well as for CO2 recycling. Plasma assisted catalytic conversion may help achieving this goal. To gain insights and optimize the conversion procedure, numerical models based on the Lattice Boltzmann methods and zero dimensional simulations will be employed. The goal is to achieve an efficient conversion way.

Progress
A 2D Lattice Boltzmann (LB) model is developed for plasma/flow problems. Simulations of simplified plasma fluid with electric breakdown reaction in a homogenous packed spheres bed are studied. And a phenomenological model is developed based on the simulations to help optimize the control perimeters of the test.



Simplified plasma fluid with electric breakdown reaction in a homogenous packed spheres bed. A fluid with only neutral species A flows through a regularly packed spheres bed. A pair of electrodes is placed at the top and at the bottom of the bed. The reaction is A □ B+ + C- with rate k, and k is a function of the electric field. A constant pressure drop is applied in the X direction and bounce back conditions are applied in the Y direction.

TU/e – Applied Physics– Fluids and Flows

Project leaders
RPJ Kunnen, HJH Clercx

Research theme
Complex dynamics of fluids

Participants
M Madonia, AJ Aguirre Guzman, JS Cheng, HJH Clercx, RPJ Kunnen

Funded by
ERC

Funded %
EU 100 %
Scholarships -

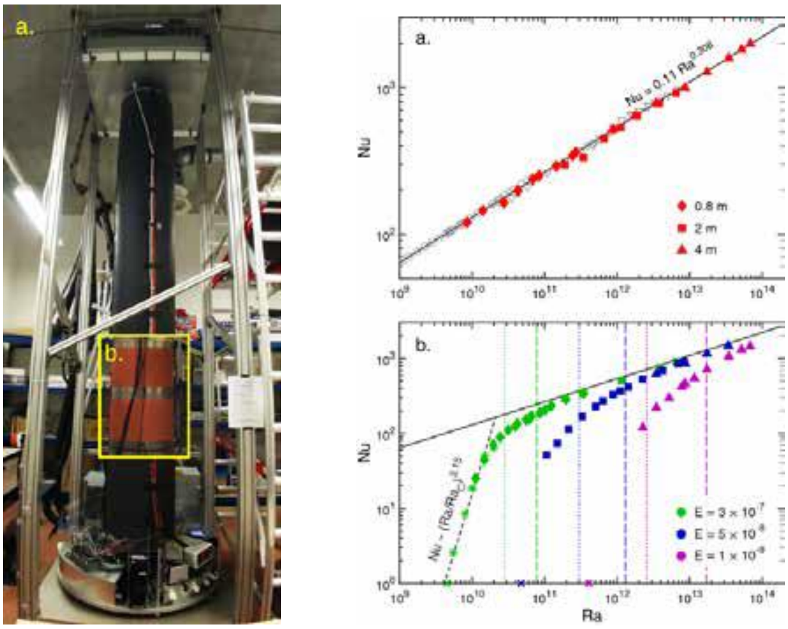
Start of the project
2016

Information
RPJ Kunnen
r.p.j.kunnen@tue.nl

Project title
TROCONVEX : Turbulent rotating convection to the extreme

Project aim
Many geophysical and astrophysical systems are driven by buoyancy and affected by rotation. The flow behaviour in these large-scale systems shows remarkable differences with the small-scale regimes usually studied, making extrapolations from current scaling models impossible. In recent years evidence of a new scaling regime has been observed. This so-called geostrophic regime is expected to be relevant to these large-scale flows. TROconvex is an experimental setup that is able to reach new extreme parameters through a 4 m high rotating tank, allowing us to have an unprecedented insight into these flows.

Progress
Completed collection of heat transfer data, also from 4 m tank, covering three different Ekman numbers. Calculation of heat transfer efficiency at different regimes, with temperature profile analysis. Visualization of the different flow regimes in the experimental setup. Analysis and characterization of the different flow regimes in Rotating Convection. Major operations of maintenance and repair of the setup, in order to solve major technical issues. Calculation of heat loss of the setup. Design and realization of a new transparent segment to perform Stereo PIV, including a calibration grid system and illumination.



TROCONVEX 4 m tank

TU/e – Applied Physics– Fluids and Flows	
Project leaders AA Darhuber, JCH Zeegers Research theme Complex dynamics of fluids Participants V Murali. AA Darhuber, JCH Zeegers Cooperation N Tomozeiu (Océ), H Wijshoff (Océ), J Harting, S Luding (UTwente) Funded by STW + Océ Funded % NWO 70% Industry 30% Start of the project 2017 Information AA Darhuber a.a.darhuber@tue.nl	Project title Spreading and imbibition of water-based printing inks in porous media - experiments Project aim Aqueous inkjet printing performs superbly on expensive paper coated with microporous layers, but the print quality on uncoated, recycled copier paper is generally less optimal. Fundamental understanding of the underlying processes is mandatory to improve water-based printing. In collaboration with Océ, we will investigate the complex multiscale and multiphase ink-substrate interactions. This will allow answering challenging questions such as: What is the role of surfactants in the imbibition dynamics? How does the nanostructure of the medium affect absorption/swelling, and how can one account for it at larger scales? Progress Systematic experiments concerning the water imbibition dynamics in paper substrates were conducted using infrared thermography. A corresponding model was developed based on unsaturated flow coupled with heat and mass phenomena in the adjacent gas phase. The numerical simulations reproduce the experimental findings well.

TU/e – Applied Physics– Fluids and Flows

Project leaders
J Harting, S Luding, WK den Otter

Research theme
Complex dynamics of fluids

Participants
Q Xie

Cooperation
The project is part of the programme "Fundamental Fluid Dynamics Challenges in Inkjet Printing (FIP)"

Funded by
Océ, UT, TUE, and FOM/NWO

Funded %
University 25%
NWO 25%
Industry 50%

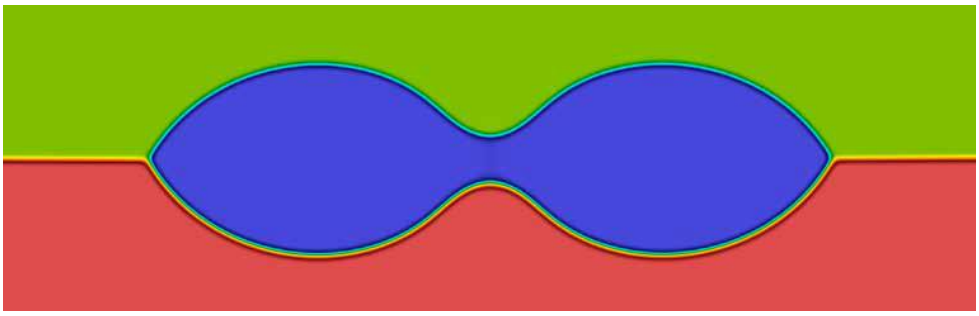
Start of the project
2018

Information
Qingguang Xie
q.xie1@tue.nl

Project title
Liquid penetration into paper: Numerics & Lattice Boltzmann

Project aim
The main scientific goal of the "Fundamental Fluid Dynamics Challenges in Inkjet Printing (FIP)" programme is to obtain insight into unresolved issues in the current inkjet process and to improve and extend the functionality of inkjet printing to meet future requirements. The current Project aims at a mesoscale model describing the deposition and imbibition of droplets of ink in paper.

Progress
1. We investigated numerically the coalescence of liquid droplets on a thick film with a multi-component lattice Boltzmann method. The growth of the bridge height follows h ~ t2/3. The bridge profiles and horizontal velocity profiles show self-similar dynamics. The simulation results are in agreement with experimental results and the theoretical prediction. Moreover, the effect of the droplet contact angle on the dynamics of coalescence were studied.
2. We investigated numerically the capillary interactions of liquid droplets on a thin film. The capillary force increases with decreasing the thickness of the liquid film.



TU/e – Applied Physics– Fluids and Flows

Project leaders
AA Darhuber, J Snoeijer

Research theme
Complex dynamics of fluids

Participants
M Chudak, J Snoeijer, AA Darhuber

Cooperation
WU, CNRS, BASF, IPF, INM, Cambridge University

Funded by
EU H2020

Funded %
EU 100%

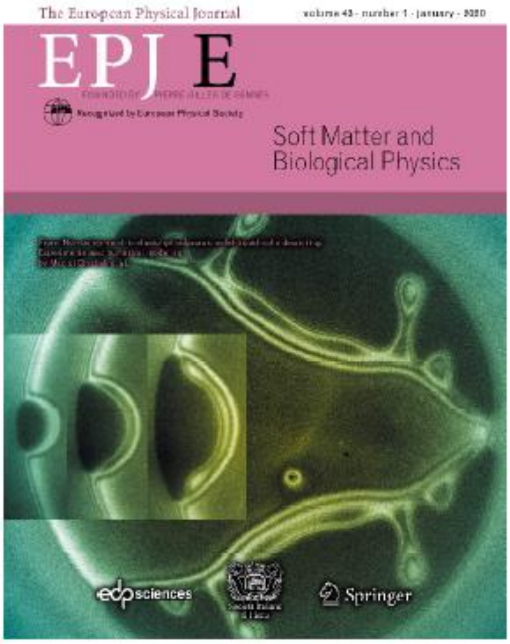
Start of the project
2015

Information
A. A. Darhuber
a.a.darhuber@tue.nl

Project title
“BioSmart” - Fluid mechanics of adhesion on wet surfaces

Project aim
The aim of this project is to elucidate the role of water in inhibiting proper adhesive contact between an adhesive label and a target surface. The goal is both to achieve fundamental understanding as well as to evaluate engineering solutions to remove the water. We will study aspects such as the wetting and dewetting dynamics, the transport of water along and through patterned and/or porous layers.

Progress
Two manuscripts on solid-liquid-solid dewetting and the dynamics of expulsion of droplets from the contact zone of two solids that are pressed together have been submitted and accepted for publication. The writing of a third manuscript on electrically enhanced adhesion has commenced.



TU/e – Applied Physics– Fluids and Flows

<u>Project leaders</u>
F Toschi, HMA Wijshoff

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
A Ghosh, F Toschi, HMA Wijshoff

<u>Funded by</u>
STW

<u>Funded %</u>
NWO 100%

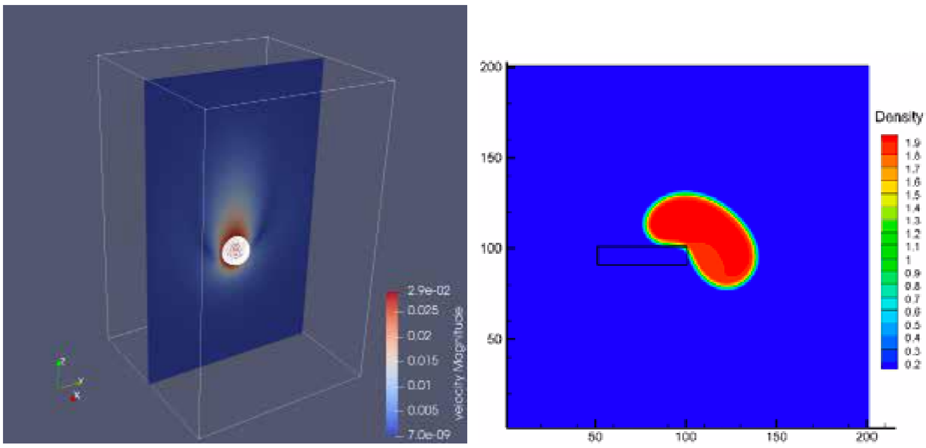
<u>Start of the project</u>
2019

<u>Information</u>
F Toschi
f.toschi@tue.nl

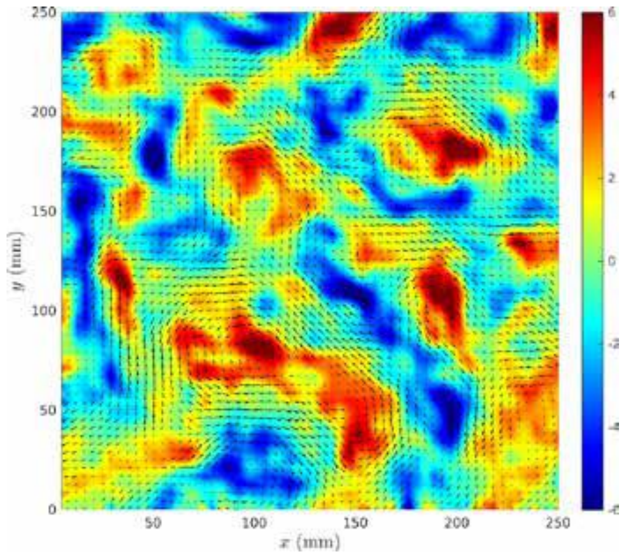
<u>Project title</u>
Physics of liquid jetting

<u>Project aim</u>
The Project aims to investigate the dynamics and development of (directional) instabilities during droplet formation in the process of inkjet printing. The project also focuses on how the presence of particulates near the nozzle boundary influences the physics of droplet formation. Lattice Boltzmann Method with high-density ratio schemes for multiphase flows will be used for generating the droplets and Immersed Boundary Methods will be applied for simulating solid-fluid interactions to study the presence of the particulates in the ink, in the hopes of understanding these processes and increase the reliability and performance of inkjet printers.

<u>Progress</u>
Started focusing on learning the Lattice Boltzmann method (LBM) and Shan-Chen multiphase method for simulating the droplet formation and multiphase flows, Implemented a basic Direct-forcing Immersed Boundary Method (IBM) in 3D in the inhouse LBM code for simulating fluid-solid interactions, validated the DF-IBM implementation with experimental results for a sedimentation of sphere under gravity, made a proof of concept simulation of a liquid droplet interacting with a solid surface (in 2D) with the Shan-Chen and DFIBM implementation.



(a) Sedimentation of a sphere under gravity, (b) Interaction of a falling droplet with an Immersed Boundary

TU/e – Applied Physics– Fluids and Flows	
Project leaders HJH Clercx, M Duran Matute Research theme Complex dynamics of fluids Participants LM Flores Ramirez, M Duran Matute, HJH Clercx Cooperation GH Keetels (TUD) Funded by CONACYT (Mexico) Funded % Scholarships 100% Start of the project 2019 Information HJH Clercx h.j.h.clercx@tue.nl	Project title Dispersion statistics in confined quasi-two-dimensional turbulence Project aim We propose to investigate the Lagrangian statistics of a forced, quasi two- dimensional turbulent flow bounded by rigid walls through numerical simulations, laboratory experiments and analytical considerations. The purpose is to determine how particle dispersion differs from its typical evolution in the unbounded 2D case as a consequence of the flow confinement by rigid lateral walls and the 3D effects caused by a finite fluid-layer thickness. Progress A spectral code for numerical simulations of two-dimensional flows has been tested and an additional module for Lagrangian particle tracking is being implemented. On the other hand, laboratory experiments of a shallow single layer of fluid driven by electromagnetic forcing are being conducted under different conditions (forcing levels, layer depths). Surface velocity fields and particle tracks have been obtained. Additionally, the velocity fields have been employed to reconstruct virtual particle trajectories. Lagrangian statistics will be calculated based on the information of the particle trajectories.  *Vorticity and velocity fields of a typical experiment. Vectors represent the horizontal velocity components and colours denote the vertical component of the vorticity.*

TU/e – Applied Physics– Fluids and Flows	
Project leaders A Corbetta Research theme Mathematical and computational methods for fluid flow analysis Participants A Corbetta, F Toschi Cooperation CNR-IAC, Rome, IT, Eindhoven Municipality, Amsterdam Municipality, ProRail BV, Glow Eindhoven, Naturalis Center, Leiden, Signify BV Funded by NWO Funded % NWO 90% Industry 10% Start of the project 2019 Information A Corbetta a.corbetta@tue.nl	Project title Understanding and controlling the flow of human crowds Project aim The project develops in the broader context of pedestrian dynamics, whose scope is to understand the stochastic and complex dynamics of human crowds in relation with fluid and physics models and with applications to serviceability of civil infrastructures. Pedestrian dynamics is a multidisciplinary field involving statistical physics, mathematical modeling and computer vision, but also civil engineering and social sciences. In this project we consider the pedestrian dynamics of visitors in public infrastructures, museums, outdoor alleys and exhibits. Targeting a major step forward in their fundamental understanding, we develop tools for quantitative monitoring, analysis and crowd control. Progress In this VENI project we addressed the development of techniques for fine-scale measurement and trajectory-based modeling of pedestrian dynamics. By leveraging on physics-informed machine learning we obtained the first-highly accurate body orientation measurement tool working in real-life and real-time. This tool, thanks to which we modeled quantitatively the connections between individual velocity and orientation, unlocks analyses of nematic ordering in dense crowds. We developed, in parallel, path-integral based tools for modeling the statistics of pedestrian trajectories in complex environments where multiple usage patterns co-exist.

TU/e – Applied Physics –Transport in Permeable Media

Prof.dr.ir. OCG Adan

In the group Transport in Permeable Media, TPM, of the department of Applied Physics at the Eindhoven University of Technology research is performed on transport and phase changes in permeable media. Our mission is to advance materials technology through an in-depth understanding of transport physics in permeable media, in support of various technology domains, such as high-tech materials, petrophysics and thermal energy storage. The interaction between transport of fluids and solutes, phase changes and material response on different scale levels -typically in the micrometre to millimetre range- forms the core of our research activities. Inherently, interdisciplinarity is in TPMs genes, encompassing transport physics, materials science, chemistry and biology. The experiment is at the heart of the group, which is due to the unique opportunities of our MRI Infrastructure, consisting of nine home-built or -modified scanners operating at fields ranging from 0.7-4.7 T. TPMs research profile is based on use-inspired basic research. Consequently, interaction with industrial players forms a cornerstone in our approach. For this reason, fruitful partnerships exist with TNO, AkzoNobel and Oce. The work is mainly funded by the Dutch Technology Foundation (STW), Materials Innovation Institute (M2I).

TU/e – Applied Physics – Transport in Permeable Media

Project leaders
H.P Huinink

Research theme
Complex dynamics of fluids

Participants
Dr.ir. H.P. Huinink, Prof.dr. O.C.G. Adan, Dr. ir. S.J.F. Erich

Cooperation
Canon, DSM, TNO, RIVM

Funded by
NWO

Funded %
NWO 100%

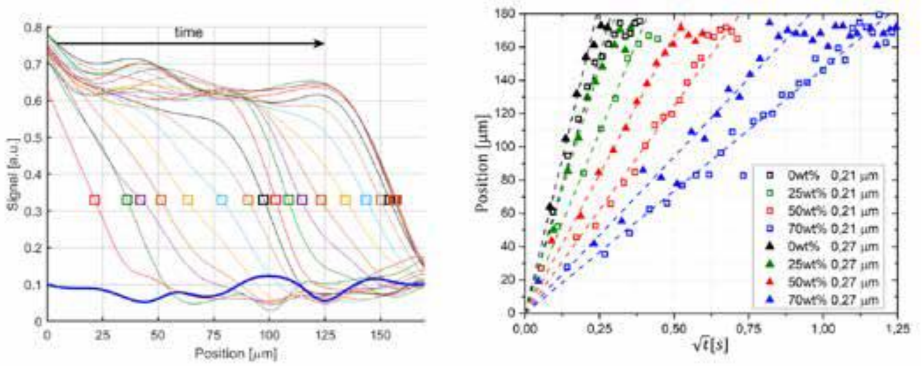
Start of the project
2010

Information
H.P. Huinink
h.p.huinink@tue.nl

Project title
Print quality and particles - Optimizing the interaction of waterbased inks and porous substrates

Project aim
Optimizing the interaction between water-based inks and porous substrates requires a better understanding of capillary driven imbibition, the process responsible for ink uptake. The aim of this project is to develop an NMR-based measurement technique, able to follow the capillary imbibition of water-based inks inside porous media. The techniques should be able to follow both particles and liquid on typical timescale found in printing media (50 – 300 ms). The experimental techniques will be used to study the capillary imbibition of microliter drops with different chemical compositions into a wide variety of porous media including printing paper.

Progress
In this study, a new high-speed NMR-based setup was developed that was able to measure liquid profiles inside non-transparent porous media with record breaking time resolution of 10ms. Using this experimental technique, studies on water-glycerol mixtures penetrating Nylon 6,6 porous media revealed a sharp liquid-air interface within the porous media. These interfaces could be used to follow front position over time revealing a Darcy like behaviour.



Left: NMR profiles of a 50wt% glycerol mixtures inside a Nylon porous membrane. Right: Front positions i.f.o. square root of time for different water-glycerol mixtures and pore radii of Nylon membranes

TU/e – Applied Physics – Elementary Processes in Gas Discharges

Prof.dr.ir. GWM Kroesen
Prof.dr.ir. U Ebert

Our mission is to study the elementary processes in gas discharges by a mix of theory, modelling, and diagnostics.

A thorough understanding of the intensity and effectivity of elementary processes like ionization, recombination, transport, radiation, excitation, de-excitation, chemical reactions and surface processes enables the group to develop predictive models of a large range of plasmas. The understanding is obtained by a strong interleaved integration of state-of-the art plasma diagnostics with advanced plasma models. Those models then in turn enable users of the plasmas to optimize the plasma source for their specific application. Over the years, the range of applications the group has worked on has shifted continuously: from plasma etching via lighting to medical applications and many other areas. However, the scientific scope of the group has not shifted: continuously the focus has been with the elementary processes in and the physics of plasmas.

Some applications fade and other applications lure, but plasma physics remains our core.

If you are not familiar with plasma and gas discharge physics, you may wish to consult the Wikipedia page about plasma physics. Alternatively, the applications and techniques sections of this site provide an explanation of the plasma sources and measurement and modelling techniques that are used in our group. If nothing else, the pictures shown there may convince you of the visual beauty of the topic of our research!

TU/e – Applied Physics – Elementary Processes in Gas Discharges

Project leaders
J. v. Dijk

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
C.E.M. Schoutrop

Cooperation
Plasma Matters B.V., eScience centre.

Funded by
NWO

Funded %
NWO 100%

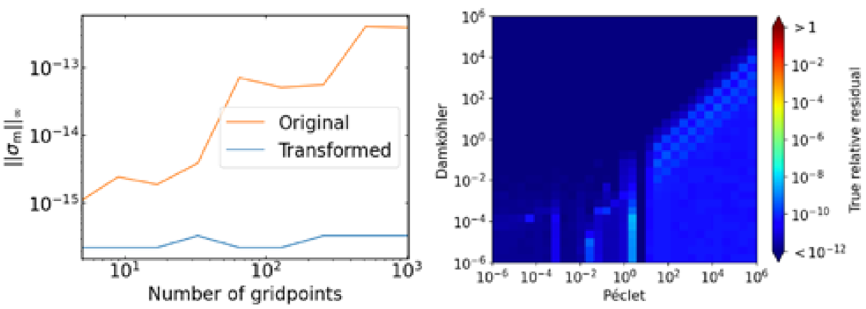
Start of the project
2018

Information
c.e.m. schoutrop
06 83705775
c.e.m.schoutrop@tue.nl

Project title
Passing XSAMS

Project aim
Expand Giovangigl's current discretization strategy by taking into account the structure of chemical source terms in the numerical method. Applying chemical reduction (model reduction) methods to plasma simulations to reduce computational cost, access more information on plasma chemistry and improve numerical stability. Provide an is an open-source implementation of an iterative solver that extends the well-known BiCGStab iterative solver for the Eigen library. Creating web-based methods for managing, distributing, and validating chemical/atomic data sets. Similar to XSAMS, however with a showcase application; MagnumPI.

Progress
A method to exploit chemical invariants present in stoichiometry has been combined with the complete flux discretization scheme. The set of governing equations can be transformed leading to fewer unknowns, better conditioned linear systems and near-exact conservation of mass and charge; "Multicomponent transport in plasmas; exploiting stoichiometry". The IDR(S) solver has been merged into the linear algebra library Eigen, development on BiCGStab(L) and IDR(S)Stab(L) has completed. Extensive comparison between the MATLAB implementations of BiCGStab, BiCGStab(L), and common modifications of these solvers is nearing completion.



Left: Comparison of the degree to which mass is conserved for a multicomponent Advection-Diffusion-Reaction problem. Right: Ongoing investigation into the reliability of Krylov subspace methods, here the residual generated by IDR(S) is shown for a wide range of Péclet and Damköhler numbers.

TU/e – Applied Physics – Elementary Processes in Gas Discharges

Project leaders
Sander Nijdam (EPG), Jeroen van Oijen (P&F), Nico Dam (P&F), Jan van Dijk (EPG)

Research theme
Complex dynamics of fluids

Participants
Thijs Hazenberg, Ravi Patel

Cooperation
NWO, Bosch Thermotechniek, Bekaert Combustion Technology, Micro Turbine Technology, Plasma Pendix

Funded by
NWO, Bosch Thermotechniek, Bekaert Combustion Technology, Micro Turbine Technology

Funded %
NWO 85%
Industry 15%

Start of the project
2019

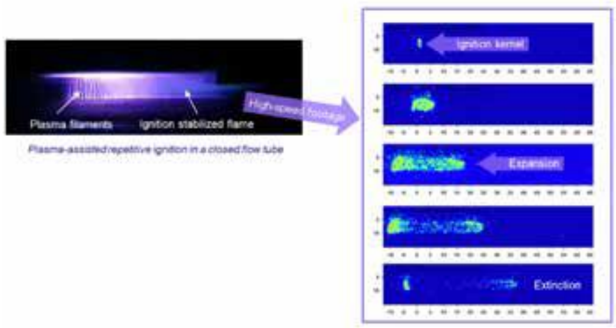
Information
S. Nijdam
s.nijdam@tue.nl

Project title
Making plasma-assisted combustion efficient

Project aim
Plasma-assisted combustion is a promising method to enhance flame stability in low-temperature flames. The reduction of flame temperature is important for reducing NOx emissions. However, control over the combustion process via the creation of radicals using plasma is not well understood. The goal is to gain an understanding of the plasma activation of combustion via both numerical and experimental works. The hope is that optimal plasma type and plasma parameters can be found which achieve maximum stabilization of the combustion processes in low temperature regime.

Progress
We observed that repetitive ignition stabilized combustion can be achieved in lean methane-air flows with a velocity much higher than the flame speed. Repetitive ignition is achieved through nanosecond-duration repetitive pulse driven DBD plasma. High-speed intensified imaging is used for visualizing repetitive ignition dynamics, and optical emission spectroscopy is employed to characterize ignition and plasma properties. Initial results suggest that near atmospheric pressure DBD plasma has a significant non-thermal contribution to the observed ignition. Furthermore, an extensive study of the modelling of thermodynamic properties of excited species has been performed. We found that great care for (excited state) species thermodynamics is required to obtain the correct thermodynamic equilibrium in plasma-assisted combustion reaction mechanisms. When insufficient care is taken, flame temperatures can be wrong by as much as 100 K. Finally, we formulated a consistent methodology to obtain correct thermodynamic data sets.

2020 Publications
R. Paulauskas, D. Martuzevičius, R.B. Patel, J.E.H. Pelders, S. Nijdam, N. J. Dam, M. Tichonovas, N. Striūgas, K. Zakarauskas, "Biogas combustion with various oxidizers in a nanosecond DBD microplasma burner", Experimental Thermal and Fluid Science, Vol. 118, pp. 110-166, 2020. DOI:10.1016/j.expthermflusci.2020.110166.



Left: Digital camera image of plasma-assisted repetitive ignition stabilized flame. Right: Highspeed intensified camera images capturing ignition dynamics in a closed tube.

TU/e - Mechanical Engineering - Energy Technology

Prof.dr. HA Zondag
Prof.dr.ir. DMJ Smeulders
Prof.dr.ir. EH van Brummelen

Research in the field of Energy Technology at a Mechanical Engineering Department requires the combination of fundamental research and the study of engineering systems and devices. The fundamental research is in the field of Heat Transfer, and the engineering system focuses on small-scale Energy Systems with a strong emphasis on sustainability.

The approach is to combine advanced experimental, analytical and numerical techniques to investigate fundamental topics in heat transfer, and to design, construct and test real energy conversion systems. In this way, the research also contributes to the engineering and research training of the mechanical engineering students. The research is concentrated on three topics:

a. Heat transfer and transitional flows.
The research in this area is aimed at a better understanding of the fundamental characteristics of transitional flows in general. Flow cases that are studied are bypass transition along a flat plate (related to turbine blade cooling), laminar thermal transport in compact systems and boiling process control (for heat removal and thermal homogenisation in, for example, lithographic systems). Another research line concentrates on non-equilibrium phase transitions in gas-vapor mixtures.

b. Micro-scale heat transfer and flow phenomena
The aim of this research line is to achieve a better understanding of the heat and mass transfer processes at the small scales. The focus is on evaporative cooling of electronic components, on multi-scale analysis for compact heat storage materials and permeable geothermal reservoirs, and the dynamics of integrated fluid drivers in micro systems. On the smallest scales the physical processes are studied by coupling Molecular Dynamics analysis with a Direct Simulation Monte Carlo model.

c. Heat transfer engineering
The research activities in this area focus more on the system level rather than on the phenomenological level. Main research projects are fouling of heat exchangers used in waste- incinerators and biomass gasifiers, the design of a humidity harvesting device, and heat transfer models in the built environment. Another research line concentrates on biomass reactors for thermo-chemical applications.

More information about the research activities in these areas can be found on our website: www.energy.tue.nl

TU/e – Mechanical Engineering - Energy Technology

Project leaders
E.H. van Brummelen

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
E.H. van Brummelen, F. Toschi, A.J.H. Frijns, C. Livi, S. Mohammad Nejad, D.A.M. van der Woude, M.R.A. Abdelmalik

Cooperation
ASML

Funded by
ASML, NWO

Funded %
NWO 65%
Industry 35%

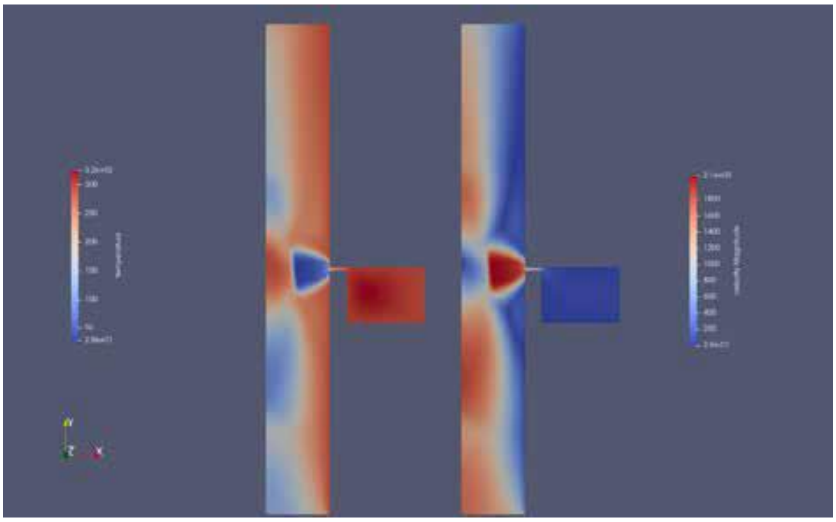
Start of the project
2018

Information
Harald van Brummelen
e.h.v.brummelen@tue.nl

Project title
RARETRANS : Transport in rarefied gases in next generation photo-lithography machines

Project aim
Investigation of gas/solid interactions for rarefied gases in simplified but representative geometries inspired by EUV PLM applications. Study of the fundamental transport physics of irregularly shaped particles in rarefied flows in the transitional regime. Investigation of the dynamic flow patterns and heat transport in the complete exposure chamber, including the interactions with contiguous solid walls.

Progress
The Moment method has been adapted in such a way that it can centre the approximation around a local estimate of the solution. This improves stability of the simulations when considering cases where multiple velocity and /or temperature scales are present in the solution.



TU/e – Mechanical Engineering - Energy Technology

Project leaders
E.H. van Brummelen, H.M.A. Wijshoff

Research theme
Complex dynamics of fluids

Participants
T.H.B. Demont

Cooperation
Canon Production Printing
University of Twente
Eindhoven University of Technology

Funded by
Canon Production Printing, NWO

Funded %
University 50%
Industry 50%

Start of the project
2017

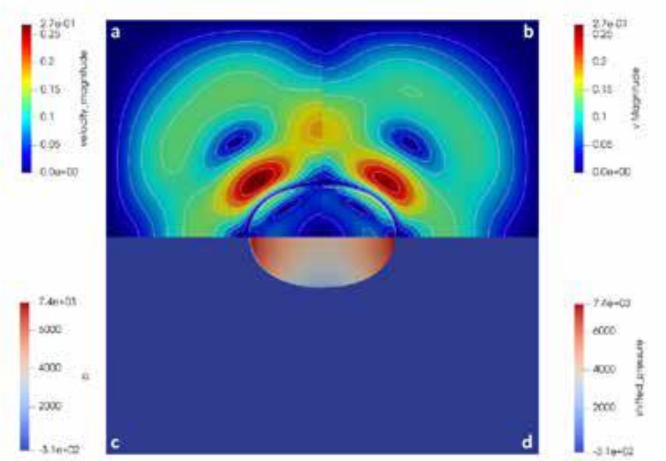
Information
Tristan H. B. Demont
06 571 409 41
t.h.b.demont@tue.nl

Project title
Modeling and simulation of dynamic wetting of binary fluids with surfactants

Project aim
The objective of this project is to develop novel mathematical models and corresponding numerical simulation techniques to investigate dynamic wetting of binary (liquid-vapor) fluids with surfactants. The envisaged modeling paradigm comprises a diffuse-interface model for the binary fluid, based on the recently proposed Navier-Stokes-Cahn-Hilliard (NSCH) model, in combination with separate transport equations for the surfactant in the liquid bulk and surface transport equations for the surfactant on the diffuse liquid-vapor and sharp liquid-solid interfaces.

Progress
An iterative solution method with Gauß–Seidel preconditioning has been implemented for the linearized systems, aiding the simulation robustness. Condition number estimates have been performed on the full NSCH system and the NS and CH subsystems. Together with adaptive time step sizes, a thin interface NSCH simulation has been performed for a — now possible to simulate — water droplet immersed in air. Results such as second moments, energies, and velocity and pressure profiles have been compared to those of the McDroplet model.

2020 Publications
van Brummelen EH, Demont THB, van Zwieten GJ. An adaptive isogeometric analysis approach to elasto-capillary fluid-solid interaction. Int J Numer Methods Eng. 2020; 1–22. https://doi.org/10.1002/nme.6388.



TU/e - Mechanical Engineering - Power & Flow

Prof.dr.ir. NG Deen
Prof.dr. JGM Kuerten
Prof.dr.ir. LPH de Goey
Prof.dr. M Golombok
Prof.dr.ir. AW Vreman
Prof.dr. DJEM Roekaerts

In view of the continuous increase in world energy demand, our vision is that combustion will remain a very important energy conversion process, even in the far future when fossil fuels are depleted, since heavy transport by road, air and water needs dense energy carriers, in other words liquid or solid fuels. An important issue in today's combustion is the shift to ultra-clean and highly efficient 'low-temperature' combustion methods. The second important issue is related to the fuel aspects: we will see increased use of biofuels, and in the longer term the emergence of fuels derived from sustainable sources like solar and metal fuels. Either way, it remains of utmost importance to optimize combustion devices, now in combination with different fuel formulations to minimize undesired emissions and maximize thermal efficiency. With the current level of development of practical combustion systems, further improvements will depend on details of the combustion-system and fuel-composition combination. More accurate and efficient validated models are required to describe the complex interplay between multiphase and/or reactive flows. All these topics fall within the broader theme of process technology, which combines complex flow phenomena with physical and chemical conversions.

The mission of the group is to provide education and to perform world-class scientific research on multiphase and reactive flows in the area of energy conversion and process technology, building a knowledge chain consisting of:

1. development of fundamental models based on first principles
2. experimental validation of these models
3. application and lab-scale demonstration of (reactive) multiphase contact equipment
4. development of predictive tools for practical and industrial applications, derived from the fundamental models based on first principles and experiment

Research themes

The research of the group is concentrated around three main topics:

1. Combustion systems and their fuels

This research topic is connected to the development of smart injection and combustion strategies of future ultra-clean and efficient combustion systems as well as with the after treatment, with a focus on future diesel engines. With respect to fuels we focus on three main activities: i) enhanced oil recovery, ii) use of bio-based fuels based on biomass components such as lignin, and iii) using micro-structuring gas-liquid bubbly flow processes to intensify biogas-to-liquid conversion.

2. Metal fuels as dense CO2-free energy carriers

This research topic is concerned with a novel type of fuels, i.e. metal powders that have a tremendously high energy density and can act as a major CO2-free energy carrier for the long term. Within the group we develop the combustion technology of metal powder, solid handling including separation and regeneration through chemical reduction.

3. Complex multiphase flows

This research topic is related to various applications in the field of process technology, all involving complex multiphase flow phenomena. This includes equipment with phase transitions, such as evaporation of sessile multi-component ink droplets, cooling of steel by water jets and water-steam flow in evaporator tubes.

TU/e - Mechanical Engineering - Power & Flow

<u>Project leaders</u>
J.G.M. Kuerten

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
RT van Gaalen, C Diddens,
HMA Wijshoff

<u>Cooperation</u>
Océ

<u>Funded by</u>
NWO, Océ

<u>Funded %</u>
NWO 50%
Industry 50%

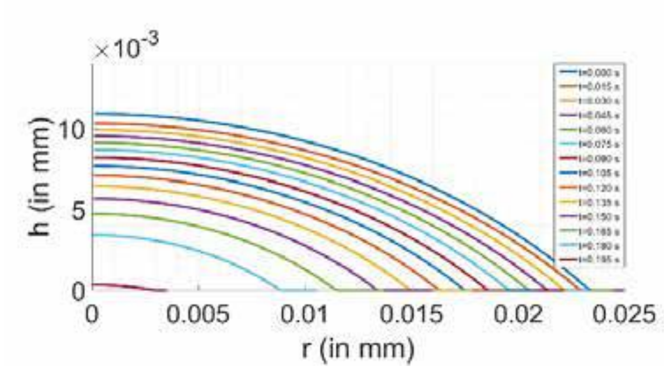
<u>Start of the project</u>
2014

<u>Information</u>
J.G.M. Kuerten
j.g.m.kuerten@tue.nl

<u>Project title</u>
Modeling drying droplets on porous substrates

<u>Project aim</u>
The aim of this research is to generalize an existing model for the evolution of a sessile ink droplet subject to evaporation and absorption into a porous substrate by three further aspects:
- the model will be extended to incorporate the presence and influence of surfactants;
- to investigate the impact of neighbouring droplets, a generalization to three dimensions is proposed;
- the absorption into the porous substrate will be extended to comprise more general types of porous substrates.

<u>Progress</u>
The model has been expanded to incorporate moving contact lines. Previously, this was done with a precursor film and now it can be done with a slipping contact line as well. Furthermore, the precursor film model has been
extended. Also, the soluble surfactant model has been improved to be more accurate and manageable. Lastly, a model involving salts in solution has been implemented.



Profile of an evaporating droplet with slipping contact line

TU/e - Mechanical Engineering - Power & Flow

Project leaders
BPM Esch

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Changliang Ye

Funded by
China Scholarship Council

Funded %
Scholarships 100%

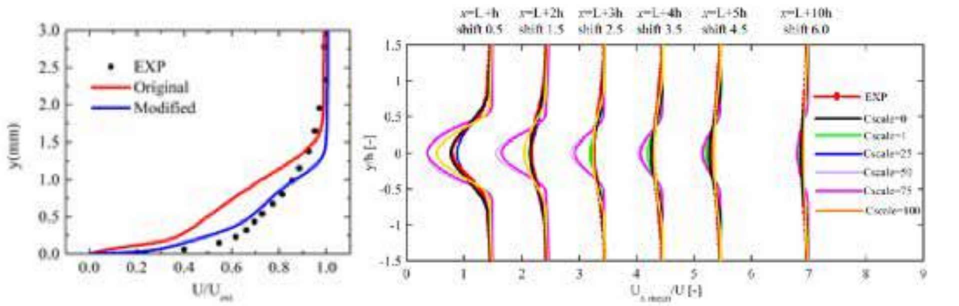
Start of the project
2018

Information
Changliang Ye
06 85220151
c.ye@tue.nl

Project title
Effect of rotor-stator interaction on rotating stall of water pump

Project aim
(1) Develop a high-precision numerical method suitable for the analysis of pump rotating stall. Focusing on solving the problem that the existing boundary layer transition model relies too much on the plate test results to reflect the large curvature effect of blade leading edge.
(2) Establish a combined hydrofoil with centrifugal pump geometric characteristics. Reveal how the rotor-stator interaction influences the hydrofoil stall by investigating the relationship between the characteristics of hydrofoil boundary layer transition.
(3) Reveal how the rotor-stator interaction influences the water pump rotating stall by investigating the pressure fluctuation characteristics of the flow field at different flow conditions.

Progress
1. The transition model SSTCC γ-Reθt considering curvature correction improves the prediction in the near wall region, the flow fields calculated by different correction coefficient are quite different. In this case, when the scaling coefficient Cscale is 100, the prediction in the near wall region and the wake region has achieved good results.
2. Modified Reθt is got by adjust the original Reθt for consistent with the LES results. The correlation function between Reθt and curvature is obtained by defining the curvature of hydrofoil. The SST γ-Reθt model is modified by applying the new empirical correlations to the original transition model. And the new model works well by verifications.
3. The unsteady flow field is divided by DMD method the characteristic frequency of the flow field and the corresponding flow mode are obtained, which can reduce the order of the actual flow.



(Left): Time average velocity distribution in X direction of wake area (considering curvature effect)
(Right): Velocity profiles at chord locations of NACA 66(mod)-312 hydrofoil (calculated by modified model)

TU/e - Mechanical Engineering - Power & Flow

Project leaders
N.J. Dam, N.G. Deen

Research theme
Complex dynamics of fluids

Participants
R. Doddema

Funded by
TU/e

Funded %
University 100%

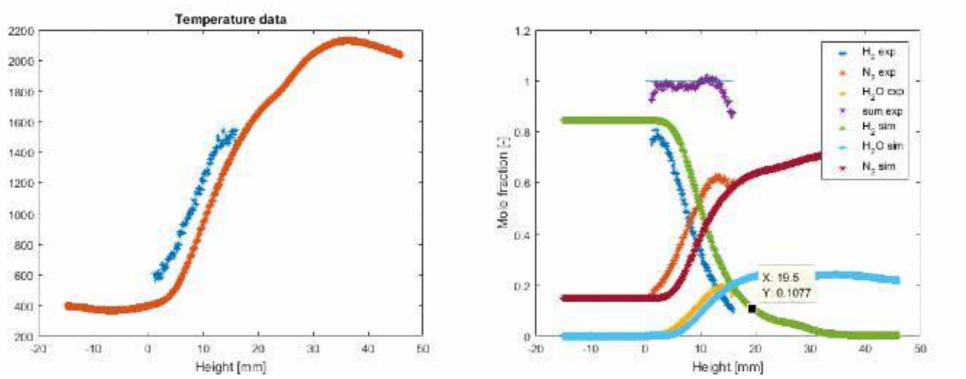
Start of the project
2015

Information
NJ (Nico) Dam
N.J. Dam@tue.nl

Project title
The role of aromatics in soot formation

Project aim
The aim of the project is to study the breakdown and reappearance of aromatics in vaporized liquid fuel non-premixed flames, using aliphatic fuels doped with aromatics. The focus lies on the development and application of (laser) optical detection techniques for intermediate species, found in the flame between the fuel pyrolysis zone and PAH formation zone, in order to gain a more detailed understanding of the found relation between aromatic fuel content and soot emission of combustion engines.

Progress
The existing Raman setup has been expanded with a 266 nm PLIF setup to study the non-premixed combustion of benzene and toluene. These aromatic fuels are vaporized and mixed in low quantities with hydrogen. The rapid outward diffusion and subsequent combustion of hydrogen creates high-temperature low-oxygen boundary conditions for the heavy dopants, while causing minimal interference in the optical experiments. Measurement results have been compared with numerical simulations and reasonable agreement is found (see the figure). Decreasing fluorescence signal of toluene with height in the flame is purely a result of strong temperature increase, not due to thermal dissociation of toluene. The optical setup has also been used to study contaminants in n-dodecane fuel, and also to characterize the oscillating density field in an acoustic levitator.



(left) Temperature derived from Raman scattering data in comparison to numerical calculations, (right), major species mole fractions from experiment and simulation

TU/e - Mechanical Engineering - Power & Flow

Project leaders
N.J. Dam, L.M.T. Somers, N.G. Deen

Research theme
Complex dynamics of fluids

Participants
N.C.J. Maes

Cooperation
Fiat Powertrain Technologies

Funded by
Industry, Fiat Powertrain Technologies

Funded %
University 67%
Industry 33%

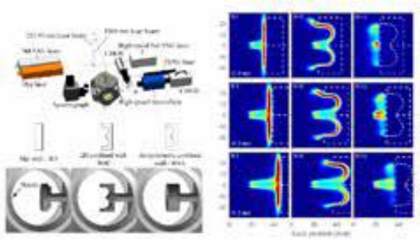
Start of the project
2014

Information
N.C.J. Maes
n.c.j.maes@tue.nl

Project title
Tracking joules: Flame-wall interaction in diesel spray combustion

Project aim
In a collaboration with Fiat Powertrain Technologies, heavy-duty Diesel sprays and the effect of flame-wall interaction are studied in detail using optical diagnostic techniques and temperature measurements in a constant volume vessel. The higher goal is to design a completely new, heavy-duty engine based on optimization through modelling. In order to achieve reliable and predictive models, it is essential to perform fundamental and reproducible experiments at relevant conditions.

Progress
To isolate the effect of flame-wall interaction from representative operating conditions of an internal combustion engine, experiments were performed in a constant-volume pre-burn vessel. Three different wall geometries were studied at distances of 32.8, 38.2, and 46.2 mm from a single-hole 0.09-mm orifice diameter fuel injector. A flat wall provides a simplified case of flame-wall interaction. To mimic the division of a jet into two regions by the piston bowl rim in an engine, a 2D confined wall is used. A third, axisymmetric confined wall geometry allows a second simplified comparison to numerical simulations in a Reynolds-averaged Navier-Stokes framework. As a limiting situation for a free jet, the distance from the injector orifice to the end wall of the chamber is 95 mm. Thermocouples installed in the end-wall provided insights in local heat-losses for reference cases without a wall insert. The test conditions were according to the Engine Combustion Network Spray A guidelines with an ambient temperature of 900 K, and an ambient density of 22.8 kg/m3 with 15% O2. Flame structures were studied using high-speed OH* chemiluminescence with integrated single-shot OH PLIF, and combined with pressure-based apparent heat-release data to infer combustion progress and spray behaviour. Soot was studied in a qualitative manner using high-speed natural luminosity imaging with integrated high-speed laser induced incandescence. Overall, increased mixing upon interaction with the surfaces is observed to increase early heat-release rate and to significantly reduce soot, with the nearest wall distance showing most effect. The flat wall gives rise to the most significant effects in all cases.



Left top: schematic overview of the constant volume combustion vessel with simultaneous high speed OH chemiluminescence, high-speed LII, OH PLIF, and spectral imaging. Left bottom: illustration of the three different wall shapes studied in this work. Right: ensemble averaged high-speed OH* stills of the different wall shapes and wall distances that were studied.*

TU/e - Mechanical Engineering - Power & Flow

Project leaders
L.M.T. Somers, F.P.T. Willems

Research theme
Complex dynamics of fluids

Participants
RC Willems, LMT Somers, FPT Willems, NG Deen

Cooperation
DAF Trucks, TNO, Shell Global Solutions, Delphi Technologies, Sensata, AVL Dacolt

Funded by
TTW, DAF Trucks, TNO

Funded %
NWO 70%
Industry 30%

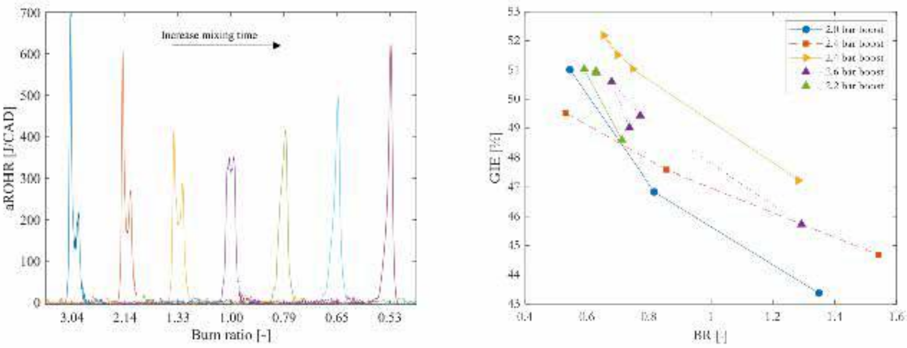
Start of the project
2016

Information
LMT (Bart) Somers
l.m.t.somers@tue.nl

Project title
Towards a HiEff engine

Project aim
This research targets a premixed combustion concept called reactivity-controlled compression ignition (RCCI), which potentially facilitates high gross indicated efficiency (GIE) and low levels of nitrogen oxides (NOx) and soot emissions.

Progress
Results of gasoline-diesel RCCI operation on the XEC platform were published. Tthe RCCI mode was compared to CDC in energy distribution diagrams and specific emissions. Key outcome is that RCCI outperforms CDC only at elevated boost pressures, as heat transfer losses are then strongly suppressed. NOx emissions are extremely low, especially at high boost due to lean mixtures, which readily prevents local high-temperature combustion. By detailed analysis of heat release a new metric is proposed that sheds light on heat release rate shape. The burn ratio (BR) is defined as (CA90-CA50)/(CA50-CA10). BR seems to predict the rate of heat release shape well, which displays a two-phase combustion for high burn ratios and transitions into a single-phase regime for lower values. The mixing time of fuel and air is indicative for the BR. Peak GIE is reached at lowest values of BR, giving the best trade-off between thermal and combustion efficiency.



Burn ratio predicts the heat release shape and peak GIE well.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
J.G.M. Kuerten, C.W.M. van der Geld

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
E.J. Gelissen

Funded by
NWO/TTW

Funded %
NWO 100 %

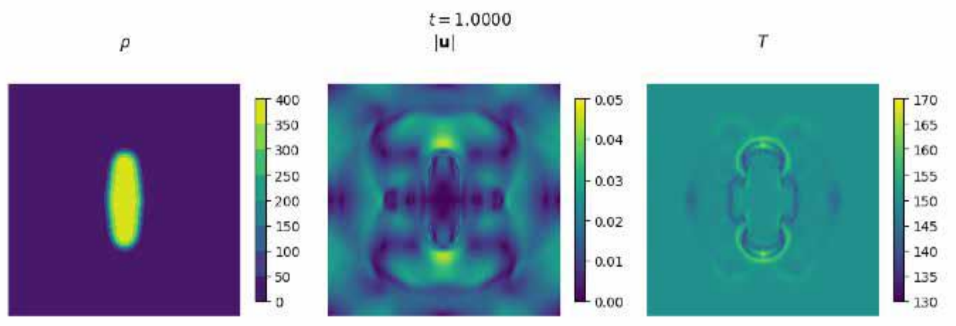
Start of the project
2016

Information
E. Gelissen
e.j.gelissen@tue.nl

Project title
The diffuse interface model for phase-transitional flows

Project aim
Studying phase-transitional flows through numerical simulations using a Diffuse Interface Model (DIM). The Diffuse interface Model is based on the Navier-Stokes-Korteweg equations and uses the Van der Waals equation as the equation of state.

Progress
DIM is used to perform simulations of droplet collisions in three spatial dimensions under non-isothermal conditions. DIM is compared with a different method often used for multiphase flows: the Local Front Reconstruction Method (LFRM). Results for fully three-dimensional simulations of droplet collisions at relatively high Weber number are presented and compared. DIM is used to perform simulations of droplet impacts on a heated solid surface, using an especially constructed solid wall boundary condition which enables simulations with different wetting conditions. The model is also extended to include the effects of surface roughness on the behaviour of the contact line dynamics.



TU/e - Mechanical Engineering - Power & Flow

Project leaders
B.P.M. van Esch, C.W.M. van der Geld, JGM Kuerten

Research theme
Complex dynamics of fluids

Participants
C.F. Gomez

Cooperation
TATA Steel R&D Ijmuiden, M2i

Funded by
NWO-I, TATA Steel R&D IJmuiden

Funded %
NWO 50%
Industry 50%

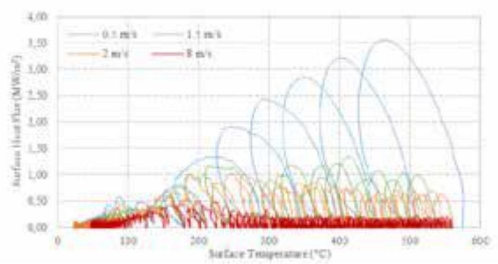
Start of the project
2016

Information
C.F. Gomez
c.f.gomez@tue.nl

Project title
Explaining boiling beyond boiling temperature: Quench cooling of hot steel plates

Project aim
During its production, steel is quenched with water jets in the so called Run Out Table (ROT). During quenching of steel at high temperature (900 °C), the boiling process has quite extraordinary features. Rewetting, i.e. contact with the surface, occurs at temperatures far beyond the boiling temperature. The first goal is to get proper understanding on the rewetting phenomena, based on direct observations of the stagnation zone during quenching. The second goal is to develop heat transfer coefficients correlations to implement in the process control system of the ROT.

Progress
The addition of a linear unit to the original setup allows quenching of moving steel surfaces, at speeds up to 9 m/s. This speed range is comparable to industrial conditions and higher than ever reported in lab studies. The updated setup was validated showing very satisfactory reproducibility. The high-speed recordings show that, as expected, the surface speed affects the boiling regimes occurring during quenching. At initial temperatures around 550 C, we observe explosive boiling at speeds between 0.5 and 6.5 m/s. At speeds higher than 6.5 m/s, the surface motion allows the formation of a vapor film, leading to film boiling. The presence of film boiling at these conditions is well known in industry. As expected, the surface speed also shows an effect on the heat flux estimations. The experimental results show that higher surface speeds lead to overall lower heat fluxes. The high temperature side of the boiling curve seems to be more affected, showing a change of trend as well. As a result of the intermittent and gentler cooling, high speeds also lead to lower temperature gradients along the plate thickness. The more homogeneous cooling is expected to reflect on more homogeneous mechanical properties.



(Left) Sandblasted surface and water jet at 25 C. Quenching of moving plates setup (Right) Effect of surface speed on the boiling curve.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
F.P.T. Willems, L.M.T. Somers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
B. Akkurt, X. Luo, N.G. Deen, M. Steinbuch

Cooperation
TTW, TNO, Sensata, DAF, Delphi

Funded by
TTW, TNO, Sensata, DAF, Delphi

Funded %
Industry 80%
TNO 20%

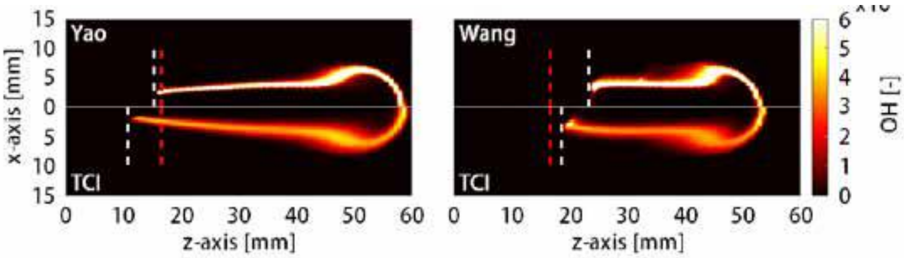
Start of the project
2014

Information
LMT (Bart) Somers
l.m.t.somers@tue.nl

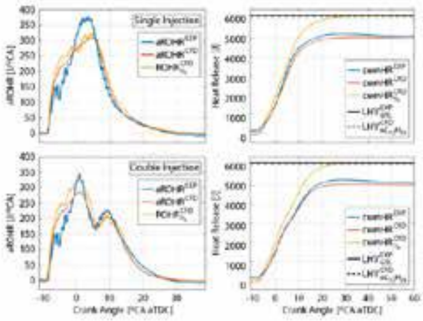
Project title
Heat2Control

Project aim
This research project focuses on the development of new modeling technique for high EGR diesel combustion concepts with multiple injection fueling systems, which is seen as an essential step towards future RCCI concepts. The CFD-FGM model, which will be extended for multi-pulse injection strategies, will be extensively validated with experimental data.

Progress
The combustion modelling part of the project is continued with model validation. Prior to engine simulations and analysis of the results, Turbulence- Chemistry interaction (TCI) is studied with the constant volume application. In this manner, Spray-A simulations are repeated with the FGM tables that considered TCI. In addition to that, the FGM-CFD model is validated with the Cyclops engine experimental data at various operating points, at which the engine speed and the load are varied (i.e. A30, B30 and B50 operating conditions), with motorized cycles (cycles without fuel injection) and single and double injection strategies. The experimental data is obtained with the CYCLOPS engine, which is a test rig based on DAF XE 355 C straight 6-cylinder heavy duty direct injection diesel engine. Also, the model validation is extended for NOx emission.



Comparison of lift-off lengths (red dashed line for the experiment and white dashed line for the simulations) and OH species mass fraction at the quasi-steady state, t=1.3 [ms], with and without TCI is considered at the nominal Spray-A condition



Comparisons of the a ROHR (left-column) and the cumulative heat release (right-column) between the CFD and experiments at B50 with single (top-row) and double (bottom-row) injections

TU/e - Mechanical Engineering - Power & Flow

Project leaders
JCH Zeegers, JGM Kuerten, AA Darhuber

Research theme
Complex dynamics of fluids

Participants
R Dellaert, S Tajfirooz, JCH Zeegers, JGM Kuerten

Cooperation
TU Delft, UTwente, UU, Radboud University, and several companies, part of Perspectief programme TTW

Funded by
TTW, Umincorp, Sumitomo, Syngenta, Dimaen, FNLI, AEB

Funded %
NWO 70%
Industry 30%

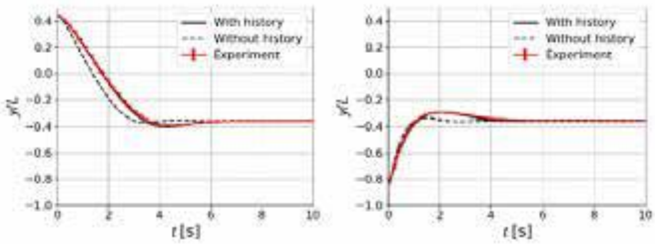
Start of the project
2016

Information
S (Sina) Tajfirooz
S.Tajfirooz@tue.nl

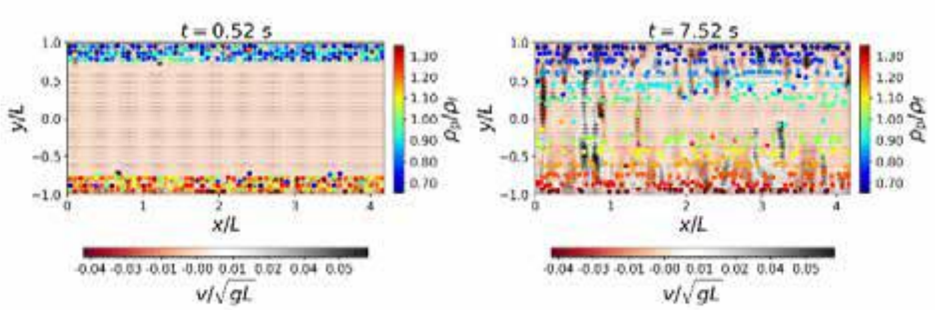
Project title
Fluid dynamics of Magnetic Density Separation (MDS)

Project aim
The project consists of two sub-projects. The aim of the first sub-project is to investigate the temporal and spatial characteristics of turbulence behind a honeycomb structure. The goal is to minimize the turbulence level behind the flow straightener used in the MDS setup. The second sub-Project aims at investigation of particle-fluid-particle interactions in an MDS setup. A combination of experimental and numerical studies is carried out. And the combined research outcomes of the two sub-projects will be used for optimizing the magnetic density separation technology.

Progress
- Direct numerical simulations are carried out to investigate the production and decay behaviour of flow instabilities behind a honeycomb structure.
- A point-particle Euler-Lagrange approach is applied to investigate the dynamical motion of spherical particle(s) in paramagnetic and superparamagnetic liquids subject to external magnetic field gradients. Numerical results of particle dynamics in single- and two-particle systems are validated against experimental observations. An excellent agreement is observed between the numerical and experimental results.
- Magnetic density separation of spherical particles in many-particle systems consisting of up to 0.5 million particles are studied numerically.



Effect of history force on levitation dynamics of a falling (left) and a rising (right) 5-mm spherical particle in a paramagnetic liquid



Magnetic density separation of 4-mm spherical particles in a paramagnetic liquid. Particles are coloured based on their mass densities. The horizontal colour bar corresponds to the vertical component of fluid velocity.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
LMT Somers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
H Bao

Funded by
China Scholarship Council

Funded %
Scholarships 100%

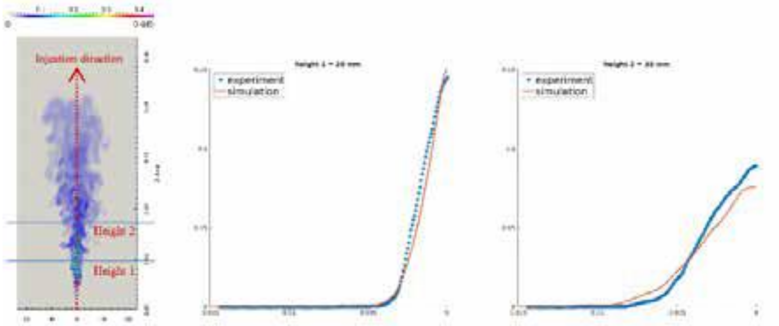
Start of the project
2017

Information
H. Bao
h.bao@tue.nl

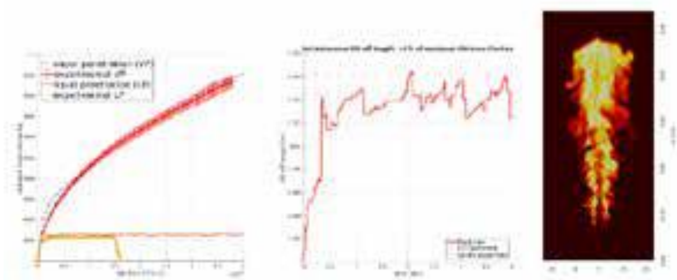
Project title
Modelling advanced engine combustion concepts with FGM in an LES framework

Project aim
Reactivity Controlled Compression Ignition (RCCI) engines realize low emissions as well as high efficiency. By blending the port-injected low reactivity fuel with the high reactivity fuel in cylinder, the fuel reactivity is tuned and the combustion phasing is thus controlled. The Project aims to provide better understand on the spray and interaction with the chemistry by numerical modelling. Code implementation and model development for RCCI engines are expected.

Progress
The turbulence model has been improved and validated by ECN (Engine Combustion Network) Spray A non-reacting case, penetration and fuel mixing can be well captured. The inclusion of strain rate effect in FGM is implemented and shows good agreement with the experiment. The new model can be further applied in simulating dual fuel cases.



(Left) Mass fraction of injected fuel (n-Dodecane)
(Right) Comparison of predicted radial mixture fraction (using the time-average from 1,2 ms to 5 ms) and experiment.



Left: Prediction of penetration Middle: Prediction of flame lift-off Right: Snapshot of temperature at 4 ms

TU/e - Mechanical Engineering - Power & Flow

Project leaders
L.P.H. de Goey

Research theme
Complex dynamics of fluids

Participants
D Ning, Y Shoshin, JA van Oijen

Funded by
Chinese Scholarship council

Funded %
Scholarships 100%

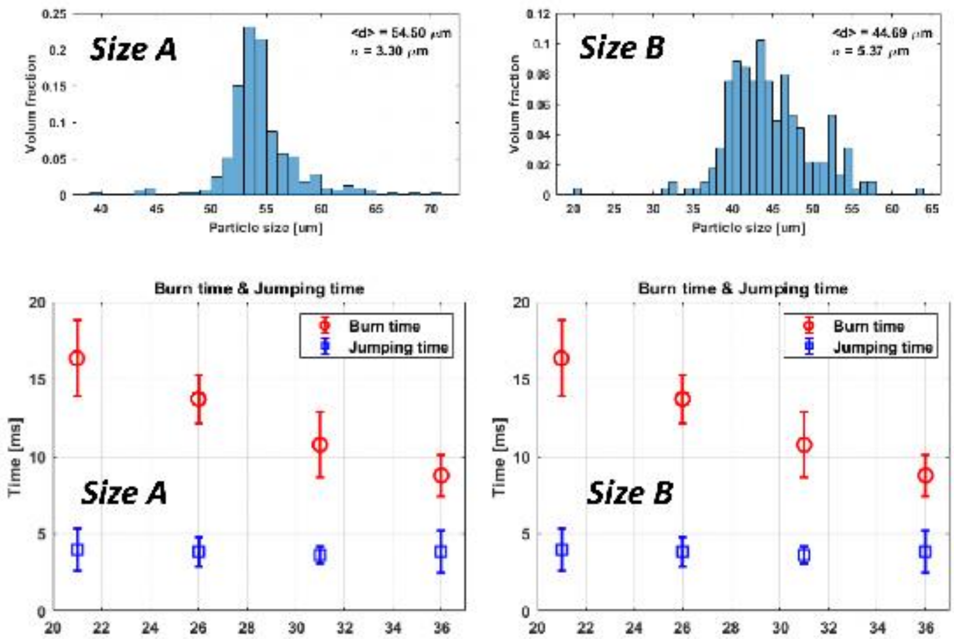
Start of the project
2018

Information
D. Ning
d.ning@tue.nl

Project title
Studies on direct combustion of iron powder as a carbon-free energy carrier

Project aim
Iron powder was thought as the primary metal fuel whose combustion product is porous solid oxide that can be easily captured. Meanwhile, it is readily recycled with well-established technology. For the successful application of the iron as a renewable carbon-free fuel, some unique combustion properties of the iron powder, compared to our traditional gaseous and liquid hydrocarbon fuels, must be considered in any combustor design for future iron-fueled engine technologies. Therefore, this project will focus on the fundamental characteristics of iron powder combustion and single iron particle burning. These research data will be very useful for the design of future iron-fueled combustors.

Progress
Burn time of micro-sized iron particle in two size ranges burning in different O2 concentration has been studied.



Measured burn time and corresponding particle size distribution

Project leaders
LPH de Goey, I Lopez Arteaga, V Kornilov

Research theme
Complex dynamics of fluids

Participants
M Kojourimanesh

Cooperation
NWO (TTW), ATAG, Remeha, Honeywell, Bekaert

Funded by
NWO (TTW)

Funded %
NWO 100%

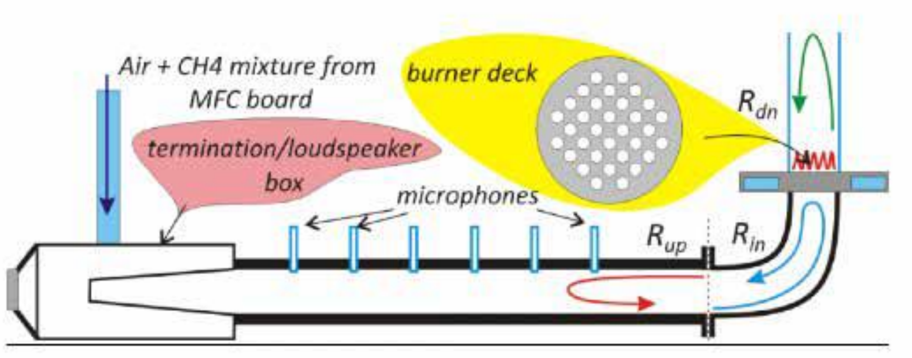
Start of the project
2018

Information
M. Kojourimanesh
m.kojourimanesh@tue.nl

Project title
System based thermo-acoustic design of central heating equipment

Project aim
The goal of this project is to introduce and develop a system level approach to address the challenge of thermo-acoustic design and control of acoustic instability in combustion appliances. This system level approach enables unique new modeling and experimental identification strategies to describe the thermos-acoustic response of flames/burners.

Progress
Internal linear thermo-acoustic properties of flames are fully described by the transfer function, temperature and flow area jumps. In order to compose a model to predict the onset of thermo-acoustic instability of combustion, one has to characterize the thermo-acoustic properties of the flame and also components of upstream and downstream sides of the burner. This kind of modelling strategy usually faces practical problems related to the measurement of the reflection coefficient at the hot downstream part of the system, Rdn. However, a novel method is presented to assess thermo-acoustic instabilities based on reflection coefficients measured only from the cold (upstream) side of the burner. Both reflection coefficients at the cold side, Rin and Rup, can be readily measured using standard acoustic techniques, like using the impedance tube shown in the figure. The proposed method is experimentally validated for burners with premixed burner-stabilized Bunsen-type flames.



Experimental setup of impedance tube and combustor.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
Y Tang, NG Deen

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
X Liu

Funded by
China Scholarship Council (CSC)

Funded %
Scholarships 100%

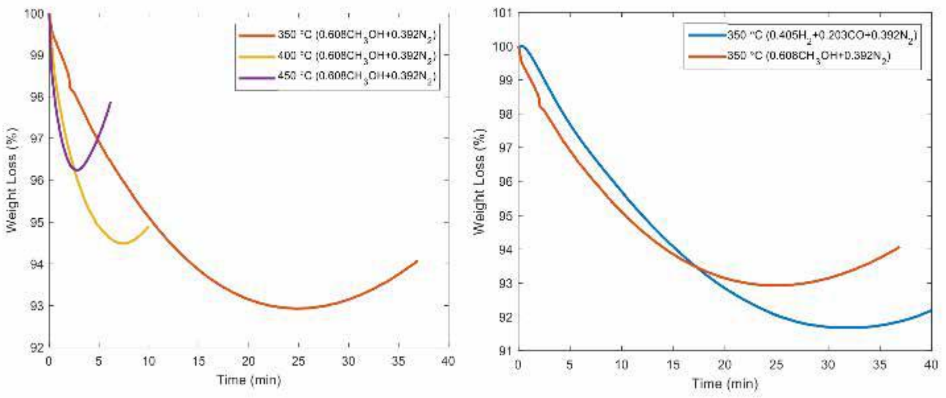
Start of the project
2018

Information
X. (Xin) Liu
x.liu4@tue.nl

Project title
Methanol reduction of recyclable iron oxide particles

Project aim
The understanding of the reduction mechanisms of iron oxide particles is important for the entire recycle process of iron fuels. In this project, experimental and numerical study on the reduction of iron oxides using methanol at low temperatures will be carried out. The aim is to get a detailed understanding of the reduction mechanism of iron oxide particles, and further provide guidance for the practical application of the project.

Progress
Thermogravimetric analysis (TGA) of the reduction of lab-made iron oxides in a continuous stream of syngas or methanol was conducted at temperatures ranging from 350 to 450 °C. The effect of reducing atmosphere and temperature on the reduction behaviour of iron oxide particles was investigated. Results showed that for methanol reduction, the optimal reduction temperature is 350 °C, and the initial conversion rate increases rapidly when the reaction temperature increases from 350 °C to 400 °C. In addition, it is easy to form carbon deposits for all test temperatures. In our current experiments, we tested the reduction performance when using the same amount of methanol and syngas. It can be seen that using syngas as reducing agent can get higher conversion degree of iron oxides, whereas atmosphere involving methanol can accelerate the rate of reduction in the initial phase of the reduction period.



Left: Weight loss curve under different temperature - Right: Weight loss curve under different reducing agents

TU/e - Mechanical Engineering - Power & Flow

<table>
<tr><td>

Project leaders
J. A. van Oijen

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
S. Karaca

Cooperation
MariGreen, TUBITAK

Funded by
MariGreen, TUBITAK

Funded %
Industry 100%

Start of the project
2016

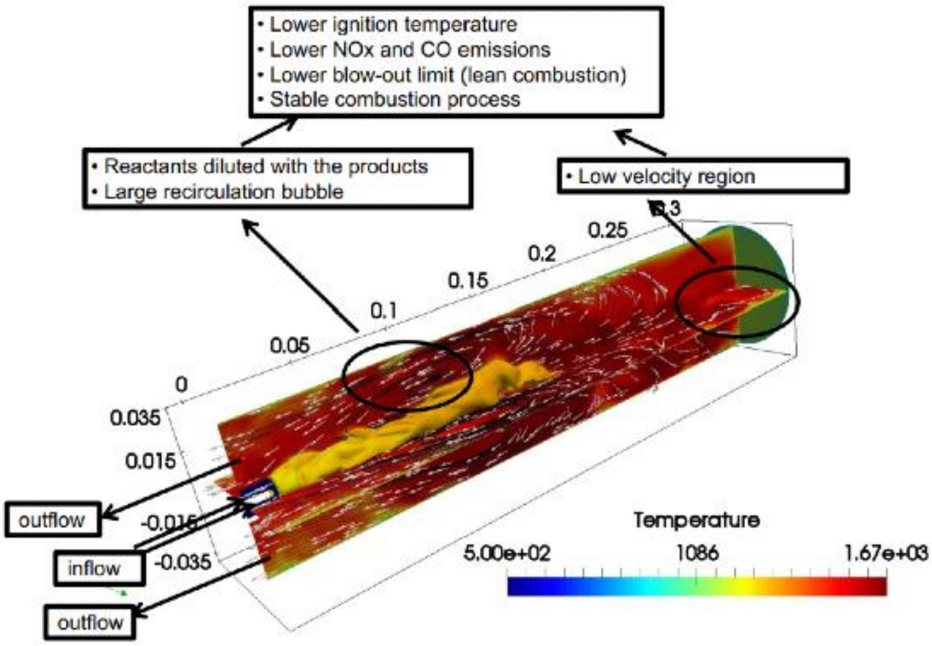
Information
J. A. van Oijen
j.a.v.oijen@tue.nl

</td><td>

Project title
Combustion modelling of reversed flow combustors

Project aim
Stabilization of flame is a crucial issue in combustors. In literature there are many ways to stabilize the flame such as swirl, bluff body and reversed flows. In this project very lean aero-engine burners with reversed flow configuration will be investigated by using chemistry reduction method – FGM. The aim is to understand flow and chemistry part of the reversed flow configuration and develop accurate numerical tools to predict emissions like CO and NOx.

Progress
A model to predict emissions like CO is constructed to the OpenFOAM solver. Heat loss effects are investigated by using different wall temperatures. Effects of turbulence momentum closure models for LES also investigated in this burner.



Stagnation Point Reverse Flow combustor configuration

</td></tr>
</table>

TU/e - Mechanical Engineering - Power & Flow	
Project leaders AW Vreman, NG Deen Research theme Complex dynamics of fluids Participants A Zarghami, TAM Homan Cooperation MT de Groot (Nouryon) J van der Schaaf (Chem. Eng.) Funded by RVO, Nouryon Funded % Industry 100% Start of the project 2018 Information A.W. Vreman a.w.vreman@tue.nl	Project title Computational fluid dynamics simulations of flows in alkaline water electrolyzers (Alkaliflex) Project aim Research on bubbly flows in electrolyzers is performed with the aim to understand and relieve limitations of alkaline water electrolysis. Progress Bubbly flows of high gas volume fraction in electrolyzers have been simulated using the Ansys Fluent solver. More specifically, the Euler-Euler approach has been used to simulate the multiphase flow in the hydrogen compartment of an electrolyzer and the effect of various outflow boundary conditions has been investigated.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
TAM Homan, Y Tang, NG Deen

Research theme
Complex structures of fluids

Participants
CJM Hessels

Funded by
TU/e

Funded %
University 100%

Start of the project
2018

Information
C.J.M. Hessels
c.j.m.hessels@tue.nl

Project title
Reduction of combusted iron using hydrogen

Project aim
To maintain a stable energy grid, it is important to look for ways to store and transport renewable energy in large amounts and at high energy density. We can envision a “metal fuel cycle” in which metal powder is combusted in slightly altered coal power plants and the produced metal-oxides are reduced back to metal powder using renewable energy. The research focusses on the reduction of combusted iron particles using environment-friendly produced hydrogen in a fluidized bed reactor. Both numerical (CFD-DEM) and experimental techniques will be used.

Progress
The focus of the first year lays on determining the reaction kinetics of a single iron oxide particle. On the numerical side a 1D single particle reduction model is being developed, based on the shrinking core model. An analytical solution is derived and the influence of hydrogen on fluid properties is analyzed. Experimentally, particle analysis on combusted iron is performed using scanning electron microscopy (SEM) and quantitative x-ray powder diffraction (Q-XRD). Initial results show hollow spherical particles, completely oxidized to a ratio of hematite and magnetite.

SEM image of combusted iron particles (~20 µm). Some particles appear to be hollow, which might be caused by impurities turning into gas during combustion

TU/e - Mechanical Engineering - Power & Flow	
Project leaders BPM Esch Research theme Complex dynamics of fluids Participants X Shen Funded by China Scholarship Council Funded % Scholarships 100% Start of the project 2019 Information X (Xi) Shen x.shen@tue.nl	Project title Investigation on the instability of tip leakage vortex cavitation and induced suction-side-perpendicular cavitating vortices in an axial flow pump Project aim (1) Elucidating the coherent mechanism of SSPCV induced by TLV and sheet cavitation. Studying the turbulence characteristics in the tip region to reveal the mechanism of PCV inception. (2) Analyzing the position of the PCV inception and the orientation of vortex vectors, to reveal the evolution of PCV, in terms of its generation, development and collapse. (3) Investigating the interference of PCV and adjacent blade in different operating conditions with high-speed photography and numerical simulation. The flow passage jamming, sudden drop of the blade load and flow instability will be discussed. Progress (1) The hydrofoil 'Delft Twist 11'is utilized to verify the applicability of numerical simulation of cavitating flow. The transient process of sheet cavitation on the surface of hydrofoil, including inception, development and collapse, will be simulated and compared with the experiments. (2) The hexahedral structured grids are used with thirty-five mesh nodes in the tip gap. The grid quality will be discussed to meet the requirements of large eddy simulation. (3) The three-dimensional structure and evolution of SSPCVs in an axial flow pump will be studied with unsteady calculations. The effects of TLV, lateral jet and pressure oscillation on the SSPCVs will be discussed, as well as the characteristics of velocity field, pressure field, vorticity field and turbulent kinetic energy and its generation. The high-speed photography and pressure fluctuation measurements are used to study the evolution of SSPCVs.

<table>
<tr><th colspan="2">TU/e - Mechanical Engineering - Power & Flow</th></tr>
<tr><td>

<u>Project leaders</u>
MT de Groot, AW Vreman

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
AM Meulenbroek, AW Vreman, NG Deen, R Lira-Garcia-Barros, MT de Groot, J van der Schaaf

<u>Cooperation</u>
Chemical Reactor Engineering, Department of Chemical engineering

<u>Funded by</u>
Rijksdienst voor ondernemend Nederland (RVO), Nouryon

<u>Funded %</u>
NWO 85%
Industry 15%

<u>Start of the project</u>
2019

<u>Information</u>
AM (Aled) Meulenbroek
a.m.meulenbroek@tue.nl

</td><td>

<u>Project title</u>
Alkaliboost

<u>Project aim</u>
The objective of the Alkaliboost project is to investigate alkaline water electrolysis that operates at high current densities and pressures so that the product capacity of the electrolytic cells can be increased. The project is divided into two components: an electrolytic cell for operation at high pressure operation is built and numerical simulations of the bubbly flow in an electrolytic cell are performed and compared to the experimental measurements. Electrolytic cell design can be a costly and time-consuming and optical access is limited, but the multi-phase simulations give additional insight and reduce the number of cell prototypes to be tested in the lab.

<u>Progress</u>
A bachelor project has been started on the rise velocity and coalescence of bubbles that should provide closure relationships for the Euler-Lagrangian model to be built in the form of an in-house code. At the moment, Comsol is used to study mass transfer, heat transfer and Marangoni flow in the vicinity of a single bubble attached to a gas evolving electrode is being performed. These results give a better understanding of the flow and transport phenomena relevant for bubbles at the electrode and this could also lead to improved closures in the Euler-Lagrangian model.

</td></tr>
</table>

TU/e - Mechanical Engineering - Power & Flow

Project leaders
NG Deen, Y Tang

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
R Subburaj

Cooperation
MCEC, P Bruijnincx (UU)

Funded by
MCEC

Funded %
Scholarships 100%

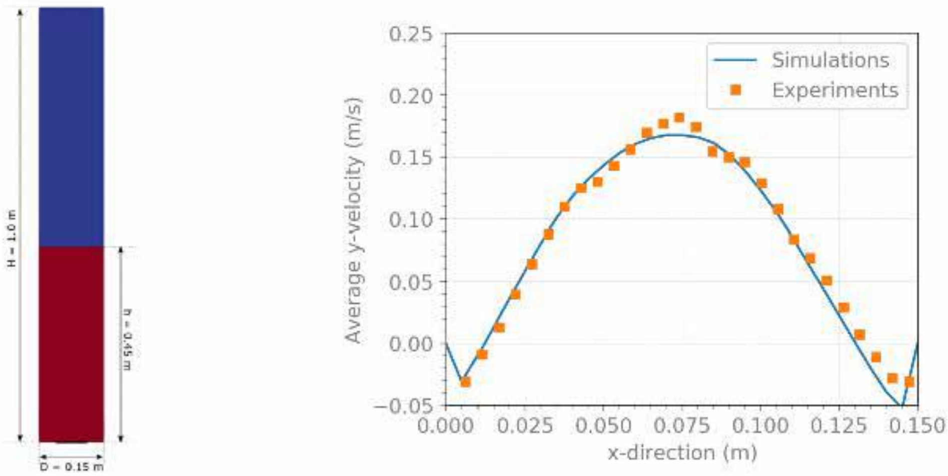
Start of the project
2019

Information
R (Rahul) Subburaj
r.subburaj@tue.nl

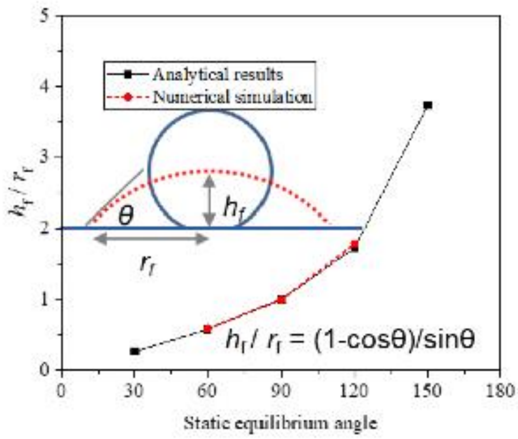
Project title
Micro structured slurry bubble columns for CO_2-to-MeOH

Project aim
We will investigate a micro-structured slurry bubble column reactor as the preferred process for Methanol production. In slurry bubble columns the reactant gases are bubbled through an inert liquid that is used to carry the catalyst and to act as a heat sink. Wire meshes are used to cut the bubbles We will first compare the slurry bubble column process with conventional reactor types using simple empirical reactor models.

Progress
The process at hand is modelled using a CFD-DEM approach. That is, the liquid is described as a continuous phase, whereas the bubbles are modelled in a Lagrangian manner. The motion of the free surface is modelled with a VOF description. The preliminary simulations were performed using the software OpenFOAM. An illustration of the domain and the average liquid velocity (compared with experiments) is shown in the figure.



Column (left), Average vertical liquid velocity profile at h = 0.25m (right)

TU/e - Mechanical Engineering - Power & Flow	
Project leaders Y Tang, NG Deen Research theme Mathematical and computational methods for fluid flow analysis Participants Tianqi Tang Cooperation Yurong He, (Harbin Institute of Technology) Funded by China Scholarship Council (CSC) Funded % Scholarships 100% Start of the project 2019 Information T. (Tianqi) Tang t.tang@tue.nl	Project title Numerical simulation on liquid transfer characteristics between colliding particles Project aim Wet particle system has a potential application in many industrial processes, such as chemical, pharmaceutical and food industries. In such processes, particles collide with droplets and wet particles themselves. It is interesting to investigate how liquid transfer between particles for further tracking agglomerations forming. In this project, numerical study on liquid transfer behaviours between colliding particles will be carried out. The aim is to get a detailed and in-depth understanding on liquid transfer mechanism between particles, and further track particle agglomeration forming and breakup in industrial processes. Progress - Design a research plan and discuss a numerical method: Volume of Fluid (VOF) is applied for tracking gas-liquid flow, Immersed Boundary Method (IBM) is applied for particle motion - A preliminary test of droplet spreading process on a solid boundary was conducted - A procedure to impose contact angle on convex solid boundary is trying to be implemented with Height Function model in order to simulate droplet spreading on a particle in Basilisk - Particle motion and collision modules will be modified and implemented based on Basilisk.  *Comparison between analytical results and numerical simulation for droplet spreading on a flat*

TU/e - Mechanical Engineering - Power & Flow	
Project leaders RJM Bastiaans, LPH de Goey Research theme Complex dynamics of fluids Participants Y Wang Cooperation Turbotec Funded by CSC Funded % Scholarships 100% Start of the project 2019 Information Y (Yalin)Wang y.wang12@tue.nl	Project title Numerical simulation and experimental study of co-combustion of biomass and coal Project aim The goal of this project is to research the co-combustion of biomass and coal from different co-combustion forms (gas, solid and liquid). The goal is to find the best mixture ratio and best co-combustion form based on simulation and experiments results and gradually decrease the use of coal. Progress Yalin Wang firstly designed a feeding device which could mix the biomass and CWS (Coal-Water-Slurry) together before the boiler. Now she is doing the simulation of the mixture gas of biomass and coal using CHEM1D software under the supervision of RJM Bastiaans and LPH de Goey. She will do the experiments on the HFM setup.

TU/e - Mechanical Engineering - Power & Flow

Project leaders
LPH de Goey, I. Lopez Arteaga

Research theme
Complex dynamics of fluids

Participants
V. Kornilov, A Aulitto

Cooperation
Bekaert Combustion Technology, Ansaldo Energia, Siemens Industry Software, IfTA GmbH, Univ. of Genoa, KTH, Keele Univ., TU Munich, Univ. of Pisa

Funded by
Pollution Know-How and Abatement (POLKA) project funded by European Union

Funded %
EU 100%

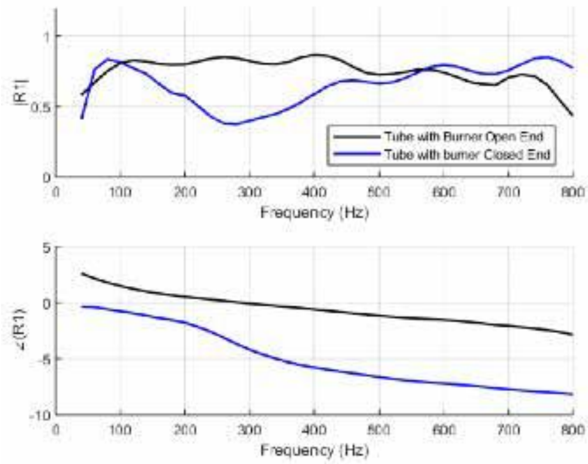
Start of the project
2019

Information
V (Vertika) Saxena
06 83788262
v.saxena@tue.nl

Project title
Passive measures of thermo-acoustic instability control in boilers with complex burner geometry

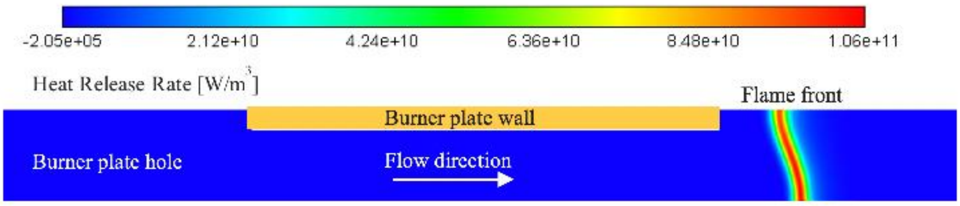
Project aim
The aim of this project is to prevent the occurrence of thermo-acoustic instability in a system. This instability results in large-amplitude pressure oscillations caused by interaction between the flame and acoustic waves; they can cause major hardware damage. The research will concentrate on developing new physical insights of the phenomenon (using both experiments and numerical simulations) which will be used in construction of stable hydrogen-fueled combustion systems.

Progress
Experiments were conducted to determine the flame transfer function (using hot wire anemometer and photomultiplier tube) for various burners in varied operating conditions (mean flow velocity and equivalence ratio of methane and air mixture). This experimental data was fitted to a polynomial function to be used in dispersion relations. Reflection coefficient of flat plate perforated burners and burners will aerodynamically complex structures was obtained to determine their burner transfer matrix. It was observed that while the former burners do not reflect sound waves, the latter burners are not acoustically transparent and have sound reflection on their surface as shown in the figure. A study on probability of stability of flat plate perforated burners with a variable reflection coefficient termination in the upstream of flame and burner was also undertaken.



Reflection coefficient of an aerodynamically complex burners

TU/e - Mechanical Engineering - Power & Flow	
Project leaders LPH de Goey, RJM Bastiaans, JA van Oijen Research theme Complex dynamics of fluids Participants M Hulsbos, Y Shoshin Cooperation DSM, NVV, Metalot 3C Funded by DSM, NVV, Metalot 3C Funded % NWO 85% Industry 15% Start of the project 2019 Information M (Mark) Hulsbos 06 51692686 m.r.hulsbos@tue.nl	Project title Metal-to-Power. Towards implementation of metal powders as CO 2-free fuels, development of experimental and numerical tools Project aim In this project we strive to provide knowledge on flame propagation of metal dusts, by studying the behaviour of elemental dust flame fronts as a function of different parameters. The main objective is to create numerical and experimental tools which can help providing this knowledge on flame propagation of metal dusts. Progress A literature study is started concerning the combustion process of iron and the pneumatic conveying of powders. Also, some PIV measurements are done to visualize particulate flows. Further, a start has been made with the build-up of the starting point heat flux burner. A graduation project regarding the dispersion of iron powders has also been started.

TU/e - Mechanical Engineering - Power & Flow	
Project leaders LPH de Goey, JA van Oijen Research theme Complex dynamics of fluids Participants MB Raida Funded by University Funded % University 100% Start of the project 2016 Information MB (Marie) Raida m.b.raida@tue.nl	Project title Further development of the heat flux method - extended pressure range and fuel variety Project aim This research project focuses on extending and applying one of the most accurate experimental methods for laminar burning velocity measurements - the Heat Flux Method. Previous measurements with this method have been typically conducted for gaseous fuels in moderate pressure ranges and for liquid fuels at atmospheric conditions. The current work will adapt the method to a greater range of pressures for a larger variety of fuels. A novel setup will be designed and built. Laminar burning velocity measurements will be performed for several gaseous and liquid fuels including new fuels and mixtures produced from more sustainable sources such as biomass and solar energy (solar fuels). Progress Extensive laminar burning velocity measurements of ethanol-air and methanol-air flames have been performed on the novel Heat Flux Burner setup for atmospheric and elevated pressures. The results have been compared with other experiments in literature and 1D modeling results using detailed kinetic mechanisms. Under conditions of elevated pressure, the flame front becomes thinner and moves closer to burner plate. Therefore, the flame may show small scale distortions, if the upstream fluctuations in flow, heat and mass transfer are not damped before the mixture reaches the reaction layer. To investigate this high-pressure behaviour in detail, the geometry of the perforated burner plate was analyzed with axisymmetric simulations in Ansys Fluent with focus on influence of the geometry of the burner plate.  *Results of numerical simulation of flame stabilization on a single hole of the burner plate at a pressure of p = 5 bar.*

TU/e - Mechanical Engineering - Power & Flow

Project leaders
JA van Oijen

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
AJ Kalbhor

Cooperation
Rolls-Royce Deutschland
Barcelona Supercomputing Center
Technische Universität Berlin
Universitat Politècnica de València
Technische Universität Darmstadt
Karlsruher Institut für Technologie
Universität Stuttgart

Funded by
EU

Funded %
EU 100%

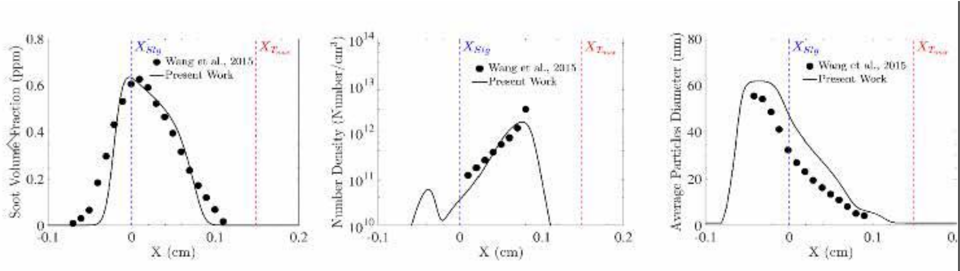
Start of the project
2018

Information
AJ (Abhijit) Kalbhor
062 218 0994
a.j.kalbhor@tue.nl

Project title
ESTiMatE: Modeling soot emissions of aero engines

Project aim
The Project aims to develop an advanced sectional method-based computational model for the prediction of soot under conditions relevant to aero engines. The focus lies on the development of an effective soot modeling approach by combining a discrete sectional method with Flamelet-Generated Manifold (FGM) tabulated chemistry for application in Large Eddy Simulation (LES).

Progress
A sectional method-based soot model coupled with detailed gas-phase chemistry was implemented in the one-dimensional flame code CHEM1D and validated with experimental measurements in non-premixed as well as premixed flames. The predictive capabilities of the model were found to be sensitive to gas-phase chemistry and the treatment of soot surface radicals. The model also showed good qualitative soot predictions for kerosene and JP-8 surrogate fuels. Moreover, the effects of both simultaneous and separate addition of H2 to the fuel and H2O to the oxidizer on soot formation in laminar, counterflow ethylene diffusion flames were studied. The detailed kinetic pathway analysis showed that the chemically inhibiting effects of H2/ H2O addition on soot formation are primarily through the reduced soot surface growth rates while the contribution of soot nucleation in the overall soot formation suppression is secondary.



Simulated soot characteristics profiles of the baseline counterflow diffusion flame studied by Wang et al. 2015; Measured (symbols) and computed (solid line).

TU/e - Mechanical Engineering - Power & Flow

Project leaders
S Nijdam (EPG), JA van Oijen (P&F), NJ Dam (P&F), J van Dijk (EPG)

Research theme
Complex dynamics of fluids

Participants
T Hazenberg, R Patel

Cooperation
NWO, Bosch Thermotechniek, Bekaert Combustion Technology, Micro Turbine Technology, Plasma Pendix

Funded by
NWO, Bosch Thermotechniek, Bekaert Combustion Technology Micro Turbine Technology

Funded %
NWO 85%
Industry 15%

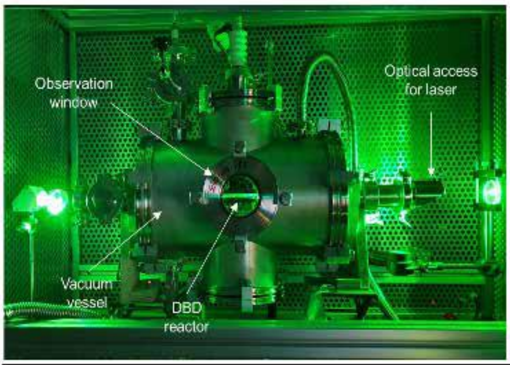
Start of the project
2019

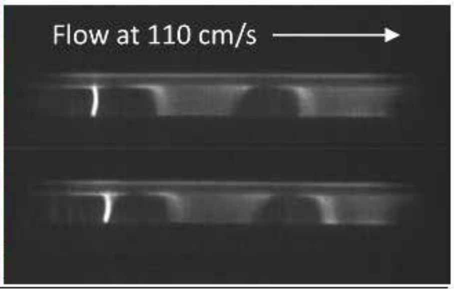
Information
JA (Jeroen) van Oijen
j.a.v.oijen@tue.nl

Project title
Making plasma-assisted combustion efficient

Project aim
Plasma-assisted combustion is a promising method to enhance flame stability in low-temperature flames. The reduction of flame temperature is important for reducing NOx emissions. However, control over the combustion process via the creation of radicals using plasma is not well understood. The goal is to gain an understanding of the plasma activation of combustion via both numerical and experimental works. The hope is that optimal plasma type and plasma parameters can be found which achieve maximum stabilization of the combustion processes in low-temperature regime.

Progress
A new experimental setup aiming at low-temperature combustion activation studies for various plasma parameters has been designed. During initial experiments, methane ignition in lean conditions is observed in streamer like plasma mode. ICCD imaging is used for plasma and ignition characterization. Along with new setup characterization, Raman spectroscopy for temperature and major species measurements is being developed for non-equilibrium conditions and results are expected soon. A numerical study of thermoacoustic mitigation using plasma has been initiated. Concurrently, an air plasma chemistry mechanism is being developed and simulated in 0D. Initial results for thermoacoustic mitigation and plasma chemistry simulations are expected soon.





Left: photograph of new experimental setup for low-temperature plasma assisted combustion studies. Right: Single pulse ICCD images capturing streamers assisted methane ignition

TU/e - Mechanical Engineering - Power & Flow

Project leaders
JA van Oijen

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
DV Efimov

Cooperation
Bosch Thermotechnology Ltd.

Funded by
Bosch Thermotechnology Ltd.

Funded %
Industry 100%

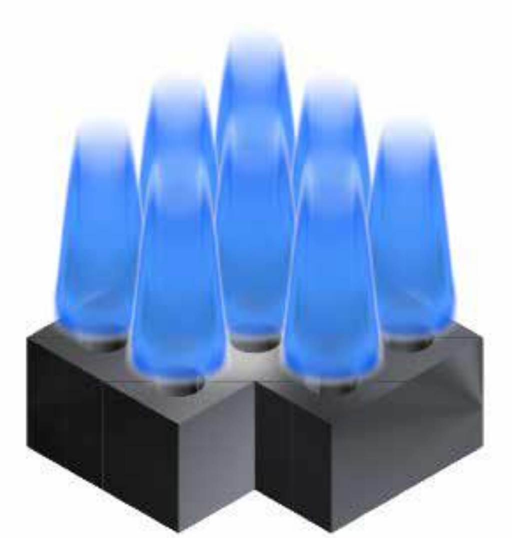
Start of the project
2019

Information
JA (Jeroen) van Oijen
j.a.v.oijen@tue.nl

Project title
Bosch hydrogen burner simulations

Project aim
In order to reduce greenhouse gas emissions, domestic heating has to almost completely be decarbonized. Hydrogen produces no CO2 at the point of use. However, there are currently no suitable products available to replace natural gas boilers. Furthermore, there are differences between methane and hydrogen combustion. The Project aims to designs a concept for future hydrogen boiler technology, by developing simulation methods. Simulation offers two benefits to such a project; in the early stages, fast coarse optimization and evaluation of novel concepts and later on, detailed optimization and sensitivity analysis to mature a chosen concept.

Progress
A method has been developed for practical and realistic simulations of hydrogen combustion. Due to challenges involved with the simulation of hydrogen flames using FGM simplified chemistry, several original methodologies are developed and implemented. Main progress was made on a novel treatment of the preferential diffusion of the FGM control variables, being very important for hydrogen combustion due to high diffusivity of hydrogen. Further the model performance was investigated including the effects of the burner conjugate heat transfer. Also, a robust method for lookup in multidimensional (3D) curved manifolds was developed.



The simulation result for hydrogen flames on a perforated burner plate

TU/e - Mechanical Engineering - Power & Flow

Project leaders
LMT Somers, NG Deen

Research theme
Complex dynamics of fluids

Participants
J Han

Funded by
Chinese government scholarship

Funded %
Scholarships 100%

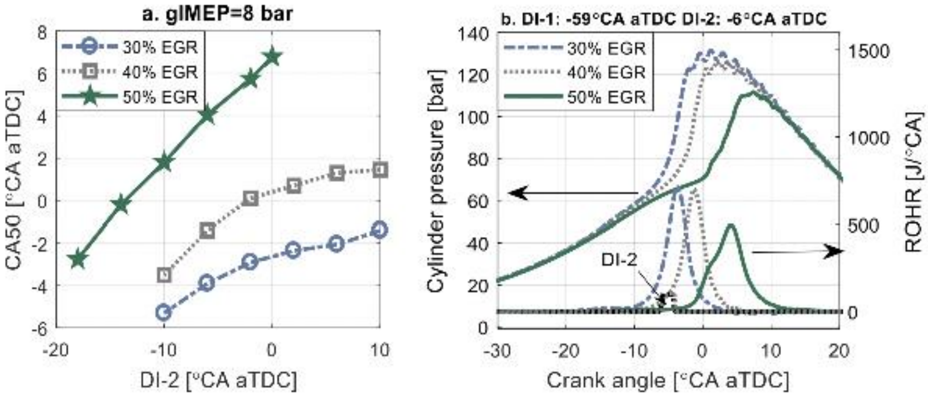
Start of the project
2019

Information
J (Jinlin) Han
J.Han.1@tue.nl

Project title
Alcohol fuel applications in HD engine

Project aim
Heavy-duty and long-haul transportation industry is confronted with challenges on air quality (pollutant emissions) and global warming (greenhouse emissions). One of the solutions is to combine advanced combustion concepts with alternative fuels to achieve the goal of a clean and efficient engine. This project mainly focuses on the possibility of applying alcohol fuels (ethanol and butanol) in low-temperature combustion concepts. Experimental investigations are performed on a heavy-duty engine setup to compare the combustion and emission characteristics of alcohol fuels when operated in different combustion concepts.

Progress
Applying butanol as a low reactive fuel in the so-called reactivity-controlled compression ignition combustion (RCCI) approach is investigated. The effects of charge preparation related parameters on RCCI are explored. Tests are performed at various inlet pressure, EGR rate, and DI strategy. The results indicate that for n-butanol/n-heptane RCCI necessitates a double DI strategy and a high EGR rate in this experimental setup. Above 50% gross indicated efficiency and Euro VI-compliant soot/NOx emissions can be achieved from 4 bar to 10 bar at the cost of deteriorated combustion efficiency. However, the high load limitation is constrained by the excessively high PRRmax and cylinder pressure. Among the investigated butanol isomers, isobutanol RCCI performs noticeably well in fulfilling the emission and safety standard with the most practical handling approach and the easiest control requirement. Though being solid at room temperature, tert-butanol RCCI shows excellent performance in soot reduction, has good combustion stability (low COVgIMEP and PRRmax) and a high GIE.



CA50 (a) and ROHR (b) vs DI-2 at a different EGR rate.

TU/e - Mechanical Engineering - Microsystems

Prof.dr.ir. JMJ den Toonder

The Microsystems group develops microsystems design approaches and out-of-cleanroom micro-manufacturing technologies that are rapid and flexible. these are applied to realize active mechanical control in micro-fluidics, to make and study meso-structured and soft materials, to create and study cells and organs on chips, and to develop advanced microsystems applications in collaboration with industrial partners. The group has a state-of-the-art microfabrication lab, and access to the Nanolab@TU/e cleanroom.

The Microsystems group is part of:

- MaTe, the Materials Technology Institute
- ICMS, the Institute for Complex Molecular Systems
- J.M.Burgerscentrum
- EM, Graduate School on Engineering Mechanics
- Human Organ and Disease Model Technologies

TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder, C Sahlgren,

Research theme
Complex structures of fluids

Participants
Jelle Jan Freerk Sleeboom

Funded by
EC 100%

Funding %
EU 100%

Start of the project
2016

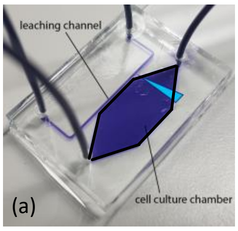
Information
Jelle Jan Freerk Sleeboom
+31 628 597 773
j.j.f.sleeboom@tue.nl

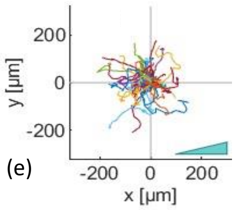
Project title
Microfluidic breast cancer model

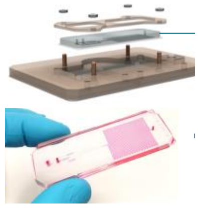
Project aim
The aim of the project is to develop chip-based models of breast-cancer prior to invasion to investigate properties of the microenvironment on the invasive behaviour of the cancer cells. By employing microfluidic techniques, physiologically relevant heterogeneous tissue models are constructed, with controlled and reproducible physical and chemical properties. The focus is on investigating the mechanical and chemical microenvironmental cues.

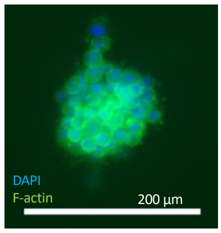
Progress
In 2020, this project was finalized with the PhD thesis of Jelle Sleeboom. The thesis reports the cancer-on-chip models we developed, based on microfluidic chips and designed to investigate two specific factors from the TME: Local oxygen gradients, and the ECM.

2020 Dissertations
Sleeboom, J.J.F. (2020). Microfluidic models of metastasis: In vitro approaches to study the tumor microenvironment. PhD thesis, Technische Universiteit Eindhoven.









TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder

Research theme
Complex structures of fluids

Participants
Andreas Pollet

Cooperation
Dr. Massimo Mischi, dr. Pieter Harpe

Funded by
STW 100%

Funding %
NWO 100%

Start of the project
2017

Information
Andreas Pollet
+31 40 247 2186
a.m.a.o.pollet@tue.nl

Project title
Locate: Integrated platform to design novel cancer localization strategies by ultrasound microvasculature imaging

Project aim
Locate aims at opening new avenues towards a cost-effective imaging solution for the localization of prostate cancer and other neo-angiogenic forms of cancer using dynamic contrast-enhanced ultrasonography (CEUS). It aims to extend fundamental knowledge through the development and application of a novel integrated validation and development platform, as well as proof-of-concept implementations. The platform consists of an experimental model of the microvasculature, tailored US imaging hardware, and dedicated US signal analysis modalities.

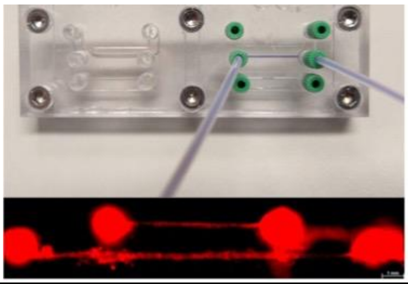
Progress
We further developed our sugar printing method to create vasculature-on-chip. An important step in 2020 was to integrate endothelial cells into the round micro-channels embedded in a hydrogel matrix, mimicking relevant biological aspects of the vasculature in the device. The blood vessels we created on the chip could be fully.

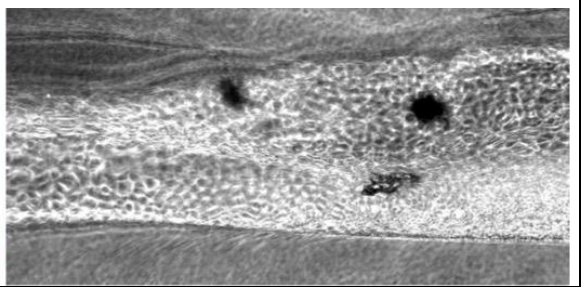
2020 Publications
Pollet, A. M. A. O., & den Toonder, J. M. J. (2020). Recapitulating the Vasculature using organ-on-chip technology. Bioengineering, 7(1), [7010017]. https://doi.org/10.3390/bioengineering7010017

Pollet, A. M. A. O., Homburg, E. F. G. A., Cardinaels, R., & den Toonder, J. M. J. (2020). 3D sugar printing of networks mimicking the vasculature. Micromachines, 11(1), [43]. https://doi.org/10.3390/mi11010043

Chen, P., Turco, S., van Sloun, R. J. G., Pollet, A., den Toonder, J. M. J., Wijkstra, H., & Mischi, M. (2020). In-vitro and in-silico porous phantoms for investigating the relationship between microvascular architecture and ultrasound-contrast-agent kinetics. In Proceedings of the IEEE International Ultrasonics Symposium (IUS) Institute of Electrical and Electronics Engineers.

Pollet, A., Homburg, F. G. A. E., & den Toonder, J. M. J. (2020). Engineering the vasculature using 3D sugar printing. Poster session presented at Annual Meeting European Organ-on-Chip Society 2020, EUROoCS Conference 2020, Uppsala, Sweden.





TU/e - Mechanical Engineering - Microsystems

Project leaders
J.M.J. den Toonder

Research theme
Complex structures of fluids

Participants
Eriola Shanko

Cooperation
Philips Healthcare

Funded by
HTSM-TKI

Funding %
NWO 100%

Start of the project
2016

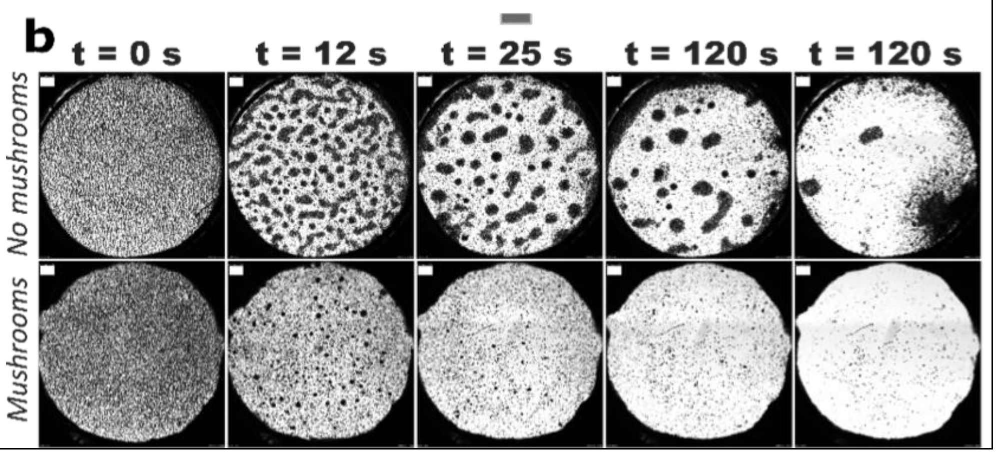
Information
Eriola Shanko
e.shanko@tue.nl

Project title
Magnetic control of micro-particles and integrated micro-actuators for high precision handheld diagnostics

Project aim
Magnetic particles actuated by magnetic fields are an attractive approach in the medical field, since they can be used in various applications, for example to capture and concentrate specific analyte, to mix fluids, and as labels for detection). In addition, to magnetic particles, also magnetic micro-actuators integrated into microfluidic devices can be used to control liquid flow and induce mixing, using externally generated magnetic fields. The aim of this project is to develop novel approaches on solving mixing problems to reach homogeneity of reagents and achieve high precision handheld diagnostics by using actuated magnetic particles and integrated magnetic micro-actuators.

Progress
In 2020 we developed a magnetic actuation method using an in-plane rotating magnetic field to induce magnetic bead mixing in a circular microfluidic chamber that allows better access with (optical) readout. We analyzed the magnetic bead dynamics, the induced fluid profiles and we quantified the mixing performance of the system. The rotating field causes the combination of (1) a global rotating flow counter to the external field rotation, with (2) local flow perturbations induced by rotating magnetic bead clusters in the central area of the chamber. This combination leads to efficient mixing performance within 2 minutes of actuated magnetic field.

2020 Publications
Shanko, E.S., Ceelen, L., Wang, Y., Burgt, Y. van de, & Toonder, J.M.J. den). Enhanced Microfluidic Sample Homogeneity and Improved Antibody-Based Assay Kinetics Due to Magnetic Mixing. ACS Sensors. Accepted for publication in 2020.



TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder

Research theme
Complex structures of fluids

Participants
Bhavana Venkataramanachar
Shuaizhong Zhang
Zhiwei Cui
Ye Wang
Roel Kooi
Tanveer Ul Islam
Tongsheng Wang

Cooperation
Prof. Patrick Onck (RUG)

Funded by
EU 100%

Funding %
EU 100%

Start of the project
2019

Information
Jaap den Toonder
j.m.j.d.toonder@tue.nl

Project title
Bio-Plan: Bio-Inspired Microfluidic Platform for Biomechanical Analysis

Project aim
We will realize a novel microfluidic platform for biomechanical analysis with unprecedented possibilities of controlling fluid flow and applying and sensing time-dependent forces at subcellular scales in controlled environments. The platform will be uniquely based on bio-inspired magnetic artificial cilia, rather than on conventional microfluidic valves and pumps. Cilia are microscopic hairs ubiquitously present in nature, acting both as actuators and sensors, essential for swimming of microorganisms, transport of dirt out of our airways, and sensing of sound, i.e. they exactly fulfill functions needed in biomechanical analysis.

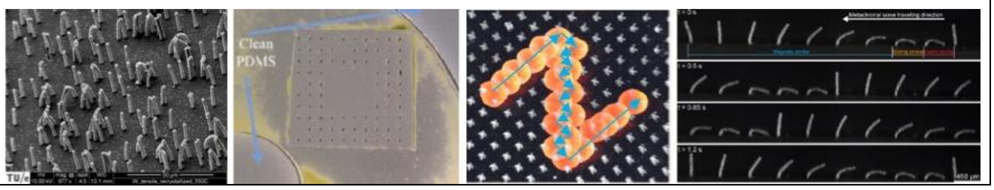
Progress
Various achievements were reached in 2020. First of all, we were for the first time able to fabricate nano-scale magnetic artificial cilia that are highly motile. Further, we have demonstrated that our micro-scale magnetic artificial cilia can be used to create anti-biofouling and self-cleaning surfaces, as well as to transport microparticles over a surface in a controlled manner. Finally, we have shown that metachronal actuation of microscopic magnetic cilia can generate strong microfluidic pumping.

2020 Publications
Zhang, S., Wang, Y., Onck, P.R. & Toonder, J.M.J. den (2020). A concise review of microfluidic particle manipulation methods. Microfluidics and Nanofluidics. 24, 24. DOI: https://doi.org/10.1007/s10404-020-2328-5

Zhang, S., Wang, Y., Onck, P.R. & Toonder, J.M.J. den (2020). Anti-Biofouling and Self-Cleaning Surfaces Featured with MagneticArtificial Cilia. ACS Appl. Mater. Interfaces. DOI: https://dx.doi.org/10.1021/acsami.0c05403

Zhang, S., Zhang, R., Wang, Y., Onck, P.R. & Toonder, J.M.J. den (2020). Controlled multidirectional particle transportation by magnetic artificial cilia. ACS Nano 14, 8, 10313–10323. DOI: https://dx.doi.org/10.1021/acsnano.0c03801

Zhang, S., Cui, Z., Wang, Y. & Toonder, J.M.J. den (2020). Metachronal actuation of microscopic magnetic artificial cilia generates strong microfluidic pumping. Lab Chip 20, 3569-3581. DOI: https://dx.doi.org/10.1039/D0LC00610F



TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder

Research theme
Complex structures of fluids

Participants
Inês Pereira
Roos van de Wijdeven
Hans Wyss

Cooperation
Prof. Henny Beckers (MUMC)
Prof. Albert Schenning (TU/e)

Funded by
InSciTe 100%

Funding %
GTI 100%

Start of the project
2019

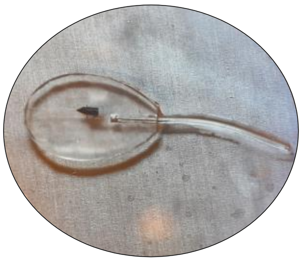
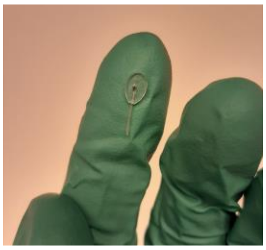
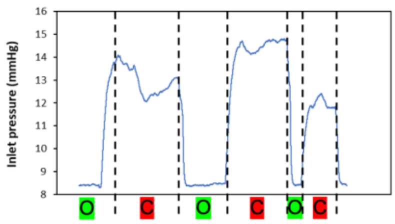
Information
Inês Pereira
i.c.figueiredo.pereira@tue.nl

Project title
Smart material and shunt design for flow control in a novel glaucoma implant

Project aim
Glaucoma is the leading cause of preventable blindness worldwide, estimated to cause bilateral blindness in 11 million people by 2020. A rise in the intraocular pressure (IOP) is considered to be the major risk factor for glaucoma. Glaucoma drainage devices (GDDs), which are typically hollow tube-like shunts surgically implanted in the eye, provide an alternative pathway through which AqH can effectively exit the anterior chamber, thereby lowering IOP. In this project, we aim at developing an innovative implant consisting of a novel glaucoma microshunt with an integrated responsive material. This will not only drive the AqH from the anterior chamber, but it will also enable to adjust the flow of AqH through the shunt non-invasively after surgery by the actuation of a responsive material.

Progress
We have designed, fabricated and tested a novel glaucoma implant. This device has an integrated magnetic valve that can be switched post-surgery. In microfluidic lab experiments, we have shown that the valve enables to change the hydrodynamic resistance of the device such that the IOP is changed by 6 mmHg. First animal trials show promising results towards clinical applications. In addition, we have conducted a literature review on glaucoma implants that has been accepted for publication.

2020 Publications
Pereira, I.C.F., Wijdeven, R. van de, Wyss, H.M., Beckers, H.J.M. & Toonder, J.M.J, den. Conventional glaucoma implants and the new MIGS devices: a comprehensive review of current options and future directions. Eye. Accepted for publication in 2020.

TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder

Research theme
Complex structures of fluids

Participants
Mohammad JouyBar

Cooperation
Moore4Medical partners

Funded by
H2020-ECSEL

Funding %
EU 100%

Start of the project
2019

Information
Mohammad JouyBar
m.jouy.bar@tue.nl

Project title
Breast cancer metastasis on chip

Project aim
Most breast cancer related deaths are not caused directly by the primary tumor, but by secondary tumors formed through metastasis to other organs. We aim to model breast cancer invasion on chip and the relevant microenvironment, in particular extracellular matrix (ECM) heterogeneity and vasculature. To this end, we will use our technologies to encapsulate cells in hydrogel and sugar printing to create microfluidic devices.

Progress
In 2020, we have made tailored devices to create heterogeneous breast cancer models and blood vessels on chip. These devices will be further developed and characterized in the coming period, and subsequently used to model metastasis.

TU/e - Mechanical Engineering - Microsystems

<u>Project leaders</u>
JMJ den Toonder

<u>Research theme</u>
Complex structures of fluids

<u>Participants</u>
Emma Moonen
Sevda Malek Kani

<u>Cooperation</u>
Dr. EduardPelssers (Philips Research)
Prof. Massimo Mischi (TU/e, EE)

<u>Funded by</u>
Penta
NWO-TTW

<u>Funding %</u>

STW	50%
EU	50%

<u>Start of the project</u>
2020

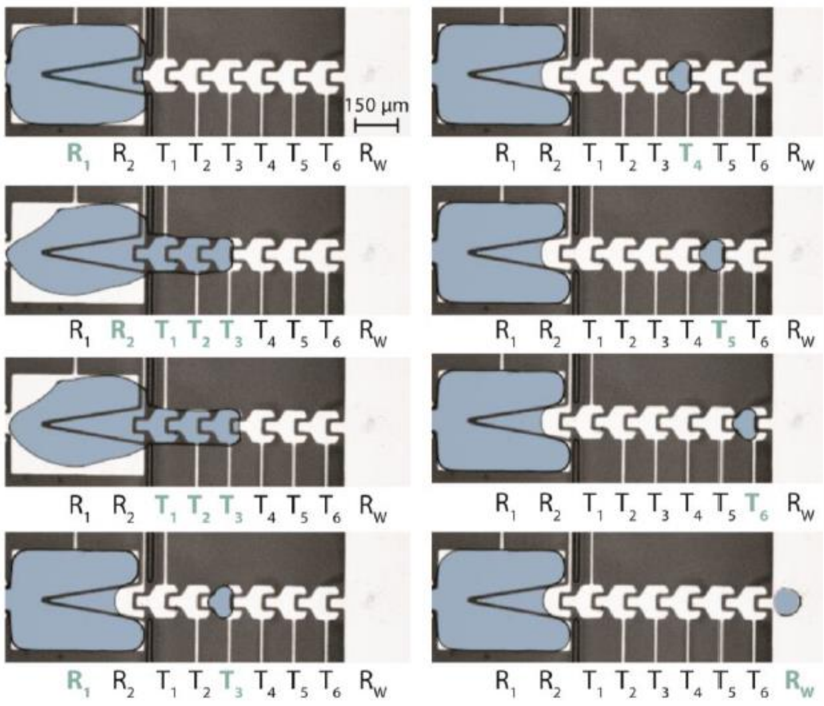
<u>Information</u>
Emma Moonen
e.j.m.moonen@tue.nl

<u>Project title</u>
Sweat sensing device for semi-continuous monitoring of hospitalized patients

<u>Project aim</u>
Sweat is a non-obtrusively accessible biofluid containing physiologically and metabolically rich information and could therefore be an ideal candidate for nonobtrusive health monitoring. Development of sweat sensing devices for monitoring individuals in sedentary state, such as hospitalised patients, is hampered by the low volumes of sweat produced by these persons (tenths of nanolitres per minute per sweat gland). In this project we will develop microfluidic principles to collect and transport such minute amounts of sweat in a wearable sweat patch, realize the devices, and show proof-of-principle.

<u>Progress</u>
We have carried out an extensive literature study which was published as a review paper. We have developed a new approach to transport minute amounts of sweat based discretised transport of droplets with volumes of around 0.1 nL. The discretised transport is based on electrowetting-on-dielectrics (EWOD) in which a voltage is applied to a dielectric stack to reduce the contact angle of a droplet on the surface. Droplets down to 80 picolitre could be pinched off successfully from the reservoir at 80 V AC at 1 kHz, and these droplets could subsequently be successfully transported.

<u>2020 Publications</u>
Moonen, E.J.M., Haakma, J.R., Peri, E., Pelssers, E., Mischi, M., Toonder, J.M.J. den (2020). Wearable sweat sensing for prolonged, semicontinuous, and nonobtrusive health monitoring. VIEW, 20200077. https://doi.org/10.1002/VIW.20200077



TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder
Ye Wang

Research theme
Complex structures of fluids

Participants
Yiqing Sun

Cooperation
EU project partners

Funded by
H2020-NMBP

Funding %
EU 100%

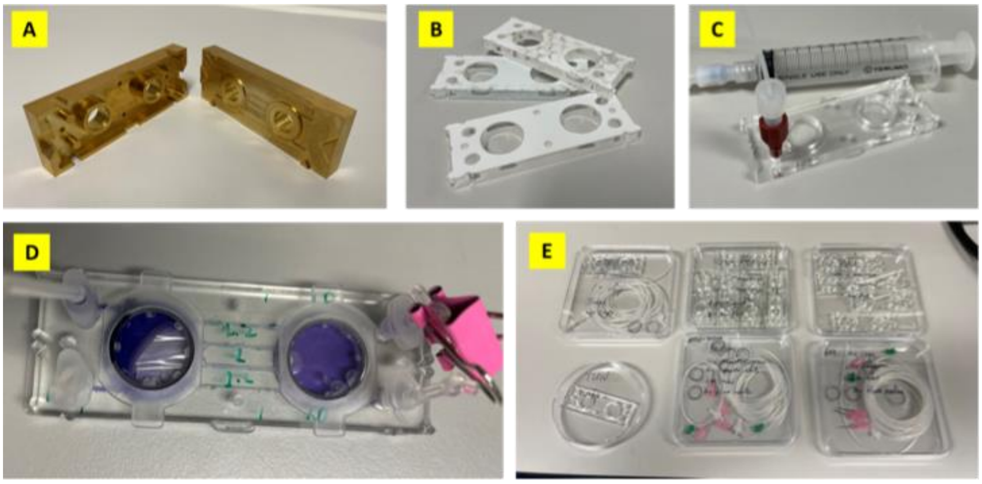
Start of the project
2020

Information
Ye Wang
y.wang2@tue.nl

Project title
Tumor-LN-oC: Tumor and Lymph Node on Chip for cancer studies

Project aim
The lymphatic system and lymph nodes (LNs) are an integral part of our adaptive immune system and many tumors exploit lymphatic vessels to spread and colonize downstream LNs. We aim to develop a microfluidic device as the central component of a robust, automated tumor-lymph node-on-chip platform that will allow us to study the interaction of primary tumors with lymph nodes, identify their chemical signature, enabling in the end personalized treatment relying on molecular characterization of lymph node metastasizing cells.

Progress
We have established requirements and specifications for the microfluidic chip, and designed and fabricated the first generation of microfluidic chips. Next is to test the chips for microfluidic performance and cell culture.



TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder
Ye Wang

Research theme
Complex structures of fluids

Participants
Sofia Gomez

Cooperation
LUMC, UT

Funded by
TU/e

Funding %
University 100%

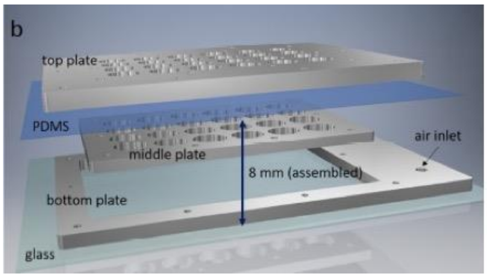
Start of the project
2020

Information
Ye Wang
y.wang2@tue.nl

Project title
Development of a pneumatically actuated heart-on-a-chip platform to promote cardiomyocyte maturation

Project aim
Heart on a Chip (HoC) platforms currently are developed for drug discovery and disease modeling to help reduce the cost and advance research. HoC aims to mimic the microenvironment of cardiomyocytes (CM), but questions remain in the maturation of CM. Different methods of maturation have been exploited, here we further explore mechanical cues. Mechanical stimulation has shown to enhance maturation of cardiac in vitro model. To promote this, we aim to develop a pneumatically actuating chip that can support a 3D culture and provide cyclic strain. By further tunning the 3D extracellular matrix, we can also incorporate stiffness cues in our model.

Progress
We have designed and fabricated the first generation HoC platform and tested its mechanical and fluidic performance. Next is to optimize the platform and start cell experiments.



TU/e - Mechanical Engineering - Microsystems

Project leaders
JMJ den Toonder

Research theme
Complex structures of fluids

Participants
Pan Zuo
Jelle Sleeboom

Cooperation
NWO Groot project partners

Funded by
NWO

Funding %
NWO 100%

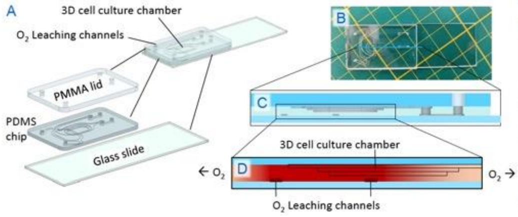
Start of the project
2020

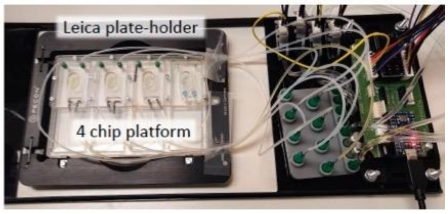
Information
Pan Zuo
p.zuo@tue.nl

Project title
The Active Matter Physics of Collective Metastasis

Project aim
Cancer metastasis is a complex process involving many steps, taken by cancer cells individually or collectively: invasion, migration, intravasation into blood vessels, circulation, extravasation out of the blood vessels. We will develop and apply microfluidic chips that enable to study in vitro (i) the invasion of tumor cell clusters in controlled and tunable microenvironments and (ii) the intravasation of tumor cells into blood vessels using high-resolution imaging.

Progress
We have designed and fabricated a microfluidic chip that enables to apply a controlled oxygen gradient in a 3D cell culture environment. This will, next, enable us to study, in a controlled manner, the influence of oxygen gradient on 3D migration of cancer cells and clusters.





TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Prof.dr.ir. JAM Kuipers

The research group Multiphase Reactors - Multi-scale Modeling of Multiphase Flows group participates amongst others in the 'OnderzoekSchool ProcesTechnologie' OSPT and the J.M. Burgerscentrum for fluid mechanics (JMBC) and focuses on fundamentals of the discipline of chemical reaction engineering. Our main area of interest is the quantitative description of transport phenomena (including fluid flow) and the interplay with chemical transformations in multiphase chemical reactors and in porous media.

Through the intended co-operation with other (application oriented) research groups, both fundamental aspects and those closely related to applications will be studied through concerted action. The main research topics of the new group SMR can be divided into the following two areas: Multiphase Reactor modeling and Advanced Experimental Techniques, which will be discussed below in more detail. An important area of attention is the development of advanced reactor models for multiphase reactors with industrial relevance.

At present our research focuses on the hydrodynamics in these reactors because it is generally recognized that the lack of understanding of the flow phenomena is one of the central difficulties in the design and scale-up of multiphase reactors. In addition, the interplay of flow phenomena with chemical reactions will be studied in great detail. We use various types of CFD models (both commercial codes but mostly "in house" made codes) to study the relevant hydrodynamic phenomena at all relevant length and time scales (i.e. at the microscopic, mesoscopic and macroscopic scale). In our group, both multifluid models and models which treat the dispersed phase (particles, bubbles or droplets) in a discrete manner accounting for possible encounters between the dispersed elements are being developed.

The second important area of our research deals with the development/ application of advanced experimental techniques to measure key quantities (i.e. local volume fractions and velocities of the dispersed and continuous phase). As an example we can mention the development and application of various optical techniques such as digital particle image velocimetry technique to measure in a non-intrusive manner the velocity map of both the liquid phase and dispersed gas bubbles in (dense) gas-liquid dispersions. This type of flow very often arises in a variety of gas-liquid contactors/reactors. In this area we co-operate with specialists within the J.M. Burgerscentrum for fluid mechanics.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M. Hadian, K.A. Buist

Funded by
ARC-CBBC

Funded %
Industry 100 %

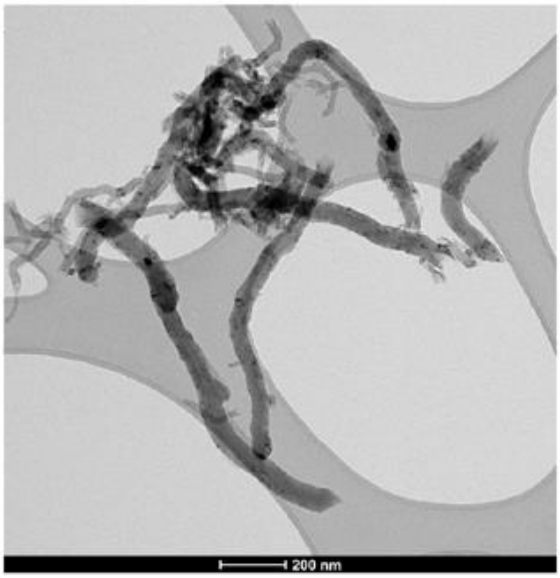
Start of the project
2019

Information
K.A. Buist
K.A.Buist@tue.nl

Project title
Small molecule activation: Pyrolytic upgrading of methane to ethylene, aromatics and carbon materials - Proof of concept and modelling of a gas-fluidized bed reactor

Project aim
Thermocatalytic decomposition of methane (TCD) next to development of proper catalysts, requires optimal design of the reactor and the process. To facilitate this, the knowledge should be gained by doing experiments and also developing models at different scales from a single particle to the reactor. This can be done only by taking smaller steps in both experimental and modelling aspects. Finally, it will be possible to conclude if TCD is a competitive technology to the conventional technologies or not.

Progress
The first step of the modelling part has been done in 2020 and a single particle model for TCD was developed and a scientific article was accepted to be published. On the other hand, it was started to perform kinetic experiments, on a commercially available catalyst from BASF. The aim of this experiments is to proof of CNF production in a fluidized bed, collect data of the performance of the reactor and derive the kinetics of the reaction. The type and purity of produced CNF is studied with TEM, TGA, and XPS. The figure shows a TEM of produced carbon.



TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
P.M. Durubal, M.W. Baltussen, K.A. Buist

Funded by
MCEC - Netherlands Center for Multiscale Catalytic Energy Conversion, European Union’s Horizon 2020 research and innovation programme under Marie Skłodowska-Curie grant agreement No 801359

Funded %
EU 100%

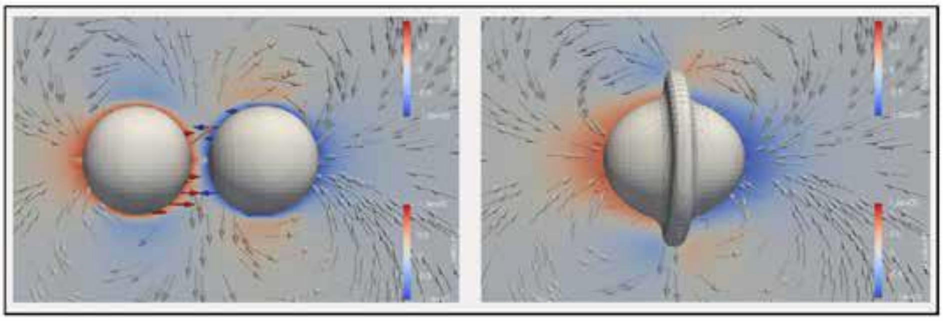
Start of the project
2019

Information
P.M. Durubal
p.m.durubal@tue.nl

Project title
Droplet and catalyst interactions in atomization of bio-oils

Project aim
The purpose is to develop a multiscale modeling approach for spray atomization of bio-oils by developing a Direct Numerical Simulation model which investigates interactions between droplets and droplets-particles at the mesoscale in a multi-component and multi-phase flow. In order to do so, a combined computational and experimental method will be employed. The study will enable a better understanding of the interactions of multicomponent droplets and the interactions of liquid droplets with (hot) catalyst.

Progress
The precision of newly implemented curvature and surface tension methods developed within the group have been tested by running a number of static bubble tests with VoF interface tracking method. After that, the binary droplet tests have been run. This test allows conducting simulations in spherical bubbles with different viscosities. Different input parameters of diameter, Weber number, impact parameter, and velocity have been used within VoF context. The tests have been performed by applying balanced force to calculate surface tension force while generalized height function and hybrid convolution to calculate curvature.



Binary droplet collisions of saccharose 40%-air for 1st step and 1000th step.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows	
<u>Project leaders</u> J.A.M. Kuipers, M. van Sint Annaland, N.G. Deen <u>Research theme</u> Complex dynamics of fluids <u>Participants</u> E. Milacic, D.M. Balice <u>Funded by</u> Dutch Polymer Institute <u>Funded %</u> Industry 100 % <u>Start of the project</u> 2017 <u>Information</u> E. Milacic e.milacic@tue.nl	<u>Project title</u> Heat management in polymerisation reactors <u>Project aim</u> Development of a set of numerical models capable of capturing the interaction between the gas phase, solid phase and liquid droplets as it occurs in gas-solid polymerization reactors. Use the developed models to get insight in the hold-up profiles of gas, solids and liquid and heat management in the reactors. Prediction and prevention of degenerate aspects. <u>Progress</u> During the course of the project, the degenerative effects of liquid injection in fluidized beds was studied. From the small-scale droplet-particle interactions, to the larger scale formation of agglomerates and complete defluidization. Numerical tools, as well as experimental methods were employed to gain a better understanding of the thermal effects of liquid injection into the fluidized bed. <u>2020 Publications</u> DPI, P.O. Box 902, 5600 AX Eindhoven, the Netherlands. The work of E. Milacic forms part of the research programme of DPI, project #803.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
JAM Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
D.R. Rieder, E.A.J.F. Peters

Cooperation
BASF, Shell

Funded by
Advanced Research Center – Chemical Building Blocks Consortium

Funded %
NWO 100%

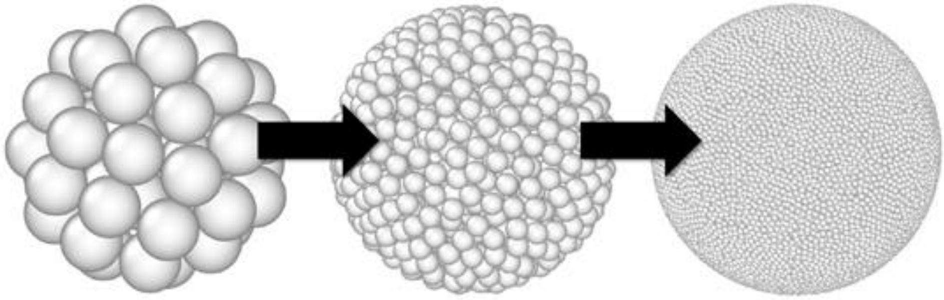
Start of the project
2019

Information
E.A.J.F. Peters
E.A.J.F.Peters@tue.nl

Project title
Modelling the preparation of supported catalysts

Project aim
Modelling the preparation process of supported catalysts including diffusion, imbibition and adsorption. By using a multiscale approach (subsequently deriving models for different length scales) the wide range of relevant dimensions, spanning several orders of magnitude will be covered. The final goal is the derivation of a general model that allows the prediction of the distribution of active material in a supported catalyst. The results will be validated with a model Ni-Pd-Silica system.

Progress
In the in-house code, the immersed boundary method was developed further towards conjugate transport across interfaces of immersed objects. Additionally, a two-step projection method was implemented and verified to solve the incompressible Navier-Stokes equations. Further, a range of initial model geometries for investigating transport in porous media were defined, as shown below. In parallel, the governing equations required for modelling the transport of ionic species were identified and several implementations tested on their stability and accuracy.



TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M Kuipers, E.A.J.F. Peters

Research theme
Complex dynamics of fluids

Participants
A. Fathiganjehlou, K.A. Buist

Funded by
NWO

Funded %
NWO 100%

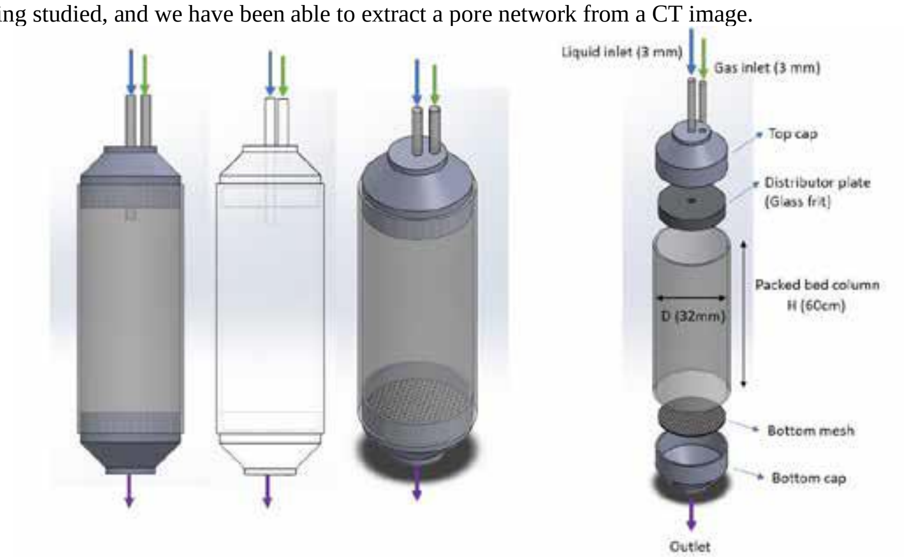
Start of the project
2020

Information
K.A. Buist
k.a.buist@tue.nl

Project title
MRI flow imaging of trickle bed reactors (TBRs)

Project aim
Imaging of the flow inside TBRs has drawn a lot of attention in the past decades. Detailed understanding of the flow hydrodynamics can pave the way for enhancements in the design and performance of TBRs. One of the methods for imaging the flow inside chemical systems is MRI. MRI is a safe technique and is capable of non-invasively imaging 3D opaque systems. Also, extending the capabilities of MRI with hyperpolarization equipment allows the imaging of the gas phase. The aim of this project is to experimentally analyze the multi-phase hydrodynamics inside TBRs using MRI and compare the results with CFD and Pore Network Modelling.

Progress
As preparation for the project, the course "Magnetic Resonance Advanced Physics" at Utrecht University has been followed. The course helped to understand the basics of MRI and how to handle an MRI machine to obtain hydrodynamic data from a fluidic system such as packed beds. The next part of the project is to design and fabricate a trickle bed system that is suitable to be imaged using the MRI machine in our lab. For this purpose, several articles have been reviewed, and a suitable design was proposed (see the figure). Meanwhile, the Pore Network Modelling of packed beds is being studied, and we have been able to extract a pore network from a CT image.



Different schematics and exploded view of the designed trickle bed column (drawn with SolidWorks™)

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers, E.A.J.F. Peters

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
C.M.Y. Claassen, A. Eghbalmanesh, A. Fathiganjehlou, M.W. Baltussen

Funded by
NWO

Funded %
NWO 100%

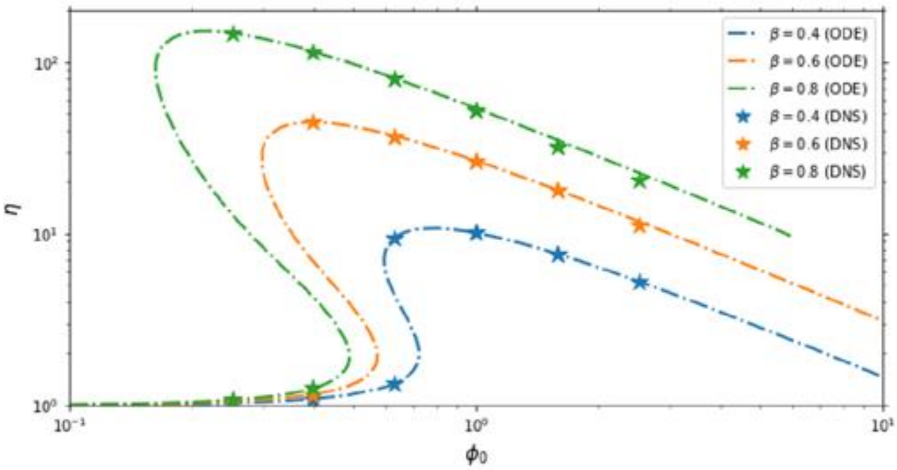
Start of the project
2020

Information
M.W. Baltussen
m.w.baltussen@tue.nl

Project title
The inside out of catalytic particles: A numerical study on the multiscale transport phenomena and reaction kinetics inside trickle bed reactors

Project aim
The project focusses on modeling the transport and kinetics occurring within the catalytic pellets of a trickle bed. The challenges to tackle are to have a proper modeling of the transport inside the particle and to have realistic reaction kinetics. The particle model will be coupled with the outside flow dynamics model from another PhD (A. Eghbalmanesh). The coupled model will be validated with MRI experiments (performed by A. Fathiganjehlou) for the hydrogenation of α-methylstyrene to cumene.

Progress
A general method for including reactions of various forms in the mass and heat balances is implemented. Next to that, the initial step for coupling inside the particle and outside the particle is realized with IBM assuming a constant bulk concentration/temperature. These combined steps are verified with the Weisz and Hicks case, non-isothermal first order reaction and diffusion inside a sphere, as shown in the figure. This case assumes Fickian diffusion inside the particle, but the more accurate Maxwell-Stefan diffusion is also already realized by S. Tadayon Mousavi.



Validation of the DNS method with the Weisz and Hicks case

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
S. Safavi Nic, J.A.M. Kuipers, K.A. Buist

Cooperation
Danone Nutricia Research
Funded by
Danone Nutricia Research

Funded %
Industry 100%

Start of the project
2018

Information
Sherwin Safavi Nic
06 25033596
s.safavi@tue.nl

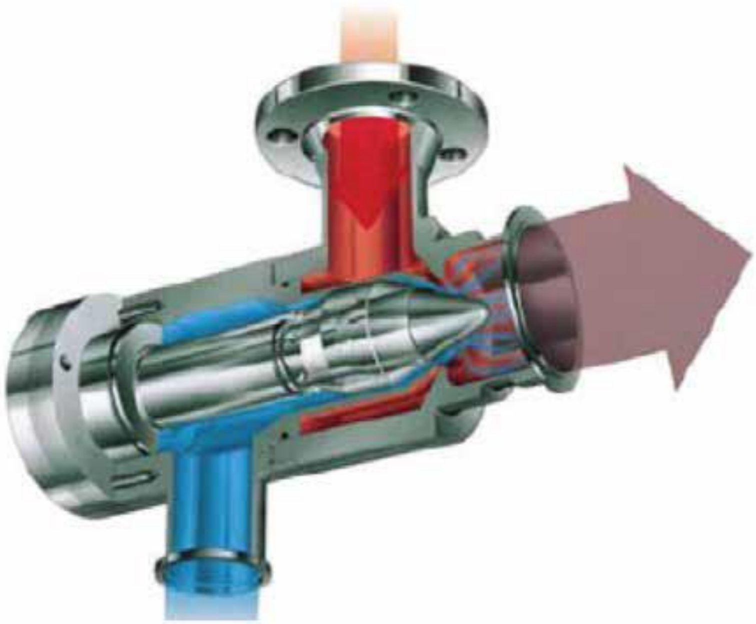
Project title
Direct steam condensation for dairy food sterilization

Project aim
Direct steam condensation is applied for the sterilization of infant milk to ensure a high level of food safety. The high temperature difference between the dairy based product and steam is often to such an extent that the product is damaged and process efficiency is lowered if the mixing and condensation does not occur in an efficient manner. The aim of this project is to develop a predictive model for the heat and mass transfer in order to optimize the design of the unit operation.

Progress
The experimental facility was completed and measurements were carried out. Model predictions were compared with the lab data. The deposit layer formation model is in agreement with the trend observed in the experiments, but is not fully capturing the process dynamics.

2020 Publications

S.S. Safavi Nic, K. A. Buist, R.E.M. Verdurmen J.A.M. Kuipers; Dynamic model to predict heat induced protein denaturation and fouling in a Direct Steam Injection process, Chemical Engineering Science X, vol. 8, 100075, 2020.



Sketch of internal flows of a DSC unit

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

<u>Project leaders</u>
J.A.M. Kuipers

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
M.W. Baltussen, D.E.A. van den Eertwegh

<u>Cooperation</u>
Eindhoven University of Technology, TU Delft, University of Twente, Shell, Nobian

<u>Funded by</u>
NWO, Shell, Nobian

<u>Funded %</u>
NWO 75%
Industry 25%

<u>Start of the project</u>
2020

<u>Information</u>
M.W. Baltussen
m.w.baltussen@tue.nl

<u>Project title</u>
Direct Numerical Simulation of multiple bubble growth and dynamics

<u>Project aim</u>
The project focusses on the effect of multiple bubbles on the electrolysis of an alkaline water solution. The goal is to investigate mass transfer and reaction phenomena in the vicinity of an electrode, which are directly influenced by bubble growth, bubble detachment and bubble rise near the electrode surface. To this end, a numerical study, taking into account all aforementioned phenomena, is performed. The transport of the charged species is treated using the Poisson-NernstPlanck equations, while bubbles are included using a Front Tracking approach. The Navier-Stokes equations are solved on a structured grid using a Direct Numerical Simulation approach.

<u>Progress</u>
A one-dimensional implementation of the Poisson-Nernst-Planck equation system was made. In this implementation, the Finite Volume Method was used for the discretization of the equations. All equations are solved in a coupled fashion to prevent instabilities. After comparison with literature, it was concluded that the in-house implementation works as expected. Future efforts will involve extending the implementation to three dimensions. Apart from the charged species transport, a start has been made with adaptive mesh refinement, which should allow the mass transport boundary layers to be properly resolved.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Complex dynamics of fluids

Participants
V. V. Swami, K.A. Buist, M.W. Baltussen

Cooperation
ISPT

Funded by
TKI E&I, FrieslandCampina, Danone-Nutricia, DSM

Funded %
NWO 50%
Industry 50%

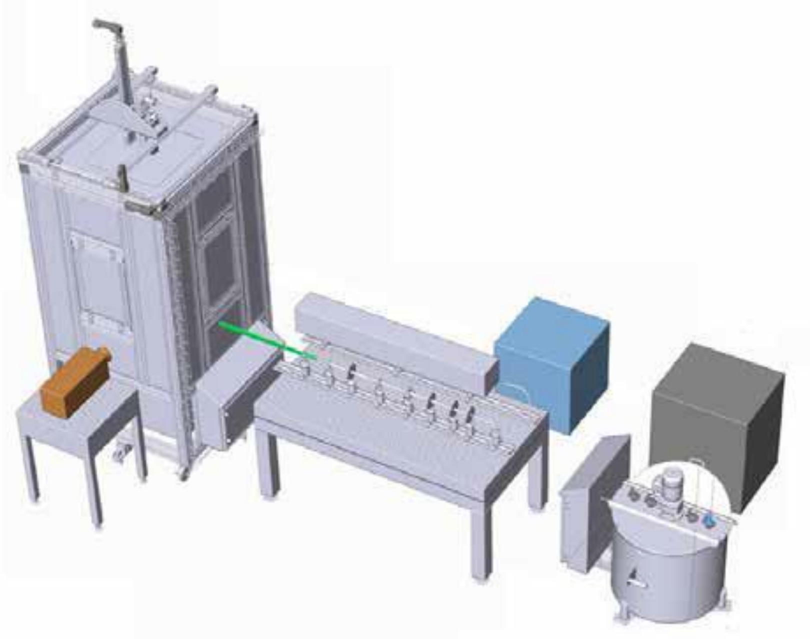
Start of the project
2019

Information
K.A. Buist
k.a.buist@tue.nl

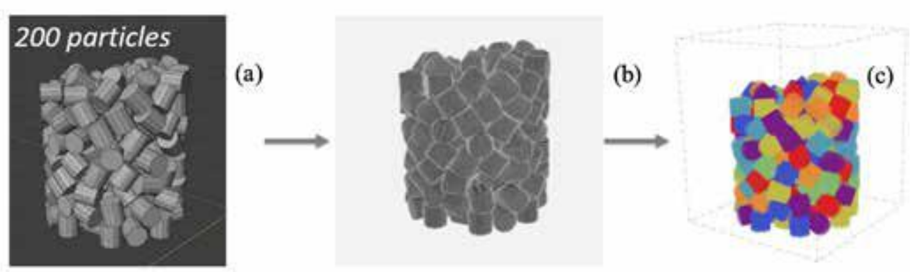
Project title
Experimental study on the atomization of complex fluids in spray dryers

Project aim
The atomization process in a spray dryer largely influences final particle size distribution and efficiency of the process. It is highly dependent on upstream fluid dynamics inside the nozzle and downstream interactions between discrete and continuous phases. However, the effect of liquids with high solids content on both fluid dynamics inside the atomizer and the atomization processes outside the nozzle is not very well understood. In addition, the high solid content in the liquid might result in non-Newtonian behaviour of the liquid. Therefore, the main goal of the project is to develop a fundamental understanding of the atomization of complex fluids, such as milk like fluids in spray dryers, using an experimental approach.

Progress
The state-of-the-art laboratory and the spray box have been developed to perform high-pressure spray characterization experiments for both Newtonian and non-Newtonian fluids using optical and laser diagnostics techniques.



3D model of spray characterization setup and the spray box

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows	
Project leaders J.A.M. Kuipers Research theme Complex dynamics of fluids Participants N. Romijn, M.W. Baltussen, K.A. Buist Cooperation Amirhossein Eghbalmanesh Funded by MCEC - Netherlands Center for Multiscale Catalytic Energy Conversion Funded % NWO 100% Start of the project 2019 Information K.A. Buist k.a.buist@tue.nl	Project title MRI flow imaging of trickle bed reactors Project aim Trickle bed reactors are high potential candidates for the production of renewable CO2 neutral fuels. Although trickle bed reactor have higher conversions due to a lower degree of back-mixing in comparison to slurry bubble column reactors, challenges arise in the scale-up and design of these systems. The main reason for these challenges is that the performance of these trickle bed reactors is closely related to phenomena that occur at particle scale. Understanding the hydrodynamics and heat and mass transfer in the particle packing is key. Hence, the aim of this project is to study and characterize the fluid flows in trickle bed reactors using Magnetic Resonance Imaging (MRI). Progress To study transport processes locally particle information, e.g. position and orientation, needs to be obtained. Simple reconstruction methods do, however, not exist for cylindrical particle packings. In this work a detection method for equilateral cylindrical particles is created, as shown in the figure.  *(a) A synthetic cylindrical particle packing generated via Blender software. (b) A 3-D volume image generated from a stack of cross-sectional images. (c) Volume points are clustered into the number of cylindrical particles present via a segmentation algorithm based on the k-means clustering function*

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers, E.A.J.F. Peters

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M.J.A. de Munck

Funded by
NWO and Industry

Funded %
NWO 100%

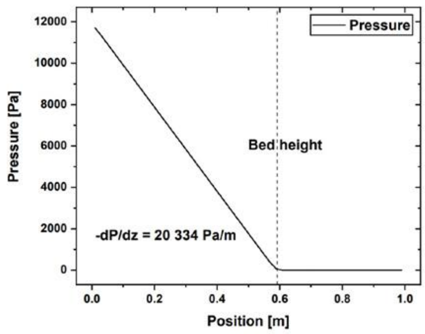
Start of the project
2020

Information
M.J.A. de Munck
m.j.a.d.munck@tue.nl

Project title
Coupled mass, momentum and heat transfer in dense particulate flows

Project aim
This project aims to properly model and validate the coupled mass, momentum and heat transfer in a drying process. A modeling/experimental CFD-DEM and PIV/DIA approach will be used to study systems including polydisperse and non-spherical particles. In a later stage, the CFD-DEM method will be extended with newly developed coarse graining methods in order to simulate an industrial-scale dryer.

Progress
The CFD-DEM code is currently fully verified for momentum, mass and heat transfer whereof one verification test is shown in the figure, where a comparison with the Ergun expression for a pressure drop over a packed bed is made. All verification tests were passed successfully. Therefore, the initial steps regarding drying of porous solid material are made.



Numerical results of the pressure drop over the packed bed for the test case described by Van Sint Annaland et al.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
T.M.J. Nijssen, K.A. Buist

Funded by
M2i, STW

Funded %
NWO 50%
Industry 50%

Start of the project
2017

Information
K.A. Buist
K.A.Buist@tue.nl

Project title
Modeling of dynamic solid particle bed and liquid behaviour – applied to the blast furnace hearth

Project aim
The bottom section of the blast furnace consists of a system of liquid iron and a packed bed of coke particles. The continuous addition of liquid iron and periodic tapping of the iron induces a cyclic movement of the solid bed, periodically packing and moving the coke particles. The state of this bed and the flows through it largely determine the erosion and lifetime of the refractory bricks in this section, however, the dynamics and interactions in this system are ill-understood. Therefore, a CFD-DEM model is used to study the effects of cyclic movement, particle size distribution and shape as well as particle dissolution rate on the dynamic solid particle bed and liquid behaviour.

Progress
Large-scale VOF/CFD-DEM simulations of the blast furnace hearth were performed. Using a 5 m diameter, full-3D geometry, the influence of burden weight, bi-disperse packing and blocked tuyeres on the liquid and solids flow within the hearth were investigated. Horizontal and vertical porosity profiles were analysed, and the influence of the dynamic liquid level on the state of the deadman was evaluated. The liquid iron flow during tapping was visualised, and the influence of a coke-free space on the flow pattern was analysed. The magnitude of the circumferential flow through the corner of the hearth was analysed, and found to decrease with increasing burden weight pressure and centre coke diameter. A significant influence of the dynamic deadman on the liquid flow pattern was found, especially in case of a floating deadman. In addition to liquid flow, the solid coke flow towards the raceways was analysed. Two pathways for coke particles towards the raceway were uncovered, one path from the actively flowing layer and one path from the raceway, coming from the deadman. The balance between these two mechanisms was found to change during the tapping cycle. Lastly, implementations for heat and dissolved carbon mass transfer were made, and demonstrated using a full-scale 10 m hearth simulation. Additional closures for heat and mass transfer rates are required, but the current model is found in good shape for future work.

2020 Publications
T.M.J. Nijssen, J.A.M. Kuipers, J. Van Der Stel, A.T. Adema, and K.A. Buist. “Complete liquid-solid momentum coupling for unresolved CFD-DEM simulations”. International Journal of Multiphase Flow, vol. 132, 2020 DOI:10.1016/j.ijmultiphaseflow.2020.103425.

Cross-sectional view of a 10 m diameter hearth simulation. Liquid iron is represented in yellow, the raceway zones in red. Blue and cyan represent the burden weight zone and particle insertion zone, respectively. Larger centre coke particles are highlighted.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
S. Tadayon Mousavi, E.A.J.F Peters

Funded by
MCEC (Multiscale Catalytic Energy Conversion consortium)

Funded %
NWO 100%

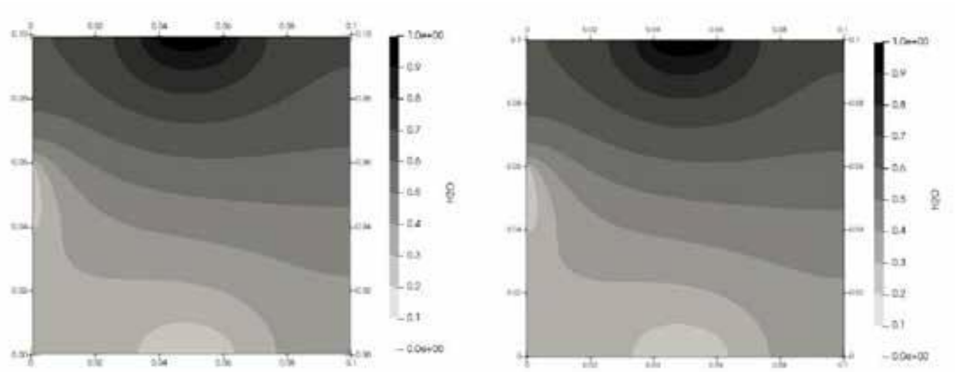
Start of the project
2020

Information
S. Tadayon Mousavi
s.tadayon.mousavi@tue.nl

Project title
Multi-scale modelling of the Fischer-Tropsch process

Project aim
The aim of the project is to model interplay of transport processes and the reaction kinetics of the Fischer-Tropsch process in a multi-tubular packed bed reactor. In such reactors, reactions occur on the internal surfaces of the pellets, where the catalytic sites are located. In order to predict the chemical conversion in catalytic pellets and subsequently the efficiency of the reactor, we will build a highly advanced computational model to describe transport phenomena within a porous pellet in full detail. Then, by coupling transport phenomena inside and outside of pellets, we aim to get insight in the scale of a packed bed reactor.

Progress
Stefan-Maxwell equations are chosen to describe the multi-component diffusion system in this project among other two candidates: Fick diffusion law and solving momentum equations for all N species in a mixture separately. The Stefan-Maxwell equations are computationally cheaper in comparison with solving momentum equations for each species, and their applicability is not limited to binary mixtures or mixtures with a dominant species like Fick diffusion model. A numerical algorithm to solve these equations is implemented in our in-house software and compared with literature.



Mass fraction of H2O in a diffusion simulation with Maxwell-Stefan for a three species H2O, H2 and N2 system where the boundary conditions are set in the middle of three faces as: west: H2O = 0, H2 = 1, N2 = 0; north: H2O = 1, H2 = 0, N2 = 0; south: H2O = 0, H2 = 0, N2 = 1. The left column is the result of implemented algorithm in our in-house software and the right one is presented in Peerenboom et al.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers, E.A.J.F. Peters, M.F. Neira d'Angelo

Research theme
Complex dynamics of fluids

Participants
A. Uslu

Cooperation
Sustainable Process Engineering group

Funded by
MCEC

Funded %
NWO 100%

Start of the project
2020

Information
M.F. Neira d'Angelo
m.f.neira.dangelo@tue.nl

Project title
Valorization of biomass derived molecules using open-cell foam catalysts

Project aim
The increasing scarcity of fossil fuels together with their effect on climate change has pushed both researchers and the chemical industry to look for alternative feedstocks for specialty chemicals. In this project the catalytic conversions of lignocellulosic biomass using structured open-cell foam catalysts. The aim of the project is to elucidate the transport phenomena and chemical behaviour governing the conversion of lignocellulosic biomass by taking advantage of the increased catalytic and hydrodynamic control provided by these foam catalysts.

Progress
Work has been performed on the preparation of stable Pd/Al2O3 coated foams. The concept method has been realized, the following step is the screening of the catalyst within the reaction involving the hydrogenation of furfural. The first shows SEM imagery of the coating. In parallel a reactor model has been developed in order to numerically represent the transport phenomena occurring within the reaction in order to differentiate between external mass transport, internal mass transport within the catalytic particle and fitting of reaction kinetics. A preview of the output given in the second figure.

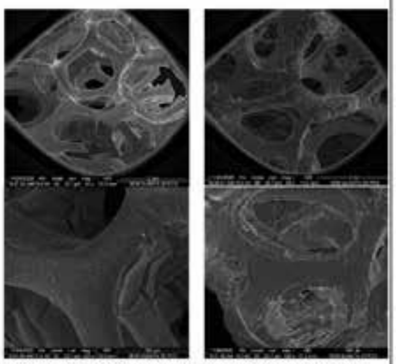


Figure 1 Pd/γ-Al2O3 coated on open-cell aluminum foams using boehmite as binder (left) and PVA (right)

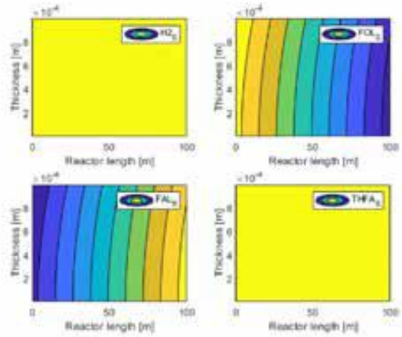


Figure 2 Concentration profiles determined by modelling within the particle

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
K.A. Buist, M.W. Baltussen, J.G. Ramirez

Funded by
NWO

Funded %
NWO 100%

Start of the project
2019

Information
K.A. Buist
K.A.Buist@tue.nl

Project title
Advanced 3D multiphase CFD method for bio-oil deoxygenation

Project aim
This project aims to develop a Multiphase Computational Fluid Dynamics (MCFD) model to study the deoxygenation of Bio-oils. The challenge of the project lies in the combined complexities of the hydrodynamic interactions between the three different phases; gas, liquid and solid, including turbulence, and the additional chemical conversion. The complex composition of the biooils add to the difficulty of both the chemical conversion and the hydrodynamic behaviour of the droplet phase.

Progress
A CFD-DEM model was developed to represent the riser system of this research. The model includes three different collision approaches with the characteristics presented in the table. Due to the presence of dilute and dense regions inside the riser, the best collision model is still an open question. Additionally, the CFD-DEM is also coupled with the Immersed Boundary Method (IBM). This allows for the representation of the elbow at the end of the riser. The figures show preliminary results for the expected Core–annulus radial profile of solid fraction and the comparison in the number of collisions predicted by the three approaches.

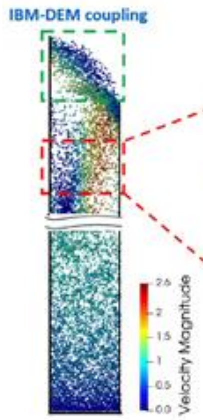
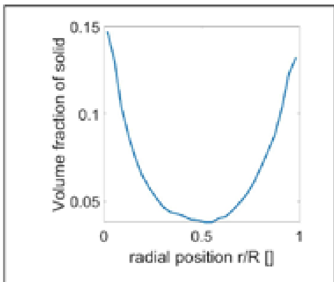
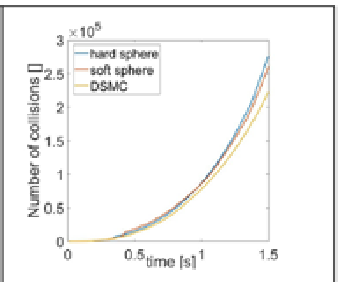


Table 1. Collision models considered in this research

Collision model	Approach	Best performance in
Hard sphere	Deterministic	Dilute systems
Soft sphere	Deterministic	Dense systems
DSMC	Probabilistic	Dilute systems



Core–annulus radial profile of volume fraction of solid.



Number of collision along the simulation for the different collision models.

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

<table>
<tr>
<td>

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M.W. Baltussen, K.A. Buist, A. Tavanaei

Funded by
Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC

Funded %
NWO 100%

Start of the project
2019

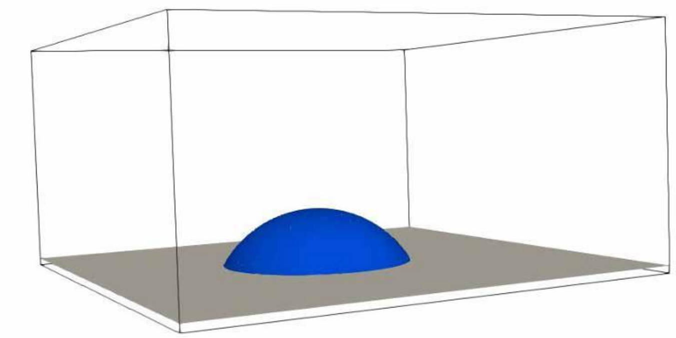
Information
M.W. Baltussen
M.W.Baltussen@tue.nl

</td>
<td>

Project title
Particle scale transport phenomena in the trickle bed reactors

Project aim
The aim of the project is to study the hydrodynamics, heat and mass transfer for the Fischer-Tropsch (FT) process with biomass feedstock in a Trickle Bed Reactor (TBR). Currently, Bubble Slurry Columns are being used to perform the FT process. The TBR has lower back mixing and hence higher conversion but the technology is not well developed mostly because of the complex particle-scale phenomena inside the reactor. The Liquid-Gas interactions will be simulated by using the Volume of Fluid technique while the Immersed Boundary Method will be used to simulate Fluid-Solid Interactions. All the simulations will be validated by MRI measurements in a parallel research project.

Progress
To improve the currently implemented Volume of Fluid method, the Balanced Force method is implemented to decrease the numerical error, which is caused by the mismatch of the discretization of surface tension force and the discretization of the pressure filed. In addition, two extra models for the calculation of the interface curvature have been added: the Standard Height Function and the Hybrid Convolution-Generalized Height Function (CVGHF) method. Meanwhile the Immersed Boundary Method (IBM) is implemented by other members of the SMR-group. Finally, to model three-phase interactions, a contact line model based on extrapolation of PLIC interface is implemented.



A droplet in equilibrium while 60-degree contact angle boundary condition is applied at solid boundary

</td>
</tr>
</table>

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
C. Garcia Llamas, M.W. Baltussen, K.A. Buist

Cooperation
TKI, FrieslandCampina, Danone, DSM, ISPT

Funded by
TKI

Funded %
Industry 100%

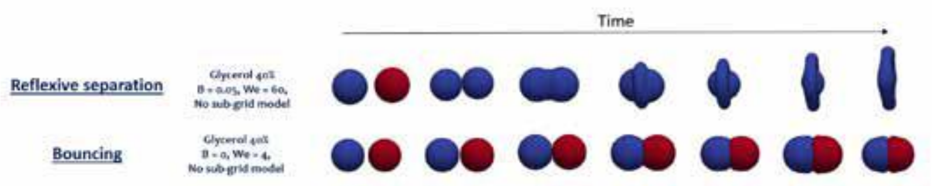
Start of the project
2019

Information
M. W. Baltussen
m.w.baltussen@tue.nl

Project title
Energy Efficient Milky Sprays (EEMS): Numerical study of the effect of solids content and main components of model liquids on atomization efficiency of the nozzle

Project aim
The spray drying efficiency is highly dependent on the atomization process, since it affects directly the droplet size distribution. In order to get more insights on the process a rheological analysis of the fluid dynamics inside a high-pressure nozzle and its subsequent break-up (i.e. primary atomization) is required. To facilitate this, Direct Numerical Simulation (DNS) of the sheet break-up is performed inside and outside the nozzle. The output of these simulations will be experimentally validated, within a parallel project, with the aid of different measuring techniques such as Phase Doppler Anemometry (PDA) or Structured Laser Illumination Planar Imaging (SLIPI).

Progress
Project is divided in two parts: First, high fidelity simulations are performed inside the nozzle in order to capture the interaction of the air core with the liquid sheets as well as its implications on the stability of the liquid sheet. Second, the simulation domain will be extended to the near region out of the nozzle in order to capture the rupture of the liquid sheet. The Local Front Reconstruction Method (LFRM) has been chosen to track the interface. This method is currently developed and being tested for its application to spray atomization (see the figure).



Collision evaluation of binary droplets using LFRM

TU/e - Chemical Engineering & Chemistry - Multi-Scale Modeling of Multiphase Flows

Project leaders
J.A.M. Kuipers, E.A.J.F. Peters

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M.W. Baltussen, A. Eghbalmanesh

Funded by
NWO

Funded %
NWO 100%

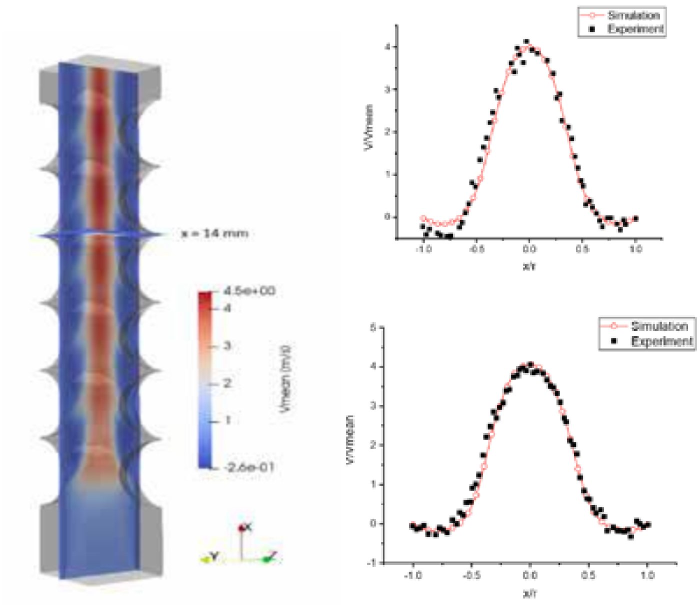
Start of the project
2019

Information
M. W. Baltussen
m.w.baltussen@tue.nl

Project title
Hydrodynamics of gas-liquid in trickle bed reactors

Project aim
Gas-liquid two-phase porous media flows are common in industrial processes as well as nature. An important industrial example is the trickle-bed, which is a column packed with catalytically active particles with the co-current downflow of gas and liquid. This project will focus on the hydrodynamics of the gas and liquid flow in such trickle-beds. Hydrodynamic parameters of trickle bed reactors are controlled by the internal bed structure and associated interactions with the gas and liquid flow. The main goal of this project is to relate the gas-liquid flow to the packing characteristics such as particle shape, size, and polydispersity of particle sizes.

Progress
In this project, multiphase Computational Fluid Dynamics will be used to study gas-liquid flow in a tube filled with particles. The cylindrical tube as well as the non-spherical particles are implemented by means of a ghost cell Immersed Boundary method. The first step of the project is simulating the single-phase flow in a spherical packed bed. The figures show the velocity contours and the comparison of the velocity profile of the simulation and experiment. The simulation of the laminar flow in the packed bed of a cylindrical packed bed is in progress for the next step of the project.



(Left) Geometry of the fluid flow in simple cubic lattice of spheres and position of the plane at x = 14:2 mm
(Right) Comparison of the normalized velocity in the normalized distance from the center of the channel between the fourth and fifth layers of spheres with (a) Re = 105.57 and (b) Re = 204.74 in the simulation of with the experimental1 results

TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification

Prof.dr.ir. M van Sint Annaland

The research group Chemical Process Intensification (SPI) is part of the Department of Chemical Engineering and Chemistry at the Eindhoven University of Technology. The main objective of the research group is the development of novel integrated reactor concepts based on improved fundamental knowledge using validated advanced (multi-phase) reactor models. This is achieved by employing a combination of state-of-the-art numerical models (at different levels of detail using the multi-level modeling approach), advanced (non-invasive) experimental techniques and experimental demonstration of novel reactor concepts (proof of concept).

A key competence of the group is the development and use of use of advanced (multi-phase) reactor models, coupled to mass and heat transfer and chemical reactions, in order to study integrated reactor concepts. Our modelling work ranges from 'as detailed as necessary' to 'as large-scale as possible'; we employ models for redox kinetics on a single particle scale and detailed simulations of gas-liquid flows, discrete bubble/particle models, up to industrial scale phenomenological models and process systems modelling. We use both in-house created models and open-source based models. A cornerstone of our research is careful (experimental) validation and verification.

Another aspect is the development of advanced, non-intrusive, experimental techniques to measure key quantities (i.e. local volume fractions and velocities of the dispersed and continuous phase). We have developed a PIV-DIA technique (particle image velocimetry coupled to digital image analysis) that measures accurately, in a non-intrusive manner, the solids fluxes in (dense) gas-solid fluidized beds. Moreover, we established an infra-red technique for whole-field concentration measurements in gas-solid fluidized beds, and we have demonstrated and built a facility that can measure gas and solids fluxes under reactive conditions (i.e. elevated temperatures up to 400 °C) using endoscopic PIV. The group also owns state of the art equipment for catalyst characterization, membrane characterization and reactor characterization. The equipment list includes – but is not limited to – magnetic suspension balance, two TGA's, DSC, XRD, SEM, AES, viscometers, different membrane permeation setups, kinetic setups, etc.

Our experimental and modelling expertise forms a strong alliance in the development of novel intensified reactor concepts. As an example we mention here the development of a fluidized bed membrane reactor concept for the production of ultra-pure hydrogen with integrated CO2 capture via chemical looping. This involves dedicated studies into various oxygen carriers used for chemical looping by experiments (e.g. thermogravimetric analysis) in conjunction with detailed particle models to describe the redox kinetics, fundamental studies into reactor design and operation of fluidized bed membrane reactors using multiphase flow models (accounting for mass transport and perm-selective membranes), and process systems modelling on industrial scale. The knowledge and tools developed in our group provide a sound basis to place this research activity on a firm footing.

TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification	
Project leaders Prof.dr.ir M. van Sint Annaland, Prof. dr.ir N. Deen, Dr.ir. I. Roghair Research theme Mathematical and computational methods for fluid flow analysis Participants Dario M. Balice Funded by DPI Funded % Industry 100 % Start of the project 2018 Information Dario Balice d.m.balice@tue.nl	Project title HEMPR: Heat management in polymerization reactors Project aim The main objective of this project is to set up a numerical simulative method for the analysis of the behaviour of a fluidized bed with the presence of liquid. Even if this kind of system is widely used (i.e. bulk polyolefin production), there are several uncertainties on design optimization and safety management. For this purpose, will be developed simulative models of decreasing complexity and increasing scale, from a low scale Euler-Lagrange-Lagrange simulation to a whole scale FB reactor using an Euler-Lagrange approach. Progress The permeability of fluidized bed towards fine droplet was investigated and quantified using CDF-DEM simulation. The result of this investigation characterizes the deposition rate of liquid droplets on the particles surface after their injection in the fluidized bed reactor. The thermal behaviour of a reacting particle partially covered by evaporating liquid was investigated building several particle scale models. This allowed to choose a model suitable for describing particle thermal behaviour in larger scale Discrete Particle Model.

TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification	
Project leaders Prof.dr.ir. M. Van Sint Annaland Research theme Complex dynamics of fluids Participants Eindhoven University of Technology Utrecht University Funded by Shell Funded % Industry 100% Start of the project 2017 Information Francesco Sabatino f.sabatino@tue.nl	Project title Assessment of dense energy carriers production routes Project aim Dense Energy Carriers are chemicals produced from carbon dioxide and water using solar energy in biological, photochemical, thermochemical or electrochemical processes. In addition to being carbon-neutral, they also represent a solution for easily storing solar energy. Recently, different Dense Energy Carriers production routes have been proposed, but their sustainability and economic potential aren't always clear. This Project aims at finding the most promising processes for Dense Energy Carriers synthesis from an economic, environmental and societal point of view. Progress In a future in which fossil fuels will not be available anymore, air could be a renewable source of carbon dioxide. For this reason, great consideration has been given to Direct Air Capture (DAC), a process aimed at capturing CO2 from ambient air. The main DAC processes described in literature have been studied and modelled, with the aim of assessing and optimizing their performances. Moreover, a novel process has been proposed and evaluated. The results obtained have shown that adsorption-based approaches to DAC are the most promising. 2020 Publications F. Sabatino, G.J. Kramer, M. van Sint Annaland, Evaluation of a Direct Air Capture Process Combining Wet Scrubbing and Bipolar Membrane Electrodialysis, Industrial & Engineering Chemistry Research 59, pp 7007-7020, 2020.

TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification

Project leaders
Prof. dr. ir. J.A.M. Kuipers
Prof. dr. ir. M. van Sint Annaland

Research theme
Complex dynamics of fluids

Participants
Ir. Alessandro Battistella, Dr. ir. Ivo Roghair, Prof. dr. ir. Martin van Sint Annaland

Cooperation
Delft University of Technology
University of Twente

Funded by
FOM (IPP), AzkoNobel, DSM, Sabic, Shell, Tata Steel

Funded %
NWO 50%
Industry 50%

Start of the project
2015

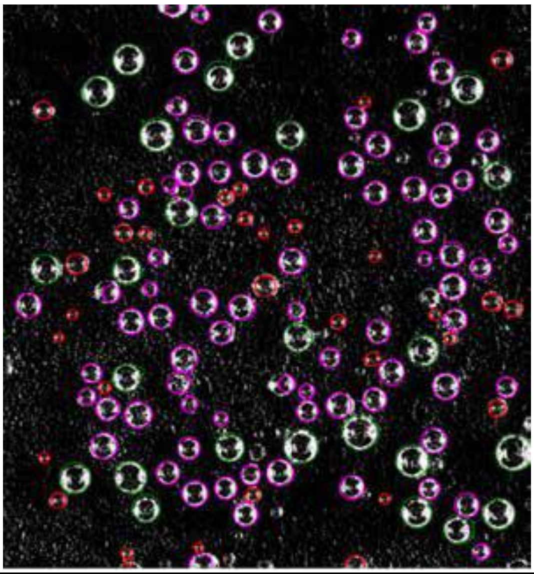
Information
Ivo Roghair
i.roghair@tue.nl

Project title
Heat, mass transport and phase transition in dense bubbly flows simulation of heat & mass transport and phase transitions in dense bubbly flows at intermediate and large scale

Project aim
The main objective of this project is to provide further insight into the interplay between the various mechanisms in bubbly flows involving mass and heat transport or phase transitions at larger scales, in particular bulk bubble nucleation on surfaces. To achieve this goal, we will use the Euler-Lagrange (E-L) and Euler-Euler (E-E) approach to study large scale systems, employing the closure information developed in other projects. Experimental validation using results from other projects for mass and heat transfer. Delivery of scaling laws for dense bubbly flows involving mass & heat transport and phase transitions.

Progress
A bubble nucleation setup has been built, in which supersaturation induced nucleating bubbles on a substrate can be observed optically. Using digital image analysis (DIA) we have analysed the effect of bulk nucleation in terms of bubble nucleation rate, growth and detachment, and linked to the simulation study. The first experiments showed a qualitative agreement with the numerical simulations: the bubble size distribution shows the growth on the substrate shifting towards bigger bubbles.

2020 Publications
On the terminal velocity of single bubbles rising in non-Newtonian power-law liquids. Battistella, A., van Schijndel, S. J. G., Baltussen, M. W., Roghair, I. & van Sint Annaland, M., Apr 2020, In: Journal of Non-Newtonian 278, 11 p., 104249.



TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification	
Project leaders Prof.dr.ir. M. Van Sint Annaland Research theme Complex dynamics of fluids Participants Milan Mihajlović, MSc Prof.dr.ir. M. Van Sint Annaland Dr.ir. Ivo Roghair Funded by EU Funded % EU 100 % Start of the project 2015 Information Ivo Roghair i.roghair@tue.nl	Project title Scale-up and intensification of granular processes Project aim This project is part of European project Intensified-by-Design (IbD®). IbD will create a holistic platform for facilitating process intensification design and optimization in processes in which solids are an intrinsic part. It will develop and upgrade methods for the handling of solids by intensification of currently existing processes, or through completely new approaches to the processing of solids. To support the design of intensified processes for solids handling, phenomenological models are developed making use of more fundamental simulations such as CFD and DEM models. The focus of this project lies on the development of predictive models for fluidized bed reactors, in particular on the effect temperature on the particle collision properties as well as on long-range forces such as Van Der Waals forces. Progress A DEM model is used to investigate the influence of inter-particle forces (van der Waals). Initial results showed the influence of the VdW forces to be higher for Geldart A then Geldart B, it was also found that due to this force particles could agglomerate. In addition, fluidization of Geldart B type particles is tested with different particles collision properties, since it was found in the literature that these properties are also changing with temperature. We have assessed the effect of the normal and tangential restitution coefficients, as well as the friction factor, on the fluidization behaviour, in particular on the minimum fluidization velocity. Moreover, we have developed a consistent modelling technique to incorporate the Van Der Waals forces in DEM models, and assessed their effect on the fluidization behaviour. It turns out that the particle fluxes recorded in experiments can be explained by an increase of interparticle (VDW) forces. 2020 Publications On the numerical implementation of the Van der Waals force in Soft-Sphere Discrete Element Models for gas-solid fluidization. Mihajlovic, M., Roghair, I. & van Sint Annaland, M., 23 Nov 2020, In: Chemical Engineering Science. 226, 115794.

<table>
<tr><th colspan="2">TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification</th></tr>
<tr><td>
Project leaders
Prof.dr.ir M. van Sint Annaland

Research theme
Complex dynamics of fluids

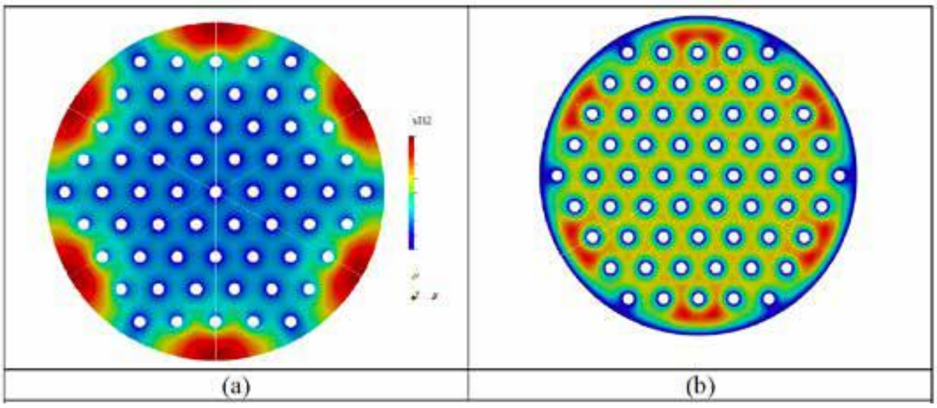
Participants
Marzieh Kordnejad, Prof.dr.ir. M. Van Sint Annaland, Dr.ir. Ivo Roghair

Start of the project
2017

Information
Ivo Roghair
i.roghair@tue.nl
</td><td>
Project title
Tribo-electrification in gas-solid fluidized bed and hydrodynamic effects

Project aim
The main object of this project is to provide fundamental understanding of triboelectrification and the effect of electrostatic force on hydrodynamics of gas-solid fluidized bed. By extending the DPM with electrostatic charge it is possible to study the hydrodynamics of a fluidized bed with charged particles and analyzing the particles motion in the bed. Finally doing experimental works can validate our model.

Progress
Triboelectrification due to particle collisions can be formed in DPM coupled with a condenser model for modelling contact electrification. To analyzing the effect of electrostatic force, predefined charged particles added to DPM model and minimum fluidization and bubbling velocity for different amount of particle charge were determined and it was found that the minimum fluidization velocity decreases and the minimum bubbling velocity increases with the degree of electrification of the particles. Besides, comparison of circulation pattern of fluidized bed showed that number of circulation loop increased in compare to uncharged system.
Experimental works on glass beads were done with charged and uncharged, the results were shown in the charged system, minimum fluidization velocity increased and considering a single bubble rising showed the bubble in charged system shrinked and elongated.
</td></tr>
</table>

TU/e - Chemical Engineering & Chemistry - Chemical Process Intensification	
Project leaders Dr. Jose L Viviente (Technalia) Research theme Mathematical and computational methods for fluid flow analysis Participants Consortium of industrial and academic partners Funded by EU Funded % University 25% Industry 25% EU 50 % Start of the project 2018 Information S.Pouw s.pouw@tue.nl	Project title Advanced MEM Branes and membrane assisted processes for pre and post-combustion CO2 captuRe Project aim The key objective of the MEMBER project is the scale-up and manufacturing of advanced materials (membranes and sorbents) and their demonstration at TRL6 in novel membrane-based technologies that outperform current technology for pre- and post-combustion CO2 capture in power plants as well as H2 generation with integrated CO2 capture. Fluid dynamic modelling is used to obtain fundamental understanding of the mass- and heat transfer characteristics with respect to improving design and performance in reactive separation systems. Progress A start on the modelling of single phase and multiphase membrane modelling for H2 purification is made. From preliminary analysis the effect of product depletion (concentration polarization) has a major effect on the separation performance. Further analysis using OpenFOAM is required to quantify this effect.  *Cross section of membrane module using 61 membranes with triangular orientation in a 1/12 symmetry domain (a) hydrogen molar fraction at retentate side (b) axial velocity profile.*

TU/e - Chemical Engineering & Chemistry - Interfaces with Mass Transfer

Prof.dr. CWM van de Geld

The group 'Interfaces with Mass Transfer' studies gas-liquid interfaces through which mass transfer is taking place. In recent years the research is carried out at the Department of Chemical Engineering and Chemistry, but early work was performed at the Department of Mechanical Engineering, with which strong ties still exist. In the past decades, single boiling bubble detachment and dropwise condensation were focal points of the group, but a variety of related topics such as steam injection, quench cooling of hot plates and endovenous laser ablation have been studied as well. Particle migration in turbulent pipe flow came up as a logical extension of the study of bubble migration in turbulent channel flow.

The approach has always been the design of dedicated experiments accompanied with theoretical analysis based on solutions of well-argued simplifications, supported by more complex numerical simulations where needed. Results encompass empirical correlations and insight in, for example, the importance of added mass forces in bubble detachment.

TU/e – Mathematics and Computer Science - Centre for Analysis, Scientific Computing and Applications (CASA)

Prof.dr.ir. B Koren
Prof.dr. MA Peletier
Prof.dr. JJM Slot

The Centre for Analysis, Scientific Computing and Applications (CASA) embodies the chairs Applied Analysis and Scientific Computing, which both participate in the J.M. Burgerscentrum. CASA's research objective is to develop new and improve existing mathematical methods - both analytical and numerical - for a wide range of applications in science and engineering. Extensive collaborations exist with researchers in other disciplines, at universities, large technological institutes as well as industries, both nationally and internationally. Current CASA research related to fluid dynamics concerns aerodynamics, aeroacoustics, magnetohydrodynamics, fluid-structure interactions, porous media flows, viscous and viscoelastic flows, free-surface flows, particle flows and shape optimization.

TU/e – Mathematics and Computer Science - Centre for Analysis, Scientific Computing and Applications

<u>Project leaders</u>
Nathan van de Wouw
W.H.A. Schilders

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
Mohammad Hossein Abbasi

<u>Cooperation</u>
Laura Iapichino (Netherlands)
Glenn-Ole Kassa (Norway)
Florent Di Meglio (France)
Bart Besselink (Netherlands)
Sajad Naderi Lordejani (Netherlands)
Naveen Velmurugan (France)

<u>Funded by</u>
European Union

<u>Funding %</u>
EU 100%

<u>Start of the project</u>
2016

<u>Information</u>
Mohammad Hossein Abbasi
31 68 799 2220
m.h.abbasi@tue.nl

<u>Project title</u>
HYdraulics modeling for DRilling Automation (HYDRA)

<u>Project aim</u>
The scientific objective of HYDRA is to develop a framework for multi-phase hydraulic modeling and model complexity reduction for drilling operations, delivered in software directly usable in industry. The resulting models uniquely combine high predictive capacity and low complexity enabling their usage in both virtual drilling scenario testing and drilling automation.

<u>Progress</u>
Application of the Reduced Basis method on the Drift Flux Model (DFM), to simulate multi-phase flows, leads to an unstable system. To circumvent this issue, a data-based approach has been investigated to train a reduced system which accurately replicates the input-output behaviour of the DFM and also preserves the key stability properties of the underlying system. Thanks to this incremental stability preservation, the reduced system behaves more robustly to input signals not seen during the training of the reduced model.

<u>2020 Dissertations</u>
M.H. Abbasi, Modeling and order reduction for hydraulics simulation in managed pressure drilling, PhD Thesis, 10 Nov 2020, Eindhoven: Technische Universiteit Eindhoven.

<u>2020 Publications</u>
M.H. Abbasi, S. Naderi Lordejani, C. Berg, L. Iapichino, W.H.A. Schilders, N. van de Wouw, *An approximate well-balanced upgrade of Godunov-type schemes for the isothermal Euler equations and the drift flux model with laminar friction and gravitation*, International Journal for Numerical Methods in Fluids, 2021, 93 (4): 1110-1142.
H. Bansal, P. Schulze, M.H. Abbasi, H. Zwart, L. Iapichino, W.H.A. Schilders, N. van de Wouw, *Port-Hamiltonian formulation of two-phase flow models*, Systems & Control Letters, 2021, 149: 104881.
S. Naderi Lordejani, M.H. Abbasi, N. Velmurugan, C. Berg, J.Å. Stakvik, B. Besselink, L. Iapichino, F. Di Meglio, W.H.A. Schilders, N. van de Wouw, *Modeling and numerical implementation of managed-pressure-drilling systems for the assessment of pressure-control systems*, SPE Drilling & Completion, 2020, 35(4): 598-619.
M.H. Abbasi, L. Iapichino, S. Naderi Lordejani, W. Schilders, N. van de Wouw, *Reduced basis method for managed pressure drilling based on a model with local nonlinearities*, International Journal for Numerical Methods in Engineering, 2020, 121(23): 5178-5199.
M.H. Abbasi, H. Bansal, H. Zwart, L. Iapichino, W.H.A. Schilders, N. van de Wouw, *Power-Preserving Interconnection of Single- and Two-Phase Flow Models for Managed Pressure Drilling*, American Control Conference (ACC), 2020: 3097-3102.
S. Naderi Lordejani, B. Besselink, M.H. Abbasi, G.-O. Kaasa, W.H.A. Schilders, N. van de Wouw, *Control-Oriented Modeling for Managed Pressure Drilling Automation Using Model Order Reduction*, IEEE Transactions on Control Systems Technology, 2020, 29(3): 1161-1174.
M.H. Abbasi, L. Iapichino, B. Besselink, W.H.A. Schilders, N. van de Wouw, *Error estimation in reduced basis method for systems with time-varying and nonlinear boundary conditions*, Computer Methods in Applied Mechanics and Engineering, 2020, 360: 112688.
M.H. Abbasi, L. Iapichino, B. Besselink, W. Schilders, N. van de Wouw, *Error estimates for model order reduction of Burgers' equation*, IFAC-PapersOnLine, 53(2): 5609-5616.

TU/e – Mathematics and Computer Science - Centre for Analysis, Scientific Computing and Applications	
<u>Project leaders</u> Wil Schilders Nathan van de Wouw Laura Iapichino <u>Research theme</u> Mathematical and computational methods for fluid flow analysis <u>Participants</u> Harshit Bansal <u>Cooperation</u> Barry Koren (The Netherlands) Hans Zwart (The Netherlands) Stephan Rave (Germany) Philipp Schulze (Germany) Siep Weiland (The Netherlands) Timo van Opstal (The Netherlands) Peter Bobbert (The Netherlands) <u>Funded by</u> Shell NWO-FOM PhD Scholarship <u>Funding %</u> FOM 50% Industry 50% <u>Start of the project</u> 2016 <u>Information</u> Harshit Bansal +31 682526971 h.bansal@tue.nl	<u>Project title</u> HYdraulic MOdelling for DRilling Automation (HYMODRA) <u>Project aim</u> The aim of the project is to develop a framework for multi-phase hydraulic modelling and model complexity reduction for Managed Pressure Drilling (MPD) operations to explore oil, gas, minerals and geo-thermal energy. Considering the safety critical aspect of MPD, the primary focus is to enable an accurate and precise control of the down-hole pressure while predicting various transient operational scenarios. The objective of this work is to develop models that uniquely combine high predictive capacity and low complexity, and thus enable their usage in virtual drilling scenario testing and in drilling automation strategies for real-time down-hole pressure management. <u>Progress</u> We enhanced the model order reduction approach, ideated by us in 2019, to obtain effective dimensionality reduction for transport-dominated problems with multiple moving and possible merging discontinuous features. Furthermore, we developed a structure-preserving spatial and temporal discretization of the infinite-dimensional port-Hamiltonian representation of commonly used two-phase flow models: the Two-Fluid Model and the Drift Flux Model. Moreover, we put the need of controlling floating-point errors during numerical computations into perspective through some numerical results in the context of port-Hamiltonian-based structure-preserving discretization of multi-phase flow models. <u>2020 Dissertations</u> Bansal, H. (2020). *Structure-preserving model order reduction for drilling automation*. Technische Universiteit Eindhoven. <u>2020 Publications</u> E.J.I. Hoeijmakers, H. Bansal, T.M. van Opstal, P.A. Bobbert, Model order reduction for wafer heating with the method of freezing, Accepted for Proceedings of 13th International Conference on Scientific Computing in Electrical Engineering (SCEE 2020), to be published in Springer series "Scientific Computing in Electrical Engineering" (https://www.win.tue.nl/~hbansal/final_scee.pdf). H. Bansal, S. Weiland, L. Iapichino, W. Schilders, N. van de Wouw, Structure-preserving Spatial Discretization of a Two-Fluid Model, 2020 59th IEEE Conference on Decision and Control (CDC), Jeju, Korea (South), 2020, pp. 5062-5067, doi: 10.1109/CDC42340.2020.9304252. M. H. Abbasi, H. Bansal, H. Zwart, L. Iapichino, W. Schilders, N. van de Wouw, Power-Preserving Interconnection of Single- and Two-Phase Flow Models for Managed Pressure Drilling, 2020 American Control Conference (ACC), Denver, CO, USA, 2020, pp. 3097-3102, doi: 10.23919/ACC45564.2020.9147405. H. Bansal, H. Zwart, L. Iapichino, W. Schilders, N. van de Wouw, Port-Hamiltonian modelling of fluid dynamics models with variable cross-section, IFAC-Papers OnLine, Part of Special Issue: 24th International Symposium on Mathematical Theory of Networks and Systems MTNS 2020: Cambridge, United Kingdom, Volume 54, Issue 9, 2021, Pages 365-372. H. Bansal, P. Schulze, M. H. Abbasi, H. Zwart, L. Iapichino, W.H.A. Schilders, N. van de Wouw, Port-Hamiltonian Formulation of Two-phase Flow Models, Systems & Control Letters, Volume 149, 104881, 2021 (https://doi.org/10.1016/j.sysconle.2021.104881)

TU/e – Mathematics and Computer Science - Centre for Analysis, Scientific Computing and Applications

Project leaders
Prof. Barry Koren,
Dr. Arris Tijsseling,

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Dr. Jan ten Thije Boonkkamp,
Dr. Jan van Dijk (Department of Applied Physics),
Dr. Hanz Martin Cheng

Cooperation
Dr. Diana Mihailova (Plasma Matters),
Dr. Jesper Janssen (Plasma Matters)

Funded by
Eindhoven University of Technology

Funding %
University 100%

Start of the project
2019

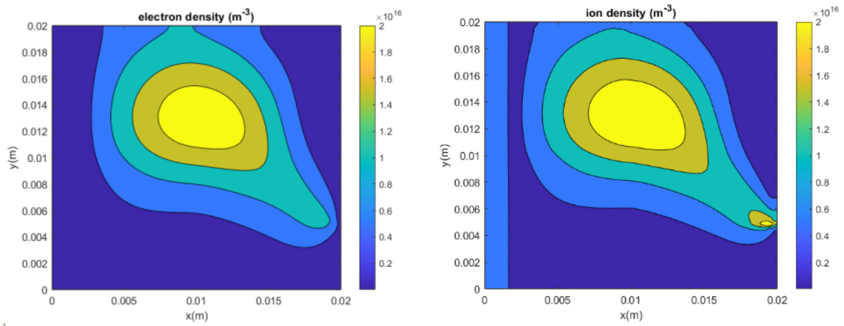
Information
Hanz Martin Cheng
+31616509041
h.m.cheng@tue.nl

Project title
Numerical schemes for advection-dominated equations involving anisotropic diffusion

Project aim
We are currently exploring numerical schemes for coupled systems of equations, with applications to plasma physics, and computational fluid dynamics. The aim of the project is to develop robust numerical schemes for coupled systems of equations, such as the coupling between the Poisson equation and the electron-ion continuity equations for plasma physics, or the coupling between the mass, momentum, and energy equations in Navier-Stokes for computational fluid dynamics. We start with 1D test cases, and aim at extending these numerical schemes so that they are applicable in higher dimensions.

Progress
We were able to propose a numerical scheme for the coupled system of equations involving the electric field and electron-ion densities for magnetized transport in plasmas.

2020 Publications
H. M. Cheng and J. Droniou. An efficient implementation of mass conserving characteristic-based schemes in 2D and 3D. SIAM Journal on Scientific Computing.
H. M. Cheng and J. ten Thije Boonkkamp. A generalised complete flux scheme for anisotropic advection-diffusion equations. Advances in Computational Mathematics.
H. M. Cheng. A fully local hybridized second-order accurate scheme for advection-diffusion equations. Submitted.
H. M. Cheng, J. ten Thije Boonkkamp, J. Janssen, D. Mihailova, J. van Dijk. Combining the hybrid mimetic mixed method with the Scharfetter-Gummel scheme for magnetised transport in plasmas. Submitted.



Numerical solutions obtained from the combined HMM-SG method (Left: electron density; Right: ion density). This numerical test illustrates that the proposed numerical scheme still gives physically expected results, even for a magnetic field of 45 degrees, which yields a highly anisotropic diffusion tensor.

TU/e – Mathematics and Computer Science - Centre for Analysis, Scientific Computing and Applications	
Project leaders Barry Koren Benjamin Sanderse (CWI) Research theme Mathematical and computational methods for fluid flow analysis Participants CWI, GTT, MARIN, Shell Cooperation TU Delft Funded by NWO-TTW Funding % NWO 100% Start of the project 2017 Information Yous van Halder y.vanhalder@gmail.com	Project title SLING Project aim Uncertainty Quantification Surrogate Modeling Machine Learning Fluid Mechanics Digital Twins Progress Developed methods for uncertainty quantification in fluid dynamics. Developed reduced order methods for fluid dynamics Considered digital twinning in fluid mechanics 2020 Publications Y. van Halder, B. Sanderse. Faster flow predictions with intrusive neural networks, ERCIM News, 122, 2020 Y. van Halder, B. Sanderse, B. Koren. Multi-level neural networks for PDEs with uncertain parameters, arXiv: 2004.13128, 2020

TU/e –Built Environment – Building Physics

Prof.dr.ir. B Blocken

Building Physics is the science and engineering of physical processes in urban areas. The work of the Building Physics group was originally focused on wind flow and related processes around buildings and in urban areas, including topics such as air pollutant dispersion, pedestrian-level wind conditions, wind-driven rain on buildings, surface convective heat transfer and natural ventilation. These topics were mainly addressed by numerical simulation with computational fluid dynamics, where validation was performed based on either dedicated measurement campaigns on site or in wind tunnels at other institutes, or on experimental data available in the literature. Recently, the work in the group has been expanded to a wider range of topics in the field of Wind Engineering and Industrial Aerodynamics beyond the realm of Urban Physics, including several projects in Sports Aerodynamics. In 2017, a new atmospheric boundary layer wind tunnel was inaugurated at TU/e, which considerably expands the experimental capabilities of the group.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken (TU/e, KU Leuven)
Dr. A. Ricci, MSc (TU/e, KU Leuven)

Research theme
Complex dynamics of fluids

Participants
Prof.dr.ir. B. Blocken
Dr. A. Ricci, MSc

Cooperation
MARIN
KU Leuven
University of Genova
Ansys CFD
Port of Rotterdam
KNMI
Port of Amsterdam

Funded by
Industry

Funding %
Industry 100%

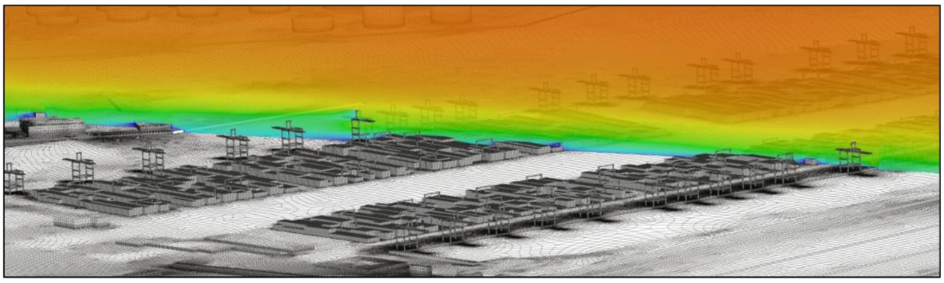
Start of the project
2019

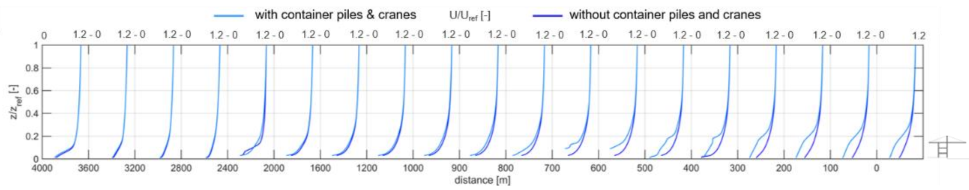
Information
Alessio Ricci
+31 6 38467814
a.ricci@tue.nl

Project title
Microscale wind conditions in seaport areas

Project aim
The present Project aims at investigating ABL winds in complex seaport areas, in order to improve the knowledge of local wind conditions for safety management of port infrastructures, safe maneuvering of ships, and reducing economic losses associated with port disruption due to strong winds. On-site measurements, wind-tunnel testing and CFD simulations are carried out on various seaport areas worldwide. Numerical data, validated with measured data, are then used to characterize and model the wind flow throughout seaports areas in order to estimate wind loads on ships and port infrastructures (e.g. container piles and cranes).

Progress
Two configurations (with and without containers and cranes) of the commercial terminal of the Port of Rotterdam (Netherlands) are simulated with 3D steady RANS approach. Vertical wind speed and turbulence intensity profiles are compared at different locations of the terminal for the two configurations. The vertical wind speed profiles are fitted to different logarithmic laws to characterize the wind inside and above the canopy layer. The standard logarithmic law, commonly adopted to describe ABL winds in open terrains, is found to hold only at about 50% of the monitored positions with a consequent impact in terms of wind forces on ships and port infrastructures.





2020 Dissertations
This is a postdoctoral research project and without a final dissertation.

2020 Publications
Ricci A., Blocken B., 2020. On the reliability of the 3D steady RANS approach in predicting microscale wind conditions in seaport areas: The case of the IJmuiden sea lock. Journal of Wind Engineering and Industrial Aerodynamics 207, 104437.
Ricci A., Janssen W.D., van Wijhe H.J., Blocken B., 2020. CFD simulation of wind forces on ships in ports: Case study for the Rotterdam Cruise Terminal. Journal of Wind Engineering and Industrial Aerodynamics 205, 104315.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken (TU/e, KU Leuven)
Prof.dr.ir. G.J.F. van Heijst (TU/e)
Dr. ir. T. van Hooff (TU/e)

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Ir. C. Alanis Ruiz, PDEng

Cooperation
ANSYS CFD

Funded by
Research Foundation – Flanders (FWO)

Funding %
Scholarships 100% (FWO Flanders)

Start of the project
2019

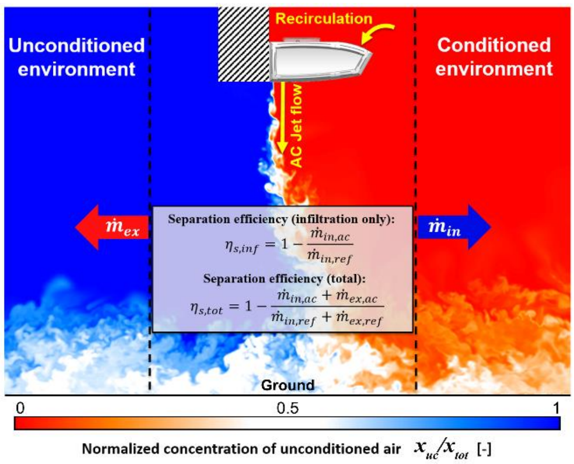
Information
Claudio Alanis Ruiz
+316 4277 6815
c.a.alanis.ruiz@tue.nl
claudio.alanisruiz@kuleuven.be

Project title
Analysis and improvement of separation efficiency of air curtains in the built environment

Project aim
Air curtains, which are plane impinging jets, are used to separate a conditioned environment, typically in terms of temperature or concentration, from an unconditioned environment, while allowing unobstructed traffic between the two environments. The present Project aims to optimize the separation efficiency of air curtains by exploring the influence of jet and environmental parameters on the jet behaviour and resulting separation efficiency. The project combines advanced numerical simulation and experimental techniques in fluid dynamics for the parametric and systematic analysis of air curtain systems and the evaluation of their performance.

Progress
This year has been dedicated to the preparation of publications, accounting for 2 conference papers and 2 journal papers on the PhD research topic that have been or will be published in 2021. Moreover, we have participated in various other research projects related to the area of fluid dynamics applied to buildings, which have led to the publication of 2 journal papers (1 as first author) in 2021. Additional progress has been made in the preparation of the experimental campaign for the PhD project, in which PIV measurements for a range of configurations will be conducted at reduced scale in the Applied Physics lab at TU/e. Its actual execution is now scheduled for summer 2021.

2020 Publications
Khayrullina, A., van Hooff, T., Alanis Ruiz, C., Blocken, B., van Heijst, G.J.F. 2020. Minimum momentum flux ratio required to prevent air curtain breakthrough in case of cross-curtain pressure gradients: CFD versus analytical equation. Building Simulation 13(4): 943-960.



Graphical abstract of peer-reviewed paper 'Air curtain performance: Introducing the adapted separation efficiency' published in journal 'Building and Environment' (Alanis Ruiz et al., 2021).

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken
Dr.ir. Alessio Ricci
Dr. Hamid Montazeri, MSc

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
L. Xia, MSc

Cooperation
PlasmaMade
Ansys CFD
Johan Cruijff ArenA
Signify

Funded by
China Scholarship Council (CSC)

Funding %
Scholarships 100%

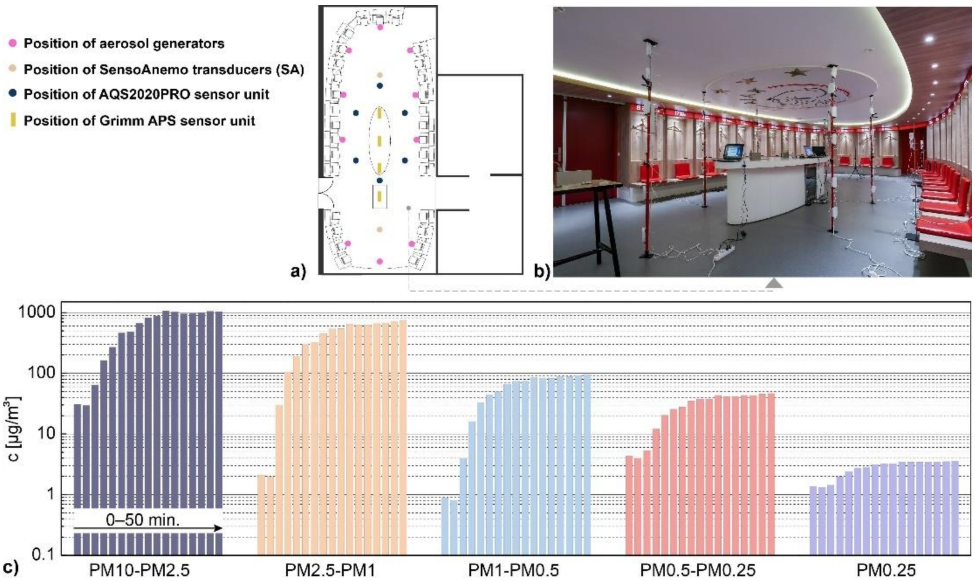
Start of the project
2017

Information
Lili Xia
068522974
l.xia@tue.nl

Project title
Measurements and numerical simulation of aerosol particle dispersion in complex indoor and outdoor airflows

Project aim
The aim of the project is CFD model development and validation for the assessment of aerosol particle dispersion in airflows occurring in a variety of (semi-)indoor environments. This includes aerosol particle release by mist spraying nozzles employed for direct evaporative cooling but also respiratory aerosol particles that can transmit the SARS-CoV-2 virus.

Progress
In this stage of the project, aerosol particle concentrations in two players' dressing rooms of the Johan Cruijff ArenA Football Stadium in Amsterdam have been analyzed. Aerosol particles were generated artificially and aerosol and CO_2 concentration, air temperature, relative humidity and air velocity were measured. The measured data were used to validate the simulated data obtained with 3D steady RANS simulations. A good agreement between the measured and simulated data was found in terms of aerosol concentration with an average deviation of about 16% and 30 % for room 1 and room 2, respectively.



(a) Measurement setup of aerosol concentration in the dressing rooms of the Ajax team; (b) Photo of the measurement setup; (c) 3-minute average aerosol concentration from the four Grimm sensors installed.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken
Dr.ir. H. Montazeri

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
X. Zheng MSc

Cooperation
Ansys CFD

Funded by
China Scholarship Council

Funding %
Scholarships 100%

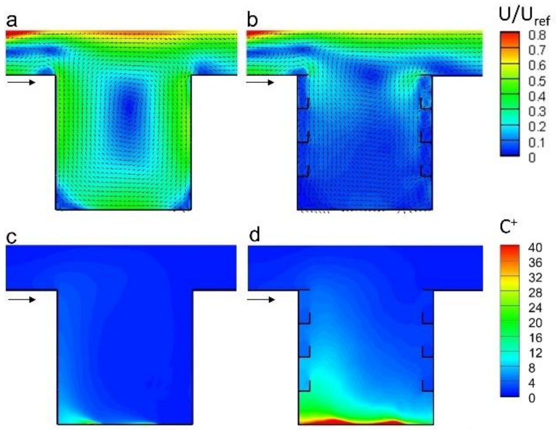
Start of the project
2016

Information
Name Xing Zheng
+31 627154455
x.zheng.1@tue.nl

Project title
Computational analysis of the impact of façade geometrical details on wind flow and pollutant dispersion

Project aim
The main objective of this research is to obtain fundamental knowledge about the impact of façade geometrical details on the wind flow and pollutant dispersion around buildings. This leads to the following sub-objectives: (1) Compare and validate CFD models for buildings with façade geometrical details; (2) Investigate the impact of balcony geometrical characteristics on wind speed on balcony spaces and surface pressure for isolated buildings; and (3) Explore the impact of building balconies on wind flow and pollutant dispersion in generic street canyons.

Progress
LES simulations have been performed for street canyons with and without balconies to investigate the impact of building balconies on pollutant dispersion in street canyons. The evaluation was based on validation with wind-tunnel measurements of pollutant concentration for a generic urban street canyon. Four cases were considered: (1) a street canyon without balconies; (2) a street canyon with balconies on both windward and leeward sides; (3) a street canyon with balconies on the windward side; and (4) a street canyon with balconies on the leeward side.



Normalized mean wind speed (U/U_{ref}) in the cross-section (center plane) of (a) street canyon without balconies, (b) street canyon with balconies on both windward and leeward sides, (c-d) Same for normalized mean concentration (C^+).

2020 Publications
X. Zheng, H. Montazeri, and B. Blocken. "CFD simulations of wind flow and mean surface pressure for buildings with balconies: Comparison of RANS and LES." Building and Environment 173 (2020): 106747.

TU/e – Built Environment – Building Physics

<u>Project leaders</u>
Dr.ir. H. Montazeri
Prof.dr.ir. B. Blocken
Dr. ir. A. Rezaeiha

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
S. Sahebzadeh, MSc

<u>Cooperation</u>
Ansys CFD

<u>Funded by</u>
Ministry of Science, Research and Technology, Iran

<u>Funding %</u>
Scholarships 100%

<u>Start of the project</u>
2018

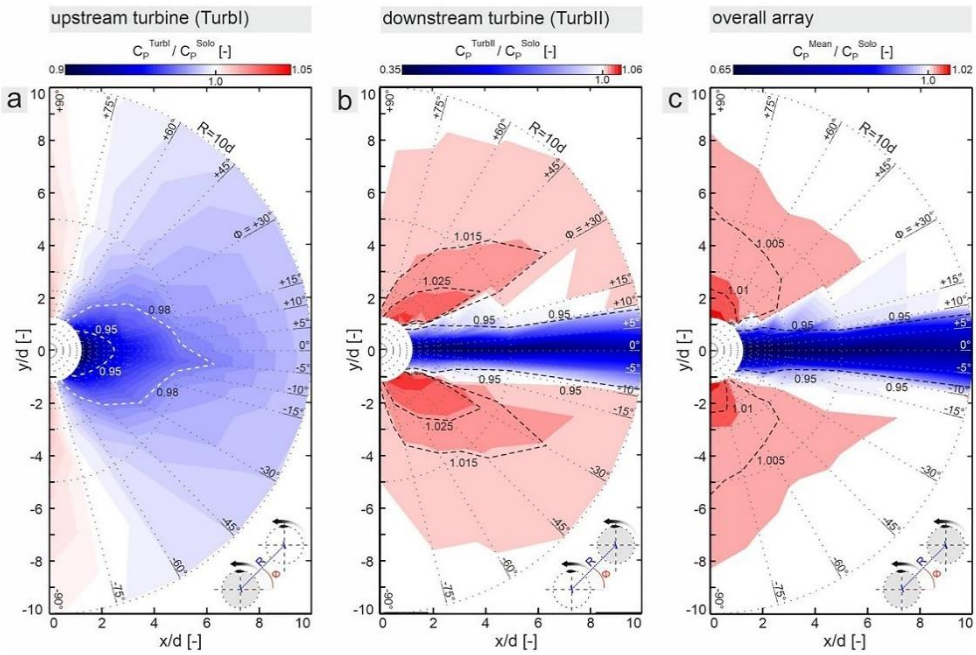
<u>Information</u>
Sadra Sahebzadeh
s.sahebzadeh@tue.nl

<u>Project title</u>
Numerical and experimental analysis of the potential for building-integrated wind energy harvesting in urban areas

<u>Project aim</u>
This PhD research Project aims to provide fundamental knowledge on (i) the optimal geometry and positioning of building-integrated ducted wind turbines for wind energy harvesting purposes; and (ii) the aerodynamics, power performance and wake interactions of an array of vertical axis wind turbines (VAWTs) placed in proximity for layout optimization of urban VAWT wind farms.

<u>Progress</u>
In the second part of the project, the effect of the relative distance and angle on the power and aerodynamic performance of two closely spaced vertical-axis wind turbines was studied with URANS simulations (see the figure). The analysis was expanded to include the effect of relative rotational direction, phase lag, positioning and the number of blades. In addition, the impact of building-integrated ducts on their wind energy potential was investigated with RANS simulations in combination with design of experiments and artificial neural network modeling.

<u>2020 Publications</u>
Sahebzadeh, Sadra, Abdolrahim Rezaeiha, and Hamid Montazeri. "Towards optimal layout design of vertical-axis wind-turbine farms: Double rotor arrangements." *Energy Conversion and Management* 226 (2020): 113527.
Sahebzadeh, Sadra, Abdolrahim Rezaeiha, and Hamid Montazeri. "Impact of relative spacing of two adjacent vertical axis wind turbines on their aerodynamics." *Journal of Physics: Conference Series*. Vol. 1618. No. 4. IOP Publishing, 2020.



Contour plots of the power performance for the individual turbines and the overall array in R-Φ space (Note the difference in the colourbars).

TU/e – Built Environment – Building Physics

Project leaders
prof.dr.ir. G. Solari (U Genoa)
prof.dr.ir. B. Blocken
Dr. M. Burlando, MSc (Genoa)
Dr. A. Ricci, MSc

Research theme
Complex dynamics of fluids

Participants
J. Žužul, MSc

Cooperation
University of Genoa (Italy)
KU Leuven
Ansys CFD
University of Western Ontario (Canada)

Funded by
European Research Council (ERC)

Funding %
EU 100 %

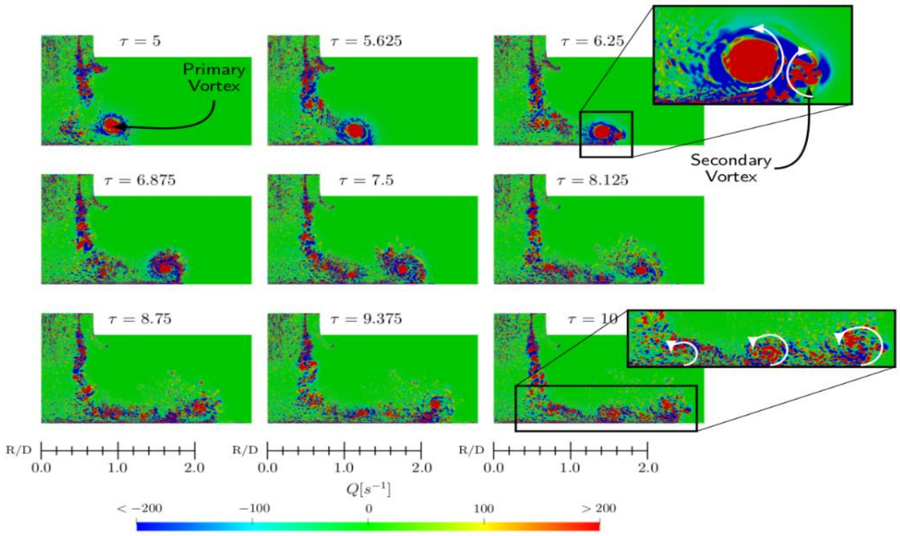
Start of the project
2018

Information
Name: Josip Žužul
j.zuzul@tue.nl
josip.zuzul@edu.unige.it

Project title
CFD modeling of the thunderstorm downbursts

Project aim
This PhD project is a part of the wider European Research Council (ERC) project "THUNDERR - Detection, simulation and modeling of thunderstorm outflows to design wind-safer and cost-efficient structures". It aims at extending the knowledge about the complex and hazardous thunderstorm downburst extreme winds and their devastating effects on low-rise buildings and structures by CFD simulations. When simultaneously used with the wind-simulator and full-scale measurement data, it should provide adequate knowledge in order to propose an easy-to-use engineering standard for designing the low-rise structures able to withstand such wind loads.

Progress
In the first stage, URANS and SAS simulations were performed to reproduce the flow of the wind-simulator facility "WindEEE Dome" previously used to generate the same downburst winds experimentally. In the second stage, LES simulations were performed in order to resolve the flow at higher level of complexity. The most important flow characteristics (*e.g.* maximum outflow velocity, fluxes, turbulence intensity, etc.) were examined throughout the entire lifespan of the phenomenon to determine the most critical aspects for structural loading. Special focus was placed on the analysis of the ring vortices, which significantly contribute to the complexity and destructive behaviour of the flow.



TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken
Dr. A. Ricci, MSc

Research theme
Complex dynamics of fluids

Participants
P. Qin, MSc

Cooperation
Ansys CFD

Funded by
China Scholarship Council (CSC)

Funding %
Scholarships 100%

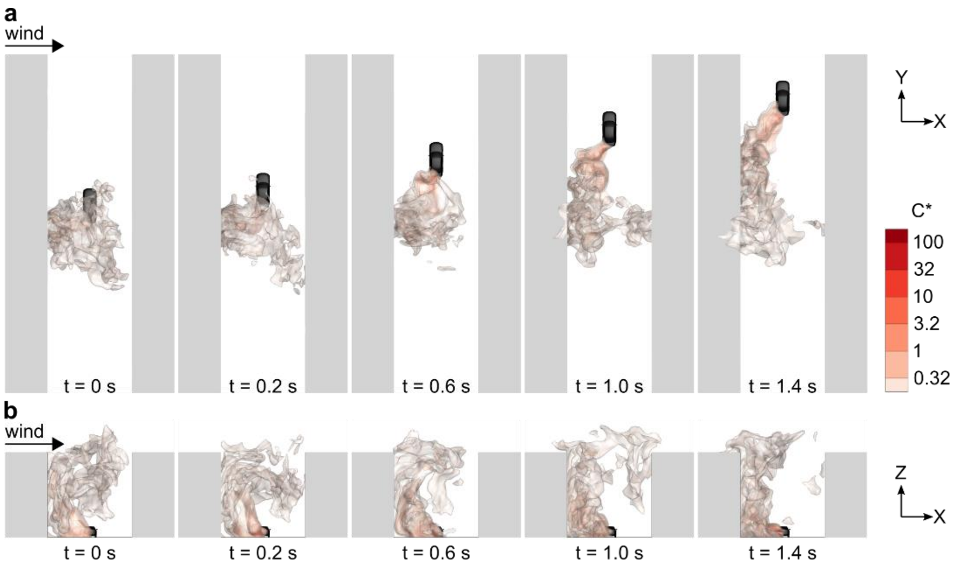
Start of the project
2018

Information
Peng Qin
p.qin@tue.nl

Project title
Numerical and experimental analyses of pollutant dispersion in urban areas

Project aim
The dispersion of pollutants is of great relevance for people living in urban areas and this is also the reason why the interest in urban physics is rapidly growing. Streets and squares are generally the main zones where human activities take place but also where pollutants show a higher concentration compared to open areas. The present Project aims at investigating the basic processes of pollutant dispersion caused by vehicles by means of three idealized case studies (i.e. a street, a street canyon and an urban area), using different approaches for modeling of the pollutant sources, different wind conditions and elements of vegetation to mitigate the effect of pollutants on human beings.

Progress
In the first stage, a validation study based on the WT data from the literature was carried out with an "ideal source" placed in the centre of a single street canyon. The impact of grid resolution, turbulence models, turbulent Schmidt numbers and numerical approaches (i.e. RANS, SAS and LES) were systematically investigated. Thereafter, the best-performing numerical setting was used to investigate the pollutant dispersion in the same street canyon but using a static and dynamic "realistic source", such as the "DrivAer" car model. A comparison of numerical results in terms of wind speed, turbulent kinetic energy and pollutant concentration inside the street canyon was performed.



LES results for dispersion process of a dynamic realistic source in a street canyon at different times.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken (TU/e, KU Leuven)
Dr.ir. T. van Hooff (TU/e)
Dr.ir. R. Gijsbers (ENS)

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Ir. R. Vervoort, PDEng

Cooperation
ENS Technology
ANSYS CFD

Funded by
ENS Technology

Funding %
Industry 100%

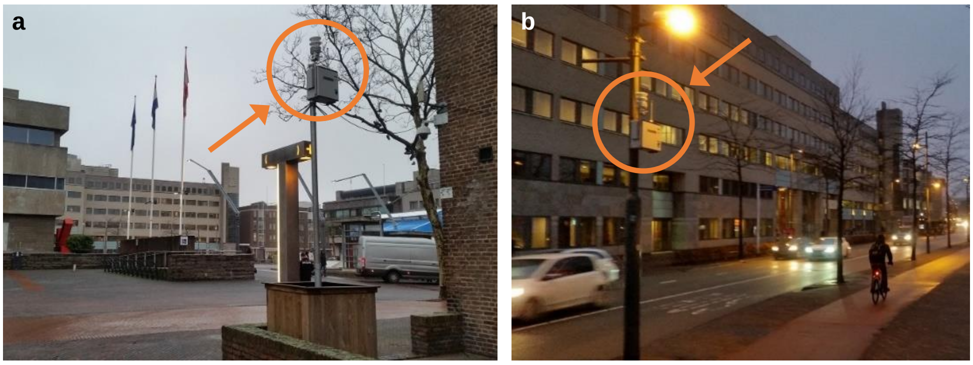
Start of the project
2019

Information
Rob Vervoort
+31681804108
r.p.m.vervoort@tue.nl

Project title
Numerical and experimental assessment and optimization of particulate matter removal in the built environment

Project aim
Particulate matter (PM) is one of the most dangerous forms of air pollution. Adverse health effects are well documented, supporting a consensus regarding a (dose-dependent) causal relationship between PM exposure and human morbidity and mortality. This study focuses on the assessment and optimization of indoor and outdoor PM concentration reductions in the built environment, by local removal using electrostatic precipitation (in semi-enclosed environments). The study comprises multiple main objectives in which both experimental (full-scale and reduced-scale) and numerical work is conducted.

Progress
One of the Project aims is to generate experimental data to support CFD modeling of local PM mitigation strategies in the built environment. A measurement campaign with 20 to 30 electrostatic precipitation units was conducted in the city centre of Eindhoven. To gain a better understanding of the dynamic behaviour of PM in relation to (local) weather conditions, and potential (general) implications for experimental assessment of PM mitigation strategies, firstly, an extensive set of data - collected at 10 locations within the inner-city centre of Eindhoven over a period of 110 days - was studied in detail. Analyses and reports are being finalized momentarily.



Photographs of 2 (out of 10) monitoring stations (Grimm 164DM spectrometer + Luft WS500 weather sensor) as adopted in the measurement campaign in the city centre of Eindhoven.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken

Research theme
Complex dynamics of fluids

Participants
Ir. T. van Druenen

Cooperation
Cycling Team Jumbo-Visma
AGU
Sportconfex
KU Leuven
Ansys CFD

Funded by
Various industry sources

Funding %
Industry 100%

Start of the project
2017

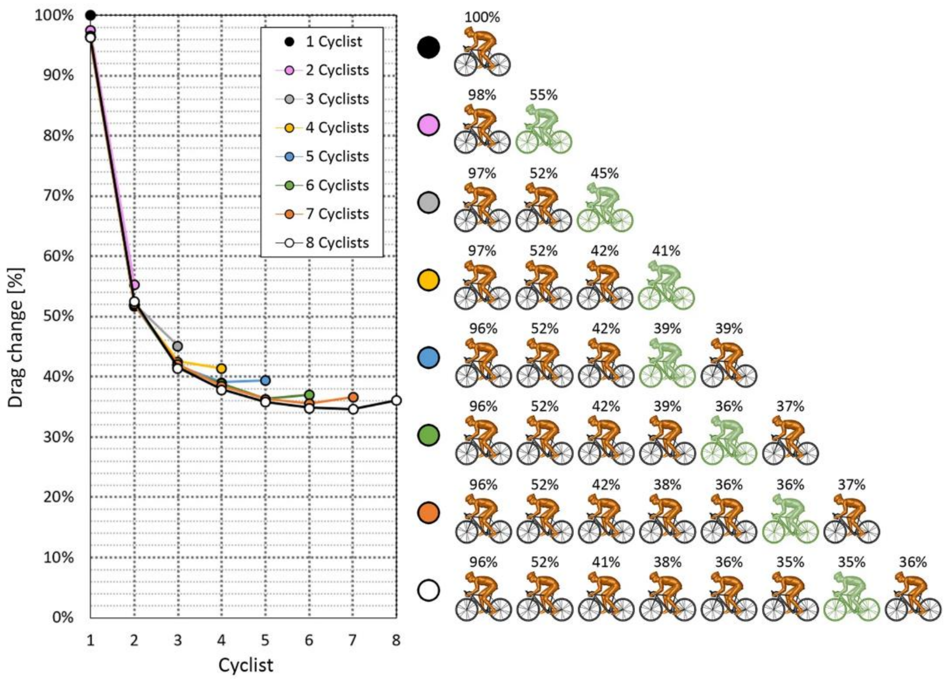
Information
Thijs van Druenen
t.v.druenen@tue.nl

Project title
Cycling aerodynamics: optimization of individual and team time trial aerodynamics.

Project aim
At racing speeds exceeding 50 km/h the aerodynamic drag contributes of more than 90% of the cyclist's total resistance. As margins between winning and losing are often very small, even minor reductions in drag can have a large impact on the final result. In this project, insight in cycling aerodynamics is obtained by the combination of wind tunnel experiments and numerical (CFD) simulations. Drag reduction is obtained by optimizing the cyclist's posture on the bike or position within a group. In addition, the effect of roughness/texture on fabrics and equipment is investigated.

Progress
Numerical (CFD) studies are being performed to evaluate the effect of drafting in uphill, breakaway and (team) time trial configurations in cycling. Also the impact of different cycling positions and the numerical modeling of roughness on cycling suits is assessed. The results are supported by grid-sensitivity analyses and validation by wind tunnel measurements.

2020 Publications
Blocken B, Malizia F, van Druenen T, Gillmeier SG. 2020a. Aerodynamic benefit for a cyclist by drafting behind a motorcycle. Sports Engineering 23: 19.
Blocken B, van Druenen T, van Hooff T, Verstappen PA, Marchal T, Marr LC. 2020b. Can indoor sports centers be allowed to re-open during the COVID-19 pandemic based on a certificate of equivalence? Building and Environment 180: Article nr. 107022



Drag of every cyclist in configurations up to eight cyclists, as a percentage of the drag of the isolated cyclist. The cyclist with the lowest drag is coloured in green.

TU/e – Built Environment – Building Physics

<u>Project leaders</u>
Prof.dr.ir. B. Blocken
Dr. H. Montazeri, MSc
Dr.ir. A. Rezaeiha
Prof.dr. A.S. Yang

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
Y.H. Juan, MSc

<u>Cooperation</u>
Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, Taipei, Taiwan
Ansys CFD

<u>Funded by</u>
Ministry of Science and Technology, Taiwan (Contract No. 107-2917-I-027-001)
Ministry of Education, Taiwan

<u>Funding %</u>
Scholarships 100%

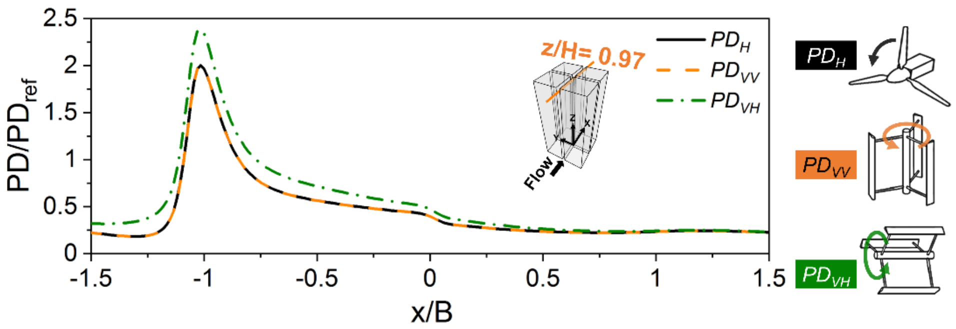
<u>Start of the project</u>
2018

<u>Information</u>
Yu-Hsuan Juan
+31633150210
y.juan@tue.nl

<u>Project title</u>
Analysis of urban wind energy potential around generic high-rise buildings in close proximity using computational fluid dynamics

<u>Project aim</u>
The PhD Project aims to improve urban wind energy potential around generic high-rise buildings placed in close proximity by variations of the urban and building morphological parameters. The evaluations are based on wind-tunnel measurements and computational fluid dynamics (CFD) simulations.

<u>Progress</u>
Urban wind energy potential for a generic 2×2 high-rise building array placed in close proximity is investigated using wind-tunnel measurements and steady Reynolds-averaged Navier-Stokes (RANS) simulations. The studied parameters are (i) building arrangement and height, (ii) building corner modifications, (iii) wind turbine type and orientation, and (iv) wind direction. The results show that large improvements can be obtained by building corner modifications and that the horizontally-mounted vertical axis wind turbine is the most promising option for wind energy harvesting in the passage between the buildings and on the rooftop.



Profiles of dimensionless power density for different turbine types, i.e. HAWT (PD_H), VAWT (PD_{VV}), and horizontally-aligned VAWT (PD_{VH}), along horizontal lines at z/H = 0.97.

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. B. Blocken (TU/e, KU Leuven)

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Dr.ir. A. Rezaeiha (KU Leuven, TU/e)

Cooperation
KU Leuven

Funded by
FWO (Fonds Wetenschappelijk Onderzoek)

Funding %
FWO 100%

Start of the project
2019

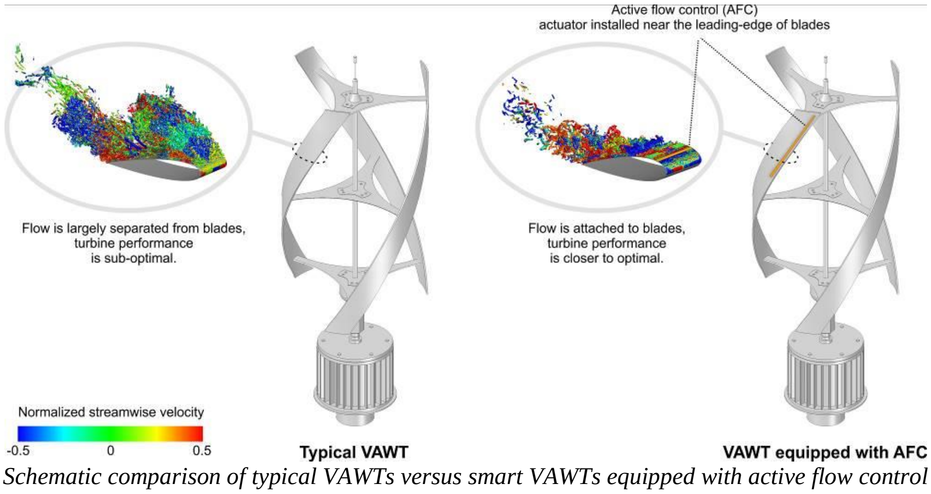
Information
Abdolrahim Rezaeiha
a.rezaeiha@tue.nl

Project title
Experimental and numerical analysis of smart wind turbines

Project aim
The aim of this research project is to obtain fundamental knowledge on active manipulation of complex flow phenomena relevant to Vertical axis wind turbines (VAWTs) to provide new insights and move towards novel smart VAWTs with optimal omni-directional performance.

Progress
The project is the continuation of the PhD project of the same researcher. During the last phase of his PhD, he found promising results for power enhancement of VAWTs by using steady boundary layer suction. Then, he investigated the employment of unsteady suction, activated only during a selected azimuthal region of the turbine rotation, to suppress dynamic stall, improve the turbine aerodynamic performance and minimize the power consumption of the suction system. With his PhD student, he is also studying optimal wind farm layout design for VAWTs.

2020 Publications
Rezaeiha A, Montazeri H, and Blocken B, "A framework for preliminary large-scale urban wind energy potential assessment: Roof-mounted wind turbines," Energy Conversion and Management, vol. 214 (C), p. 112770, 2020.
Sahebzadeh S, Rezaeiha A, and Montazeri H, "Towards optimal layout design of vertical-axis wind-turbine farms: Double rotor arrangements," Energy Conversion and Management, vol. 226 (C), p. 113527, 2020.
Rezaeiha, A., Montazeri, H., & Blocken, B. (2020). Scale-Adaptive Simulation (SAS) of dynamic stall on a wind turbine Notes on Numerical Fluid Mechanics and Multidisciplinary Design - Progress in Hybrid RANS-LES Modelling (HRLM 2018) (1 ed., Vol. 143, pp. 323-333): Springer International Publishing.
Rezaeiha A, Montazeri H, and Blocken B, "CFD investigation of separation control on a vertical axis wind turbine: Steady and unsteady suction," Journal of Physics: Conference Series, vol. 1618, p. 052019, 2020.
Sahebzadeh S, Rezaeiha A, and Montazeri H, "Impact of relative spacing of two adjacent vertical axis wind turbines on their aerodynamics," Journal of Physics: Conference Series, vol. 1618, p. 042002, 2020.



Schematic comparison of typical VAWTs versus smart VAWTs equipped with active flow control (AFC).

TU/e – Built Environment – Building Physics

Project leaders
Prof.dr.ir. Bert Blocken
Dr. Romain Guichard
Dr. Alessio Ricci
Dr. Stefanie Gillmeier

Research theme
Complex dynamics of fluids

Participants
A. K. Radhakrishnan Jayakumari

Cooperation
INRS (French National Research and Safety Institute for the Prevention of Occupational Accidents and Diseases)
Ansys CFD

Funded by
INRS

Funding %
INRS 100 %

Start of the project
2018

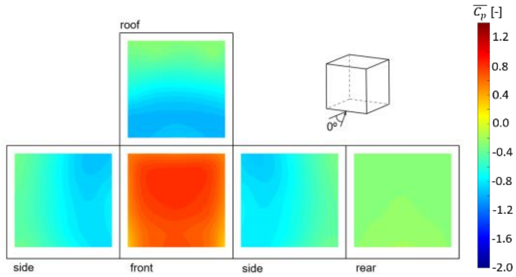
Information
A. K. Radhakrishnan Jayakumari
+31 6 23360946
a.k.radhakrishnan.jayakumari@tue.nl

Project title
Analysis of the effect of wind on the performance of industrial ventilation systems

Project aim
The aim of this project is to systematically investigate and quantify the effect of meteorological conditions on the performance of mechanical ventilation systems in pollutant containment zones for example during asbestos removal, by predicting the internal-external pressure differences. Understanding the impact of the wind can help to enhance the performance of these ventilation systems and prevent hazardous pollutants escaping into the atmosphere. The investigations will be carried out using (i) field measurements, (ii) wind-tunnel (WT) testing; (iii) computational fluid dynamics (CFD) techniques and (iv) simulations with a ventilation network model.

Progress
In the first stage, field measurements were carried out in an active asbestos removal worksite in France. To gain further understanding of the problem, internal-external pressure differences at various locations of the containment zone, wind velocities and local wind directions were simultaneously measured. In the second stage, wind tunnel (WT) tests are performed in order to understand the applicability of large WT scales (such as 1:40) for buildings equipped with a mechanical ventilation system. Pressure coefficients are measures on the building facades at two different scales (i.e. 1:300 and 1:40) at the WT facility of TU/e.

Photograph of the WT setup



Mean pressure distribution for 0° wind direction

REPORTS OF INDIVIDUAL RESEARCH GROUPS

UNIVERSITY OF TWENTE.

Faculty of Science and Technology

Physics of Fluids (PoF)
Physics of Complex Fluids
Soft Matter, Fluidics and Interfaces

Faculty of Engineering Technology

Department of Thermal and Fluid Engineering

Engineering Fluid Dynamics
Thermal Engineering
Multiscale Mechanics

Department of Civil Engineering

Water Engineering and Management

Faculty of Electrical Engineering, Mathematics and Computer Science (EEMCS)

System Analysis and Computational Science (SACS)

Mathematics of Computational Science
Multiscale Modelling and Simulation

UT – Science and Technology - Physics of Fluids

Prof.dr. D Lohse
Prof.dr. D van der Meer
Prof.dr. M Versluis
Prof.dr.ir. J Snoeijer
Prof.dr. X Zhang

The Physics of Fluids group in Twente works on a variety of aspects in fluid mechanics, in particular on those related to bubbles. The focus of our work is the fundamental understanding the phenomena of the physics of fluids, bubbles and jets, which we undertake by experimental, numerical and theoretical means. Besides in the J.M. Burgerscentrum, our research is embedded in the Research Institute of Mechanics, Processes and Control IMPACT, the MESA+ Institute, and the Research Institute for Biomedical Technology BMTi of the University of Twente. The group receives external research funds mainly from FOM, but also from STW, NWO, SenterNovem, EU and several companies. The focus research areas of the group are:

Turbulence and Two-Phase Flow
Fully developed turbulence is one of the big unsolved problems in fluid dynamics. The main question is the distribution of rare events, which has important implications for, e.g., flight safety. We approach this problem from a fundamental point of view, both experimentally, theoretically, and numerically. One particular important type of turbulence is turbulence (partly) driven by body forces, such as buoyancy. This can happen by either thermally driving the turbulence or also by driving the turbulence through bubbles or dispersed particles. Both will be advected by the flow but also act back on the surrounding liquid (two-way coupling). To be able to describe flow with many bubbles or particles efficiently, one needs an effective force description, on which and with which we work in several projects within our group. Finally, we are also interested in the radial dynamics of single bubbles in hydrodynamic or acoustic fields.

Granular Flow
Granular flows are fundamentally different from any other type of flow. In our research we focus on the clustering phenomenon that finds its origin in the inelastic collisions between the particles. There is much emphasis on the onset of clustering, which happens via a phase transition which is studied in both compartmentalized and continuous systems. Another line of our research deals with the impact of objects on very fine, decompactified sand, in which we explore the applicability of fluid models to granular systems. We uncovered links to distant phenomena like asteroid impact and a dry variety of quicksand.

Micro- and Nanofluidics
The physics of fluids at the microscale can be quite different from macrofluidic behaviour. Here we study disturbing bubbles in microchannels found in ink jet printing. By patterning surfaces on sub-micron scales we try to identify individual ‘nanobubbles’ which may lead to a quantitative understanding of wall slip. These patterned surfaces may also serve as nucleation sites for cavitation bubbles generated through intense negative pressures.

UT – Science and Technology - Biomedical Flow

Prof.dr. A Prosperetti
Prof.dr. R Verzicco

Bubbles have various applications in the biomedical field. Coated microbubbles are used in ultrasound imaging to enhance the contrast in cardiac or liver perfusion images. Bubbles can be targeted to specific cells for molecular imaging to non-invasively detect the presence and location of diseases such as cancer or atherosclerosis. Furthermore, the bubbles can be exploited to generate acoustic streaming and jetting near cell boundaries which leads to permeation, destruction or removal of target cells.

UT - Science and Technology - Physics of Fluids

Project leaders
Prof. Jacco Snoeijer

Research theme
Complex dynamics of fluids

Participants
J.H. Snoeijer
M. Kansal
M. Essink
C. Datt
A. Bouillant

Cooperation
B. Andreotti (Paris)
J. Eggers (Bristol)
H. van Brummelen (TUE)

Funded by
VICI NWO

Funding %
NWO Other 100%

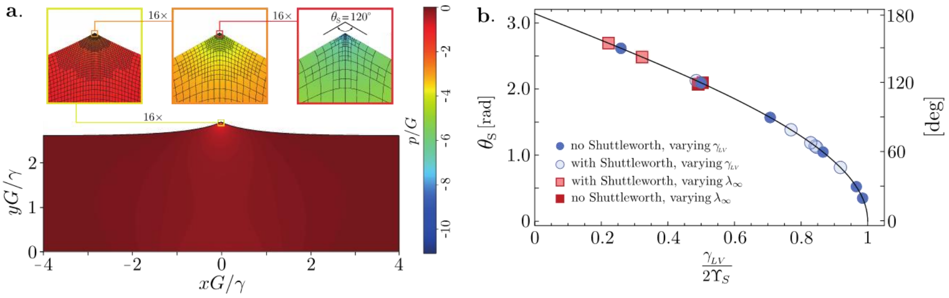
Start of the project
2018

Information
Jacco Snoeijer
j.h.snoeijer@utwente.nl

Project title
Soft Contact

Project aim
The contact of soft interfaces lies at the core of many natural phenomena and technologies: from adhesion of nanoparticles, design of self-cleaning surfaces, to placing a contact lens on the tear film of an eye. The dynamics during contact of soft interfaces can neither be described by the classical laws of solid adhesion, nor by standard interface mechanics. This program aims to establish unifying framework for Soft Contact, with a combined input from fluid physics, solid mechanics and soft matter. Results will be applied to a variety of problems, from the wetting of viscoelastic fluids to interfacial instability of soft elastic and biological materials.

Progress
We resolved a hotly debated controversy on the nature of the singularity below a contact line wetting a soft solid. Specifically, the role of surface tension with regards, and the selection of the solid contact angle via Neumann's law, was demonstrated in simulations, analytical theory and experiments. Further progress was made on the relationship between the flow of viscoelastic liquids and exact solutions in elasticity theory. This led to the first consistent (non-slender) theoretical description of the breakup of polymer threads and offers a new route of explorations of strongly unsteady viscoelastic flow after establishing contact.



Elastic deformation below a liquid drop wetting a soft solid. The elastocapillary singularity is characterized by Neumann's laws.

2020 Publications

B. Andreotti & J.H. Snoeijer, "Statics & Dynamics of Soft wetting", Ann. Rev. Fluid Mech. 52, 285 (2020)

J.H. Snoeijer, A. Pandey, M.A. Herrada & J. Eggers, "The relationship between viscoelasticity and elasticity", Proc. Roy. Soc. A, 476, 20200419 (2020).

A. Pandey, B. Andreotti, S. Karpitschka, G.J. van Zwieten, E.H. van Brummelen, J.H. Snoeijer, "Singular nature of the elastocapillary ridge". Phys. Rev. X 10 031067 (2020)

M. van Gorcum, S. Karpitschka, B. Andreotti, J.H. Snoeijer, "Spreading on viscoelastic solids: Are contact angles selected by Neumann's law", Soft Matter 16, 1306 (2020).

J. Eggers, M.A. Herrada, J.H. Snoeijer "Self-similar breakup of polymer threads as described by the Oldroyd-B model", J. Fluid Mech. 887, A19 (2020).

Project leaders
Richard Stevens

Research theme
Wind-farm modeling

Participants
Ariane Emmanuelli, Srinidhi N. Gadde Luoqin Liu, Jessica Strickland, Anja Stieren

Cooperation
Charles Meneveau (Johns Hopkins University, USA)
Dennice F. Gayme (Johns Hopkins University, USA)
Luis A. Martínez-Tossas (NREL, USA)
Joachim Peinke (Oldenburg, Germany)
Mengqi Zhang (NUS, Singapore)
Didier Dragna (Lyon, France)

Funded by
NWO

Funding %
NWO 80%
EU 20%

Start of the project
2016

Information
Dr. Richard Stevens
r.j.a.m.stevens@utwente.nl

Project title
Wind farm fluid dynamics

Project aim
In this project we use large eddy simulations to model the interaction between wind farms with many turbine rows in the downstream direction and the atmospheric boundary layer. We focus on the effect of the layout of the wind farm on its total power output and power fluctuations. We want to understand the influence of the properties of the very large-scale motions in the atmospheric boundary layer and the role of the turbulent fluctuations on the wind farm performance. Subsequently we want to translate this understanding to simpler models that can be used to predict properties of extended wind farms.

Progress
We developed large-eddy simulation methods to study the performance of wind farms in hilly terrain, which agrees excellently with measurement data. In particular, we find that the flow speeds when going over the hill, which is beneficial for turbines on the hilltop. In addition, the hill increases vertical kinetic energy entrainment downstream of the hill. This can increase the performance of turbines downstream, particularly in stable boundary layers when the energy of a low-level jet can be entrained. We also studied the effect of flow blockage effects on wind farm performance and the performance of wind farms with very large inter turbine spacings.

Visualization of the low speed wind regions, knowns as wind turbine wakes, in a very large wind farm. The intensity of blue cloud indicates the strength of the wind turbine wakes. Visualization by Srinidhi N. Gadde.

2020 Publications
M. Zhang, R.J.A.M. Stevens, Characterizing the coherent structures within and above large wind farms, Bound. Layer Meteorol. 174, 61-80 (2020)
L. Liu, R.J.A.M. Stevens, Effects of two-dimensional steep hills on the performance of wind turbines and wind farms, Bound.-Layer Meteorol. 176, 251-269 (2020)
L. Liu, R.J.A.M. Stevens, Wall modeled immersed boundary method for high Reynolds number flow over complex terrain, Computers and Fluids 208, 104604 (2020)
J.M.I. Strickland, R.J.A.M. Stevens, Effect of thrust coefficient on the flow blockage effects in closely-spaced spanwise-infinite turbine arrays, J. Phys. Conf. Ser., 1618, 062069 (2020)
R.J.A.M. Stevens, C. Meneveau, Large eddy simulation study of extended wind farms with large inter turbine spacing, J. Phys. Conf. Ser., 1618, 062011 (2020)
I. Neunaber, M. Hölling, R.J.A.M. Stevens, G. Schepers, J. Peinke, Distinct turbulent regions in the wake of a wind turbine and their inflow-dependent locations: The creation of a wake map, Energies 13, 5392 (2020)

UT - Science and Technology - Physics of Fluids

Project leaders
Detlef Lohse, Roberto Verzicco

Research theme
Complex dynamics of fluids

Participants
Detlef Lohse, Roberto Verzicco, Richard Stevens, Dominik Krug, Chong Shen Ng, Kai Leong Chong, Haoran Liu, Guiquan Wang, Alexander Blass, Pieter Berghout, Martin Assen, Robert Hartmann, Naoki Hori, Sreevanshu Yerragolam, Chris Howland, Rui Yang

Cooperation
E. Bodenschatz, O. Shishkina (Göttingen)
G. Ahlers (Santa Barbara)
H. Clercx, R. Kunnen, G. van Heijst, F. Toschi (Eindhoven)
I. Marusic, N. Hutchins, D. Chung (Melbourne)
R. Ostilla-Mónico (Houston)
Z. Wan (Hefei)
K. Xia (Shenzhen, Hong Kong)
X. Zhu (Göttingen)
Y. Yang (Beijing)

Funded by
ERC, FOM, NWO, MCEC, ERC, DFG

Funding %
NWO 80 %
EU 20 %

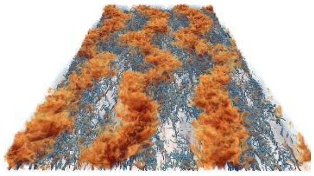
Start of the project
2015

Information
D. Lohse
poftnw@utwente.nl

Project title
Direct Numerical Simulations of Highly Turbulent Flows

Project aim
This Project aims to gain a deeper and broader insight into the dynamics of highly turbulent fluid flow. The research is carried out by means of high fidelity and massively parallel computer simulations. In particular we focus on Taylor-Couette turbulence, Rayleigh-Bénard convection, and Couette flow with unstable stratification. Furthermore, we investigate heat transfer in double diffusive convection systems.

Progress
We explored the different flow regimes in sheared thermal convection, i.e. the buoyancy dominated, the transitional, and the shear dominated regime. As a consequence of these different flow regimes, for fixed Ra and with increasing shear, the heat transfer first decreases, due to the breakup of the thermal rolls, and then increases at the beginning of the shear dominated regime. Furthermore, we showed that the number n of convection rolls, their mean aspect ratios Γ_r=Γ/n, and explored the transport properties of the flow, as function of the control parameters Rayleigh and Prandtl number. The effective scaling exponent β in $Nu \sim Ra^{\beta}$ is found to depend on the realized state and thus Γr, with a larger value for the smaller Γr,



Visualization by Alexander Blass of sheared thermal convection showing the formation of streamwise elongated rolls.

2020 Dissertations
- Alexander Blass, Sheared Rayleigh-Bénard Turbulence
- Qi Wang, Turbulent thermal convection: From Rayleigh-Bénard to vertical convection

2020 Publications
R. Yang, K.L. Chong, Q. Wang, R. Verzicco, O. Shishkina, and D. Lohse, Periodically Modulated Thermal Convection, Phys. Rev. Lett. 125, 154502 (2020)
Q. Wang, R. Verzicco, D. Lohse, and O. Shishkina, Multiple States in Turbulent Large-Aspect-Ratio Thermal Convection: What Determines the Number of Convection Rolls?, Phys. Rev. Lett. 125, 074501 (2020)
Y. Yang, W. Chen, R. Verzicco, and D. Lohse, Multiple states and transport properties of double-diffusive convection turbulence, Proc. Natl. Acad. Sci. USA 05669, 1–6 (2020)
P. Berghout, R. Verzicco, R.J.A.M. Stevens, D. Lohse, and D. Chung, Calculation of the mean velocity profile for strongly turbulent Taylor–Couette flow at arbitrary radius ratios, J. Fluid Mech. 905, A11 (2020)
Q. Wang, K.L. Chong, R.J.A.M. Stevens, R. Verzicco, and D. Lohse, From zonal flow to convection rolls in Rayleigh-Bénard convection with free-slip plates, J. Fluid Mech. 905, A21 (2020)
K.L. Chong, R. Yang, Q. Wang, R. Verzicco, and D. Lohse, Café latte: spontaneous layer formation in laterally cooled double diffusive convection, J. Fluid Mech. 100, R6 (2020)
Blass, X. Zhu, R. Verzicco, D. Lohse, and R.J.A.M. Stevens, Flow organization and heat transfer in turbulent wall sheared thermal convection, J. Fluid Mech. 897, A22 (2020)
S. Liu, L. Jiang, K.L. Chong, X. Zhu, Z. Wan, R. Verzicco, R.J.A.M. Stevens, D. Lohse, and C. Sun, From Rayleigh–Bénard convection to porous-media convection: how porosity affects heat transfer and flow structure, J. Fluid Mech. 895, A–18 (2020)
P. Berghout, R.J. Dingemans, X. Zhu, R. Verzicco, R.J.A.M. Stevens, W. van Saarloos, and D. Lohse, Direct numerical simulations of spiral Taylor–Couette turbulence, J. Fluid Mech. 887, A18–1–A18–16 (2020)
Y. Yang, R. Verzicco, D. Lohse, and R.J.A.M. Stevens, What rotation rate maximizes heat transport in rotating Rayleigh-Bénard convection with Prandtl number larger than one?, Phys. Rev. Fluids 5, 053501 (2020)
R.J.A.M. Stevens, D. Lohse, and R. Verzicco, Toward DNS of the Ultimate Regime of Rayleigh-Bénard Convection, Direct and Large Eddy Simulation XII 27, 215–224 (2020)

UT - Science and Technology - Physics of Fluids

Project leaders
Devaraj van der Meer, Detlef Lohse

Research theme
Complex dynamics of fluids

Participants
Utkarsh Jain

Cooperation
MARIN, NL

Funded by
SLING programme, STW Perspectief

Funding %
NWO 65%
Industry 35%

Start of the project
2016

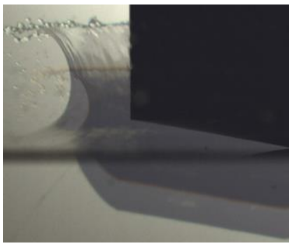
Information
D van der Meer
E-mail: d.vandermeer@utwente.nl

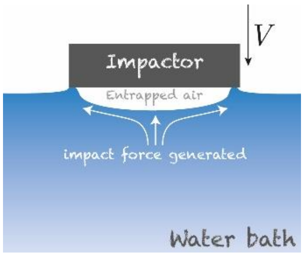
Project title
SLING (Sloshing of liquified natural gas)

Project aim
To understand the role of free surface instabilities in
- Sloshing of liquids when close to their boiling temperature
- Hydrodynamic loading on a solid structure

Progress
In any violent hydrodynamic slamming event in nature, the ambient air gets trapped in between the solid and the liquid of its own accord. This trapped air layer may serve to cushion the impact. This was studied in a lab experiment by slamming impactors such as discs, cones, wedges, and other shapes, onto a stationary water bath.

The water surface's response prior to contact could be measuring using the technique developed in earlier years. The pressures and forces on the impactor were also measured. In case of a flat plate slamming, a clear reduction in pressure-impulse was shown, which could be ascribed to the air cushioning effect. In cases impacting a finite deadrise angle cone, no air-trapping was found despite significant `air cushioning' effect which deflected the free surface away also under the cone. Using high fidelity sensors, a thorough study of Wagner slamming (and associated slamming models) pressures was conducted. All measurements of the free surface behaviour could be successfully reproduced using two-fluid boundary integral simulations.





UT - Science and Technology - Physics of Fluids

Project leaders
Alvaro Marin

Research theme
Complex structures of fluids

Participants
Mathieu Souzy
Iker Zuriguel
Edgar Ortega-Romero
Devaraj van der Meer
Stefan Luding

Cooperation
Iker Zuriguel (Universidad de Navarra, Spain)

Funded by
European Research Council
University of Twente

Funding %
University 50%
EU 50%

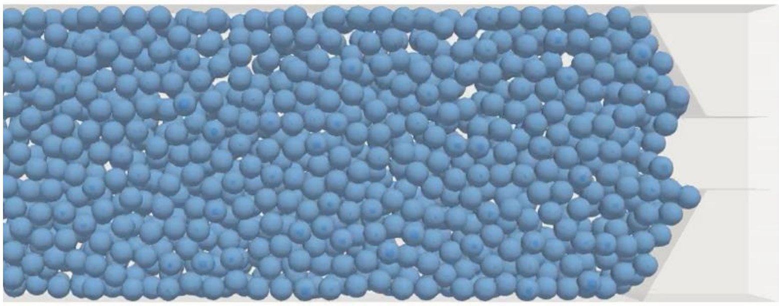
Start of the project
2019

Information
Alvaro Marin
a.marin@utwente.nl

Project title
Transitions from clogging to flowing in constricted suspensions

Project aim
When suspended particles are pushed by liquid flow through a constricted channel, they might either pass the bottleneck without trouble or encounter a permanent clog that will stop them forever. However, they may also flow intermittently with great sensitivity to the neck-to-particle size ratio. In this work, we experimentally explore the limits of the intermittent regime for a dense suspension through a single bottleneck as a function of this parameter. To this end, we make use of high time- and space-resolution experiments to obtain the distributions of arrest times between successive bursts, which display power-law tails with characteristic exponents.

Progress
The distribution of time arrests show clear exponents which surprisingly compare well with the ones found for as disparate situations as the evacuation of pedestrians from a room, the entry of a flock of sheep into a shed, or the discharge of particles from a silo. Nevertheless, the intrinsic properties of our system (i.e., channel geometry, driving and interaction forces, particle size distribution) seem to introduce a sharp transition from a clogged state to a continuous flow, where clogs do not develop at all. This contrasts with the results obtained in other systems where intermittent flow, with power-law exponents above two, were obtained. In order to understand this peculiar behaviour of suspensions, we are currently making use of discrete particle simulations using the software developed at the University of Twente MercuryDPM in order to reproduce our experimental results gain a better understanding of the crucial differences found with the discharge of dry particles in silos or the evacuation of pedestrians from a room, and very specially, the role of the fluid flow on the behaviour.



Discrete element simulation (MercuryDPM) of a suspension passing through a constriction (at the right side of the image) under constant liquid pressure. The current image does not show a permanent clog, but an intermittent one.

2020 Publications
Transition from clogging to continuous flow in constricted particle suspensions,
Mathieu Souzy , Iker Zuriguel and Alvaro Marin, Physical Review E 101, 060901(R) (2020)

UT - Science and Technology - Physics of Fluids

Project leaders
Alvaro Marin

Research theme
Complex structures of fluids

Participants
Myrthe Bruning
Laura Loeffen

Funded by
European Research Council

Funding %
EU 100%

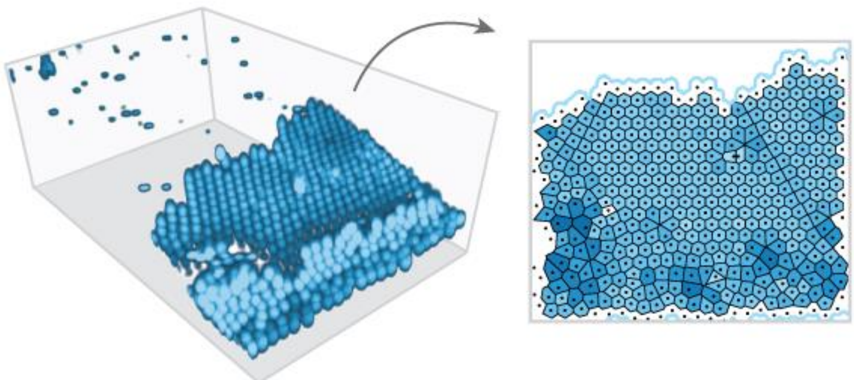
Start of the project
2019

Information
Alvaro Marin
a.marin@utwente.nl

Project title
Particle monolayer assembly in evaporating salty colloidal droplets

Project aim
While evaporating water droplets containing particles give rise to ring-shaped stains (the well-known "coffee-stain"), salt solutions develop a completely different flow structure due to the presence of strong Marangoni flows. Nonetheless, the resulting stain is also ring-shaped. In this work, we study particle-per-particle how the assembly of the colloids aggregate during the evaporation of droplets for different initial concentration of sodium chloride and initial particle dilution.

Progress
Our results show that the particle aggregation occurs in a monolayer along the liquid-air interface, a radically different structure as in the classical 'coffee-stain effect'. Since we have access to individual particles, we compare our results with classical diffusion-limited deposition models and open up an interesting scenario of deposits via interfacial particle assembly, which can easily yield homogeneous depositions by manipulating the initial salt and particle concentration in the droplet.



Left: three-dimensional reconstruction of the colloidal monolayer generated by Marangoni flow at the edge of an evaporating salty droplet. Right: projection of the monolayer with Voronoi cell analysis.

2020 Publications
Particle monolayer assembly in evaporating salty colloidal droplets, Myrthe A. Bruning, Laura Loeffen, and Alvaro Marin, Physical Review Fluids 5, 083603 (2020).

UT - Science and Technology - Physics of Fluids

<u>Project leaders</u>
Alvaro Marin

<u>Research theme</u>
Complex structures of fluids

<u>Participants</u>
Carola Seyfert
Erwin Berenschot
Niels Tas
Arturo Susarrey-Arce

<u>Funded by</u>
European Research Council

<u>Funding %</u>
EU 100%

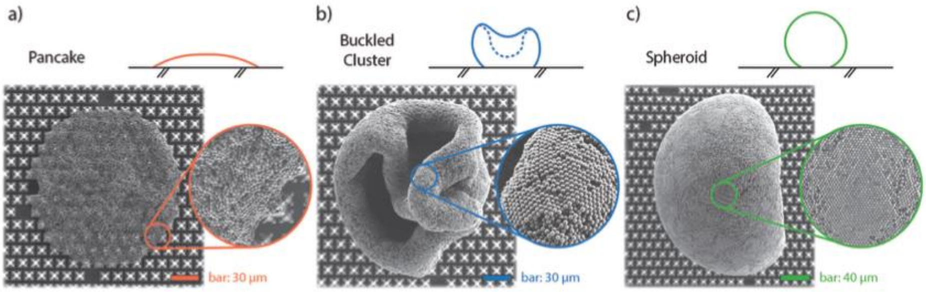
<u>Start of the project</u>
2019

<u>Information</u>
Alvaro Marin
a.marin@utwente.nl

<u>Project title</u>
Evaporation-driven colloidal cluster assembly using droplets on superhydrophobic fractal-like structures

<u>Project aim</u>
A large variety of three-dimensional colloidal macro-agglomerate clusters (''supra-particles'') can be obtained by systematically varying the initial particle concentration in an evaporating droplet on a superhydrophobic substrate.

<u>Progress</u>
Our results show a clear transition from quasi-2D to 3D clusters as a function of the initial particle concentration, and a clear transition from unstable to stable 3D spheroids as a function of the evaporation rate. The origin of such shape transitions can respectively be found in the dynamic wetting of the fractal-like structure, but also in the enhanced mechanical stability of the particle agglomerate as its particle packing fraction increases.



Overview of the three different categories of cluster shapes for increasing initial particle concentration from a) to c). All panels present a side view schematic, a top view SEM image of a typical cluster and a zoom-in view of the cluster surface. The colloidal particles (building blocks) are 0.98 µm in diameter.

<u>2020 Publications</u>
Evaporation-driven colloidal cluster assembly using droplets on superhydrophobic fractal-like structures. Carola Seyfert, Erwin J. W. Berenschot, Niels R. Tas, Arturo Susarrey-Arce and Alvaro Marin. Soft Matter 17, 506. (2020). DOI: 10.1039/d0sm01346c

UT - Science and Technology - Physics of Fluids

Project leaders
Devaraj van der Meer

Research theme
Complex dynamics of fluids

Participants
Bernardo Palacios Muñiz, Ellis Fan, Nayoung Kim

Cooperation
Marin

Funded by
NWO TTW (vici)

Funding %

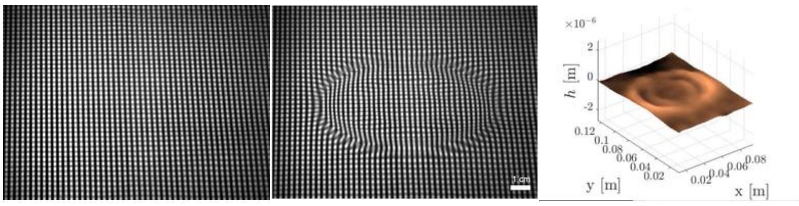
STW	95%
Industry	5%

Information
Devaraj van der Meer
d.vandermeer@utwente.nl

Project title
Impact of a boiling liquid

Project aim
The physics of the impact of a liquid onto a solid surface change dramatically when the liquid is at its boiling point and surrounded by its vapor. The impacting energy induces phase change of liquid, consequently, it generates microbubbles and cavitation. The aim of this project is to unravel the role of phase change during such boiling impact problems and identify the dominant contributions to forces exerted on this phenomenon.

Progress
1. Droplet A fully sealed chamber has built to study the impact of boiling droplets over a slightly heated surface. Instrumentation and data acquisition of the setup is undergoing testing
2. Disk impact: A sealed and thermally insulated chamber with temperature and pressure control is being built. Progress is being made on improving the TIR deflectometry measurement method and adapting its setup for the study on pre-impact gas cushioning effects during boiling liquid impact.



(a) (b)

Total internal reflection (TIR) deflectometry: (a) deformation of a reference pattern due to interface deflection and (b) reconstructed 3D liquid surface.

Image sequence of a droplet impacting the substrate at low pressure conditions

UT - Science and Technology - Physics of Fluids

Project leaders
Michel Versluis
Guillaume Lajoinie
Tim Segers

Research theme
Complex dynamics of fluids

Participants
Nathan Blanken, Sarah Cleve, Benjamin van Elburg, Charlotte Nawijn, Erik Groot-Jebbink, Jelmer Wolterink, Christoph Brune

Cooperation
University of Ghent (Belgium)
Bracco (Geneva, CH)
TU Delft
UMC Utrecht
Oxford University (UK)
INRIA Institute (France)
University Hospital of Nice (France)
NTNU Trondheim

Funded by
Univ Twente
4TU Precision Medicine
NWO
EU
Connecting Industry TKI

Funding %

University	20 %
NWO	70%
Industry	10 %

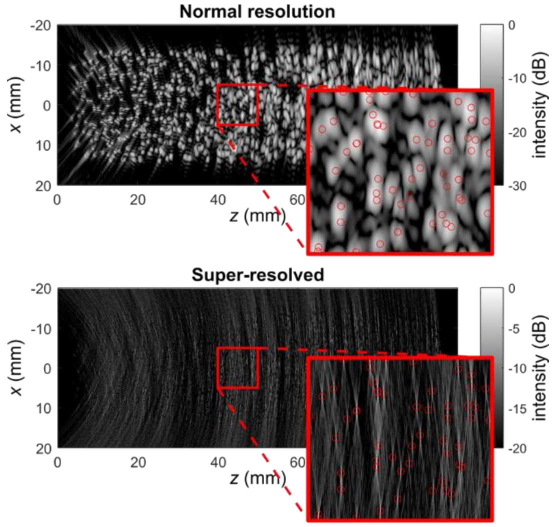
Start of the project
2019

Information
Michel Versluis
m.versluis@utwente.nl

Project title
Ultrasound imaging and nonlinear scattering by microbubbles

Project aim
This Project aims at understanding, on a fundamental level, the physics of monodisperse microbubble contrast agents. This includes their production, composition and their response to medical ultrasound.

Progress
We have achieved new understanding in the role of the bubble shell on the bubble response. We have demonstrated the feasibility of using the nonlinear bubble response to an incoming signal to measure the ambient pressure, both analytically and using deep-learning. We have furthermore used deep-learning for super-resolution imaging with microbubbles. We have achieved new understanding in the underlaying physical mechanisms in a flow-focusing device for the production of monodisperse microbubbles and we have finalized a device allowing for feedback-controlled production. These monodisperse bubbles were further used for drug-delivery, in vitro, to demonstrate the shortcomings in our current understanding of the bioeffects of shear.



Super-resolved ultrasound contrast image obtained using deep-learning.

2020 Publications
- Eleanor Stride, Tim Segers, Guillaume Lajoinie, Samir Cherkaoui, Thierry Bettinger, Michel Versluis, Mark Borden. Microbubble Agents: New Directions (review). Ultrasound Med. Biol. 46, 1326–1343 (2020).
- Tim Segers, Emmanuel Gaud, Gilles Casqueiro, Anne Lassus, Michel Versluis, and Peter Frinking. Foam-free monodisperse lipid-coated ultrasound contrast agent synthesis by flow-focusing through multi-gas-component microbubble stabilization. Appl. Phys. Lett. 116, 173701 (2020).
- Michel Versluis, Eleanor Stride, Guillaume Lajoinie, Benjamin Dollet, and Tim Segers. Ultrasound contrast agents modeling: a review. Ultrasound Med. Biol. 46, 2117–2144 (2020).

UT - Science and Technology - Physics of Fluids

<u>Project leaders</u>
Detlef Lohse

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Detlef Lohse, Michel Versluis, Jacco Snoeijer, Tim Segers, Christian Diddens, Yaxing Li, Michiel Hack, Maaike Rump, Yogesh Jethani, Uddalok Sen, Hans Reinten, Herman Wijshoff, Marc van den Berg, Youri de Loore

<u>Cooperation</u>
University of Twente, Technical University of Eindhoven, Canon Production Printing

<u>Funded by</u>
NWO and Canon Production Printing

<u>Funding %</u>

University	10%
NWO	45%
Industry	45%

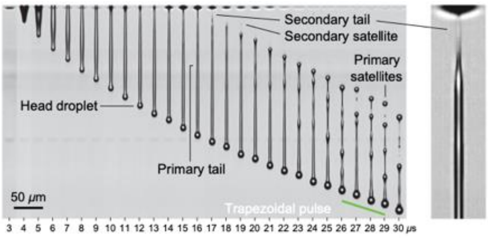
<u>Start of the project</u>
2017

<u>Information</u>
D. Lohse
poftnw@utwente.nl

<u>Project title</u>
Fundamental Fluid Dynamics Challenges in Inkjet Printing (FIP)

<u>Project aim</u>
The main scientific goal of the program is to create insight in unresolved issues in the current inkjet printing processes and to improve and extend the functionality of inkjet printing to meet future requirements. The functional modeling of the inkjet printing process not only concerns the numerical simulations but also the physical theory, which explains the results, and the experiments, which validate the results. The topics investigated range from piezo actuators, printhead dynamics, and jetting of complex liquids to droplet evaporation and absorption of ink in porous media.

<u>Progress</u>
In addition to the principal research areas of the project, a lot of related side-projects are involved. We have published papers on a wide variety of topics related to inkjet printing. It has been revealed that the formation of secondary tails (see the figure) during inkjet printing is highly reproducible, and the perturbation leading to such tail formation is not dominated by noise, but by meniscus motion. On the evaporation side, it has been shown that segregation in a bicomponent evaporating droplet drives a Rayleigh-Taylor instability, resulting in the formation of plumes.



<u>2020 Dissertations</u>
Yaxing Li, "Evaporating multicomponent droplets", University of Twente (2020).

<u>2020 Publications</u>
- M. A. Hack, W. Tewes, Q. Xie, C. Datt, K. Harth, J. Harting, J. H. Snoeijer, "Self-similar liquid lens coalescence", Phys. Rev. Lett. 124, 194502 (2020).
- Fraters, R. Jeurissen, M. van den Berg, H. Reinten, H. Wijshoff, D. Lohse, M. Versluis, and T. Segers, "Secondary tail formation and breakup in piezoacoustic inkjet printing: femtoliter droplets captured in flight", Phys. Rev. Appl. 13, 024075 (2020).
- Y. Li, C. Diddens, T. Segers, H. Wijshoff, M. Versluis, and D. Lohse, "Rayleigh-Taylor instability by segregation in an evaporating multicomponent droplet", J. Fluid Mech. 899, A22 (2020).
- Y. Li, V. Salvator, H. Wijshoff, M. Versluis, and D. Lohse, "Evaporation-induced crystallization of surfactants in sessile multicomponent droplets", Langmuir 36, 7545-7552 (2020).
- R. T. van Gaalen, C. Diddens, H. M. A. Wijshoff, and J. G. M. Kuerten, "The evaporation of surfactant-laden droplets: a comparison between contact-line models", J. Colloid Interface Sci. 579, 888-897 (2020).
- Y. Li, C. Diddens, T. Segers, H. Wijshoff, M. Versluis, and D. Lohse, "Evaporating droplets on oil-wetted surfaces: suppression of the coffee-stain effect", Proc. Natl. Acad. Sci. USA 117(29), 16756-16763 (2020).
- M.-J. van der Meulen, H. Reinten, H. Wijshoff, M. Versluis, D. Lohse, and P. Steen, "Nonaxisymmetric effects in drop-on-demand piezoacoustic inkjet printing", Phys. Rev. Appl. 13, 054071 (2020).

UT - Science and Technology - Physics of Fluids

Project leaders
M Versluis
M Reijnen

Research theme
Complex dynamics of fluids

Participants
H Mirgolbabaee
M van Helvert
L van de Velde
S Engelhard
G Lajoinie
E Groot Jebbink

Cooperation
Rijnstate Hospital Arnhem
Erasmus Medical Center
Radboudumc

Funded by
University
Industry
NWO

Funding %
University 20%
STW 30%
Industry 50%

Start of the project
2013

Information
Erik Groot Jebbink
e.grootjebbink@utwente.nl

Project title
Blood flow and endovascular stenting, comparative in-vivo and in-vitro studies

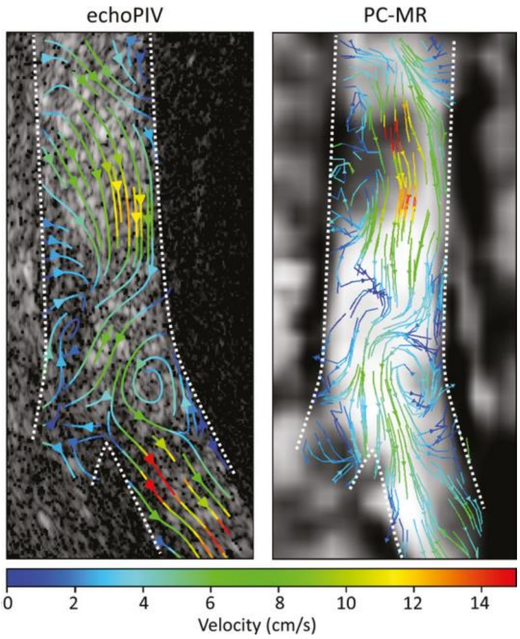
Project aim
A large number of clinical studies report about the performance of endovascular stent placement. However, no tools are available to measure and report about the important local interaction between blood flow and stents. In order to improve the outcome of treatment a more profound insight in local hemodynamics is desirable, which can augment treatment planning and follow-up, as well as reduce the rate of reinterventions. The goal of this project is to combine observations of flow dynamics in patients with the results from in vitro studies, to gain insight in fluid mechanical mechanisms that are relevant for clinical planning and surveillance.

Progress
The first clinical trials have been completed, contrast enhanced high framerate (>1000) plane wave measurements were performed in 20 and 35 patients after endovascular treatment of stenotic lesions. The manuscripts have been submitted. A third trial is still ongoing, furthermore a fourth trial in aneurysm patients is scheduled.

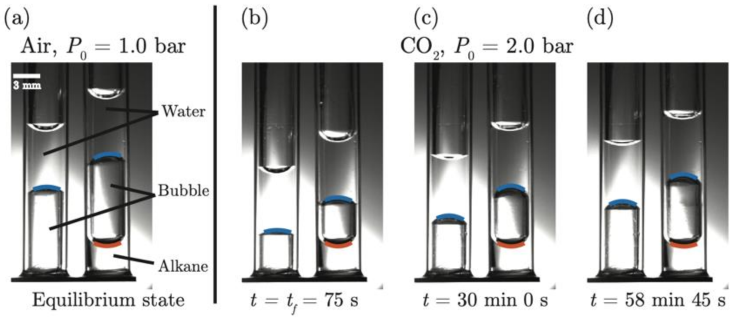
In vitro models are being constructed to investigate limb occlusions after EVAR and to optimize our PIV methods. In addition a new line of research was initiated concerning the role of local hemodynamics during radio-embolization of liver tumors, for which Erik Groot Jebbink obtained a Veni grant and a NWO TTW-OTP project was awarded.

2020 Publications
Vos, H. J., Voorneveld, J. D., Groot Jebbink, E., Leow, C. H., Nie, L., van den Bosch, A. E., & Bosch, J. G. (2020). Contrast-Enhanced High-Frame-Rate Ultrasound Imaging of Flow Patterns in Cardiac Chambers and Deep Vessels. *Ultrasound in Medicine & Biology*.

Chong, A., Sun, Z., van de Velde, L., Jansen, S., Versluis, M., Reijnen, M. M., & Groot Jebbink, E. (2020). A novel roller pump for physiological flow. *Artificial organs*, *44*(8), 818-826.



Streamline representation of blood flow velocities during early diastole. Similar flow patterns can be observed in both data sets, including a slow (counter clockwise) recirculation zone near the origin of the left common iliac artery. This recirculation zone occurred during a longer time period in the phase-contrast MRI (PC-MR) data (five of 30 phases) than in the US particle image velocimetry (echoPIV) data (10–15 msec). Dashed lines show estimated delineation of the vessel wall.

UT - Science and Technology - Physics of Fluids	
Project leaders Detlef Lohse, Devaraj van der Meer Research theme Complex dynamics of fluids Participants Pablo Peñas, Daniël Faasen Funded by MCEC, NWO Zwaartekracht Funding % NWO 100% Start of the project 2019 Information D. Lohse E-mail: d.lohse@utwente.nl	Project title Slug bubble growth and dissolution by solute exchange Project aim We study the growth dynamics of a trapped slug bubble in a vertical glass cylinder under a water barrier. We replace the ambient air atmosphere by a CO_2 atmosphere at the same or higher pressure. The asymmetric exchange of the gaseous solutes between the CO_2 rich water barrier and the air-rich bubble always results in net bubble growth, a process we refer to as solute exchange. The dominant transport of CO_2 across the water barrier is driven by a combination of diffusion and convective dissolution. Progress 1. We studied the growth dynamics of the aforementioned system both experimentally and theoretically. 2. A scaling relation between the Sherwood and Rayleigh number was determined and found to be in agreement with the Grossmann-Lohse theory. 3. For ternary water–bubble–alkane systems, the inclusion of the alkane layer bestows a buffering (hindering) effect on the bubble growth and dissolution. 

UT - Science and Technology - Physics of Fluids

<u>Project leaders</u>
Devaraj van der Meer, Detlef Lohse

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Pablo Peñas

<u>Funded by</u>
MCEC, NWO Zwaartekracht

<u>Funding %</u>
NWO 100%

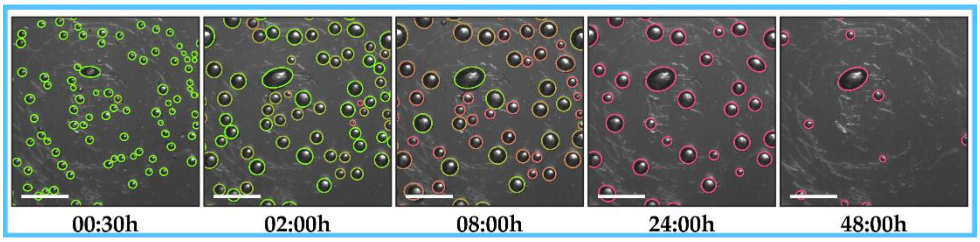
<u>Start of the project</u>
2020

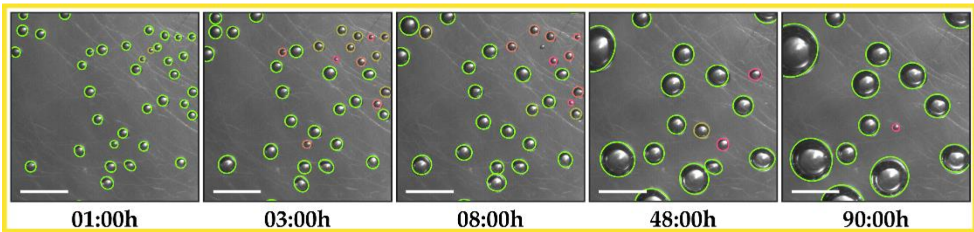
<u>Information</u>
Name D. van der Meer
E-mail: d.vandermeer@utwente.nl

<u>Project title</u>
Collective growth of air bubbles in water and water + oil systems

<u>Project aim</u>
Air–vapor bubbles growing on the surface of a glass of cold water constitute a rich system laden with complex physicochemical effects involving mass and heat transfer. These include bubble nucleation, growth and dissolution, coalescence, Ostwald ripening and water evaporation. We wish to understand how the temperature-driven growth of bubbles in water is affected by collective interactions, ambient humidity and the presence of an oil layer.

<u>Progress</u>
Our experiments comprise a scratched polystyrene Petri dish that is filled with a thin layer of cold water or with water and covered by a layer of n-hexadecane (oil) of similar thickness. Bubbles form and grow on the scratches. Collective effects are found to be most important during the fast (supersaturation-driven) growth stage; the subsequent dissolution stage is driven by surface tension. In the pure water system (top row), all bubbles eventually dissolve. In the water + oil system (bottom row), we find indefinite bubble growth above a critical size. This phenomenon is attributed to the difference in water vapor pressure between the humid bubble interior and the drier ambient air.





UT – Science and Technology - Physics of Complex Fluids

Prof.dr. F Mugele

The goal of the PCF group is to understand and control the structure and the mechanical properties of liquids and interfaces on length scales ranging from molecular to submillimetre scales. The activities fall in three main categories: i) nanofluidics, ii) (electro)wetting & microfluidics, iii) soft matter mechanics. Our nanofluidics research focuses on understanding the range of validity of macroscopic continuum physics and in its breakdown upon approaching molecular scales, where physico-chemical aspects become increasingly important. In microfluidics, many properties of fluids, in particular drops, are controlled by interfacial effects. By patterning surfaces physic and in particular by making use the electrowetting effect we control the shape, the motion, and the generation of microdrops. These processes involve various challenging fundamental issues, such as contact angle hysteresis, the dynamics of contact lines, and hydrodynamic singularities. The soft matter mechanics activities focus on correlations between the internal structure of various types of complex fluids ranging from colloidal suspensions to living cells and their macroscopic viscous and elastic properties.

By improving the physical understanding of fundamental phenomena we contribute to the improvement of various technological processes involving fluid motion on small scales, including oil recovery, immersion lithography, and inkjet printing. This work is frequently carried out in collaboration with industrial partners including BP, Shell, ASML, Océ, Liquavista, sometimes within government sponsored consortia such as FOM-IPPs, sometimes in direct collaboration. A major project on enhanced oil recovery started in late 2009 and became fully operational in 2010. In this context, the group intensified its activities in the area of physical chemistry of liquid-liquid and solid-liquid interfaces. In late 2010, Prof. Mugele obtained a NWO-VICI grant to investigate the properties of superhydrophobic surfaces that are functionalized by electric fields. One major goal of the project is to explore various applications of such smart surfaces for microfluidics, ultrasound detection, and in particular optofluidics.

UT – Science and Technology - Physics of Complex Fluids

Project leaders
Prof. Dr. Frieder Mugele

Research theme
Complex structures of fluids

Participants
Carla Annink; Ashit Rao, Frank Megens, Saravana Kumar, Duy Le-Anh, Igor Siretanu, Michel Duits, Frieder Mugele

Cooperation
Saudi Aramco

Funded by
Saudi Aramco

Funding %
Industry 100%

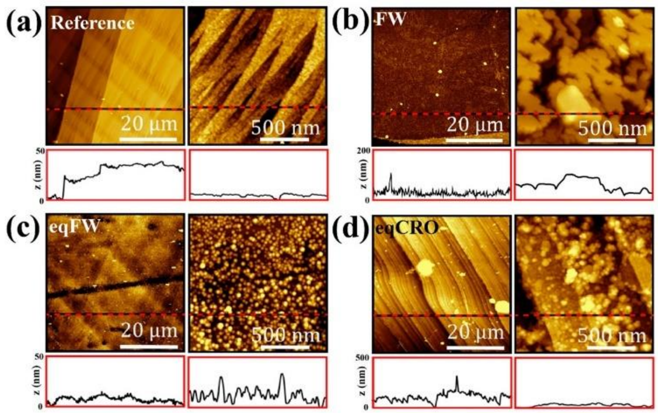
Start of the project
2017

Information
Frieder Mugele
f.mugele@utwente.nl

Project title
Wettability alteration and two-phase flow in carbonate reservoirs

Project aim
The purpose is to understand the effect of variable salt concentration, so-called 'smart brines', on the efficiency of oil recovery in carbonate reservoirs. The project focuses on the fundamental physical chemistry of model calcite-electrolyte interfaces exposed to crude oil as well as brines of variable composition. A suite of techniques is applied to study the system including macroscopic contact angle goniometry, microfluidic chips (micromodels), confocal Raman microscopy, and atomic force microscopy and spectroscopy.

Progress
The focus in 2020 was on the characterization of calcite surfaces as they were exposed to brines of variable composition and as well as crude oil according to well-defined protocols between room temperature and 95°C for several days. Morphology and chemical composition of the calcite surfaces were shown to vary mainly for two reasons: a) dissolution and reprecipitation of calcite in amorphous as well as crystalline form; b) co-deposition of organic material as well as mixed organo-carbonate material. Both have important consequences for the wettability of the surfaces.



2020 Publications
Ashit Rao, Saravana Kumar, Carla Annink, Duy Le-Anh, Subhash C. Ayirala, Mohammed B. Alotaibi, Igor Siretanu, Michel H.G. Duits, Ali A. Yousef, and Frieder Mugele, Mineral Interfaces and Oil Recovery: A Microscopic View on Surface Reconstruction, Organic Modification, and Wettability Alteration of Carbonates, Energy&Fuels 34, 5611 (2020)
Ashit Rao, Saravana Kumar, Carla Annink, Duy Le-Anh, Mohammed B. Alotaibi, Subhash C. Ayirala, Igor Siretanu, Michel Duits, Frieder Mugele and Ali A. Yousef, Artificial Diagenesis of Carbonates: Temperature Dependent Inorganic andOrganic Modifications in Reservoir Mimetic Fluids,Society of Petroleum Engineering (SPE) J. 26, 3222 (2020)

UT – Science and Technology - Physics of Complex Fluids

Project leaders
dr. Frieder Mugele

Research theme
Complex dynamics of fluids

Participants
dr. Michael H.G. Duits
dr. Cees Otto
Sachin Nair

Funded by
NWO & BP plc. Project number i40

Funding %
NWO 50%
Industry 50%

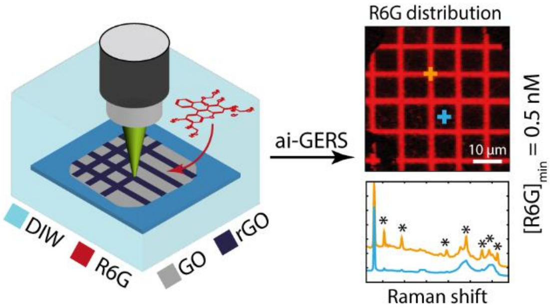
Start of the project
2017

Information
Frieder Mugele
f.mugele@utwente.nl

Project title
Distribution of inorganic and organic matter at solid-electrolyte interfaces studied by Confocal Raman Microscopy

Project aim
The goal of this project is to provide a chemical distribution map of specific adsorption-desorption phenomena using confocal Raman imaging. Some of the investigated systems include a) development of an efficient denoising technique based on principal component analysis for fast and non-invasive Raman imaging of photosensitive systems b) monitoring the release and reorganization of a surfactant coated on a mineral substrate in response to a lowered ambient brine salinity and (c) monitoring the adsorption of analyte molecules on 2D materials and characterizing their spatial distribution, using chemical enhanced Raman imaging.

Progress
Using the enhanced signal-to-noise ratio achieved by the algorithm-improved confocal Raman microscopy (ai-CRM) technique developed by us, we focus on improving the limit of detection (LOD) of fluorescent analyte molecules like Rhodamine-6g (R6G) adsorbing onto novel laser-reduced graphene-oxide flakes. We report an improved sub-nM LOD for R6G on different graphene oxide analogues. We also use in-situ Raman imaging to monitor the spatial distribution of the adsorbed R6G and probe the adsorption kinetics of R6G. Finally we show the application of our novel substrate to detect trace carcinogenic dyes in adulterated fruit juice.



2020 Publications
Nair, S., Gao, J., Otto, C., Duits, M. H., & Mugele, F. (2020). In-situ observation of reactive wettability alteration using algorithm-improved confocal Raman microscopy. Journal of colloid and interface science, 584, 551-560.

UT – Science and Technology - Physics of Complex Fluids

Project leaders
Frieder Mugele

Research theme
Complex dynamics of fluids

Participants
Frieder Mugele
Hao Wu
Niels Mendel

Cooperation
South China Normal University

Funded by
Scholarship

Funding %
Scholarships 100%

Start of the project
2016

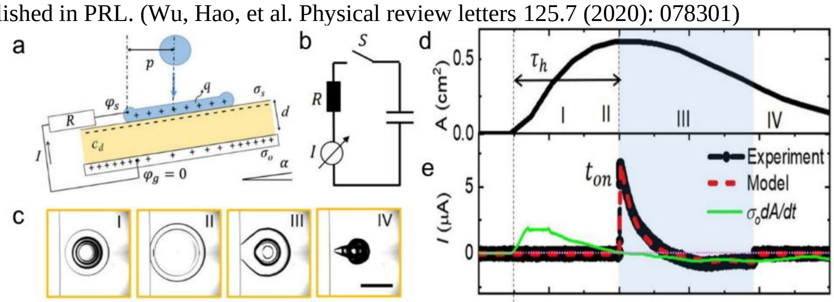
Information
Frieder Mugele
f.mugele@utwente.nl

Project title
Electrically responsive fluoropolymer surfaces and applications (Energy harvesting from water droplet impact onto charged surfaces)

Project aim
1. Investigate the electrowetting phenomenon on fluoropolymer surfaces
2. Investigate the charge trapping phenomenon of fluoropolymer films
3. Investigate energy harvesting from water droplet impact onto charged surfaces

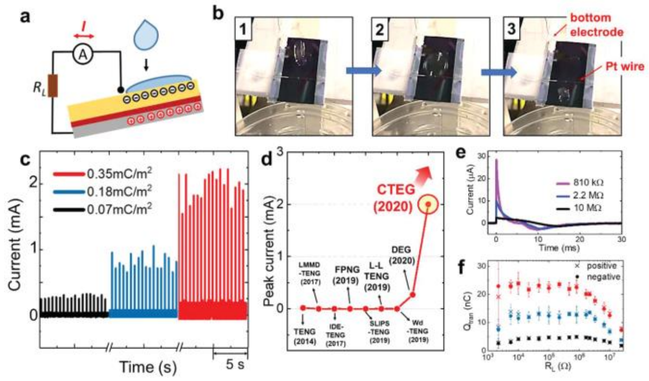
Progress
1. Invent the droplet energy harvester. A Chinese patent have been granted. (CN 201910876944.8, filed in 2019, granted in 2021)

2. Figure out the working mechanism of droplet-based electricity generator. Results have been published in PRL. (Wu, Hao, et al. Physical review letters 125.7 (2020): 078301)



Mechanism of droplet-based electricity generator.

3. Achieve high energy harvesting efficiency from water impact on hydrophobic surfaces with “electrowetting-induced charge injecton method.” Results have been published in AM (Wu, Hao, et al. Advanced Materials 32.33 (2020): 2001699) and AFM (Mendel, Niels, et al. Advanced Function Materials 31.11 (2021): 2007872)



Achieving high energy harvesting efficiency.

2020 Dissertations
Hao Wu, (2020, March 4) “Electrically responsive fluoropolymer surfaces and applications”
Supervisor: Frieder Mugele.

2020 Publications
Wu, Hao, et al. Small 16.2 (2020): 1905726.
Wu, Hao, et al. Physical review letters 125.7 (2020): 078301)
Wu, Hao, et al. Advanced Materials 32.33 (2020): 2001699

UT – Science and Technology – Soft Matter, Fluidics and Interfaces

Prof.dr.ir. RGH Lammertink

Research within the Soft matter, Fluidics and Interfaces group is directed at interfacial phenomena and processes that are relevant for mass and heat transport. We wish to study and exploit fundamental principles where fluid flow encounters structures on a sub-millimetre length scale.

Current topics of interest are:

Advanced microreactors
The fabrication and operation of dedicated microreactors, amendable to scaling are investigated. Multiphase reactor systems that incorporate membrane functionality to stabilize interfaces and perform separations are developed.

Soft interfaces
Liquid-liquid and gas-liquid interfaces are crucial in many chemical processes. Interfacial phenomena, including wetting behaviour, interfacial tension (gradients), interfacial curvature, are studied to gain understanding in related transport processes near these interfaces.

Micro- and nanofluidics
This topic addresses liquid flow in confined geometries. Its relation to mass and energy transport are studied in both experimental and numerical ways. Special attention is given to boundary layer and concentration polarization phenomena.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Dr. J.A. Wood, Prof.dr.ir. R.G.H. Lammertink Research theme Complex dynamics of fluids Participants Harmen Zwijnenberg Funded by TKI-HTSM Funded % University 35 % NWO 65% Start of the project 2019 Information Rob Lammertink r.g.h.lammertink@utwente.nl	Project title R2D2: Resource Recovery with Donnan Dialysis Project aim Selective recovery of valuable minerals from 'waste' streams such as wastewater represents an opportunity to enhance the sustainability of chemical processes. Membrane technology using ion-exchange membranes allows for transport of ions against bulk concentration gradients and for concentrating ions of interest. We will investigate the selective ion-recovery using different membrane geometries, operating conditions, ion-mixture composition under various process configurations. Progress The mass transport trough ion exchange membranes has been investigated for different systems. Using different salt solutions the ionic transport rate and equilibrium concentrations were investigated. Increasing the ion ratio and flow velocity determined to a strong extent the removal rate of target ions. The basic process has been modelled using Comsol and MatLab models to relate configurational changes to the overall ion transport.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces

Project leaders
Prof.dr.ir. R.G.H. Lammertink, Dr. J.A. Wood

Research theme
Complex dynamics of fluids

Participants
Abimbola Ashaju

Funded by
NWO

Funded %
NWO 100%

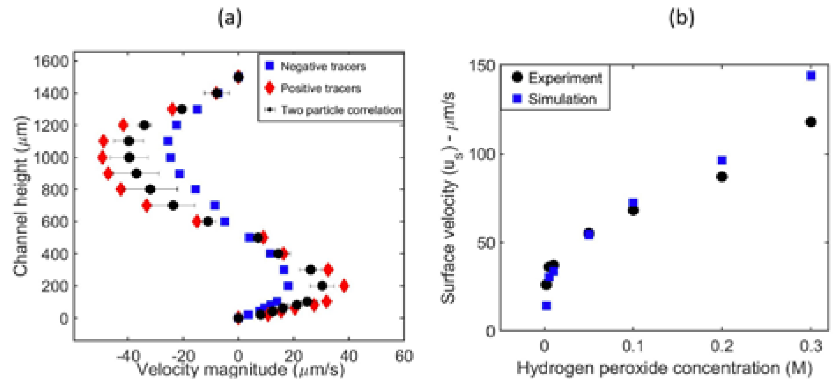
Start of the project
2017

Information
Rob Lammertink
r.g.h.lammertink@utwente.nl

Project title
Stirring the boundary layer

Project aim
PhD research: This thesis investigates the fundamental parameters that drive the an electrocatalytic reaction driven flow powered by the decomposition of hydrogen peroxide with an immobilized platinum and gold bimetallic system, through a combined experimental and numerical approach. The electrocatalytic current between the electrodes and the induced potential that governs the reactive fluxes are measured electrochemically, the reaction-induced proton concentration gradient that originates from the electrocatalytic reaction is spatially mapped using fluorescence lifetime imaging, while the fluid flow is visualized and quantified with two-particle phoresis correlation.

Progress
A single nanorod consists of a bimetallic duo that catalyzes the decomposition of a fuel via oxidation and reduction pathways, thereby creating the necessary gradients (concentration, potential) for its motion (figure a) and when its motility is restricted a fluid flow is generated (figure b). During the electrochemical reaction at the two metals (e.g. platinum and gold) with hydrogen peroxide solution, oxidation occurs at the anode (platinum) yielding protons, electrons and oxygen, while reduction takes place at the gold cathode where protons and electrons are consumed with water produced as a by-product (figure b). The resulting ion and electron flow generates an electric field that is coupled with the local non-neutral fluid, to create a body force that drives interfacial fluid motion, whose magnitude can be impacted by H2O2 concentration.



(a) Two particle correlation velocimetry used in determining the flow velocity, for 0.3 M H2O2, pH 6. The particle velocities are plotted as a function of the channel height. Flow near the surface of the bielectrode is driven from Pt to Au while the opposite movement is obtained at the upper part of the channel due to fluid continuity. (b) Comparison of experimentally estimated surface fluid velocity with simulation result as a function of H2O2 concentration.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Rob Lammertink Research theme Complex dynamics of fluids Participants Nicole Timmerhuis Funded by NWO Funded % NWO 100% Start of the project 2018 Information Rob Lammertink r.g.h.lammertink@utwente.nl	Project title Vici – Stirring the boundary layer Project aim Diffusio-osmotic flow is a surface induced velocity which origins from concentration gradients parallel to an interface. These concentration gradients are commonly salt gradients and the flow magnitude depends on the relative concentration gradient and a mobility factor. In this project we aim to study diffusio-osmotic flow induced by chemical reactions, creating an transient concentration gradient. Progress In the previous years, experimental setups were built and parameters of influence on the results were studied. The main focus has been on the reaction kinetics, finding analysis methods and designing a setup to quantify diffusion-osmotic flow. Whilst studying the reaction kinetics, an article was written about mass transfer limitations in a wall-coated photocatalytic microreactor. The limitations were estimated based on the first order reaction kinetics, dimensions of the reactor and superficial velocity of the flow.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces

Project leaders
Prof.dr.ir. R.G.H. Lammertink
Prof.dr.ir. C.G.P.H. Schroën

Research theme
Complex dynamics of fluids

Participants
Izabella Bouhid de Aguiar

Funded by
University of Twente

Funded %
University 100%

Start of the project
2020

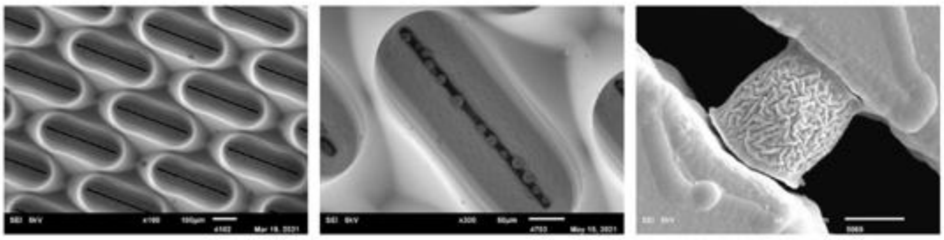
Information
Rob Lammertink
r.g.h.lammertink@utwente.nl

Project title
Behaviour of soft particles during filtration studies through model systems

Project aim
Soft particles are present in many industrial membrane filtration processes and bring additional challenges to separation in comparison with hard particles. Soft particles are deformable and compressible, what makes them susceptible to changes during processing in response to applied external forces such as pressure. Fundamental understanding of the behaviour of soft particles during membrane filtration is essential for development of innovative processes, their optimization, making them less sensitive to fouling, and consequently needing less intensive cleaning cycles.

Progress
From our observations using microfluidic devices we found that soft microgels are able to go through pores smaller than their diameter by deforming and deswelling. These changes were dependent on particle size, applied pressure and pore design. We also have performed filtration experiments using microgels and metal microsieves as the filtration medium (see the figure). We have found that the microgels are also capable of changing their conformation to go through the openings of the microsieves; the configuration of the microsieves is directly related to filtration properties as observed in the microfluidic experiments.

2020 Publications
Publication title: Microfluidics Used as a Tool to Understand and Optimize Membrane Filtration Processes. Journal: Membranes. Publication date: 29th October 2020.
https://doi.org/10.3390/membranes10110316



Scanning electron microscopy images of microsieves before (left) and after (middle and right) filtration of microgel suspension

UT – Science and Technology – Soft Matter, Fluidics and Interfaces

Project leaders
Prof.dr.ir. R.G.H. Lammertink

Research theme
Complex dynamics of fluids

Participants
Shuyana Heredia Deba

Funded by
Wetsus Leeuwarden

Funded %
University 40%
Industry 60%

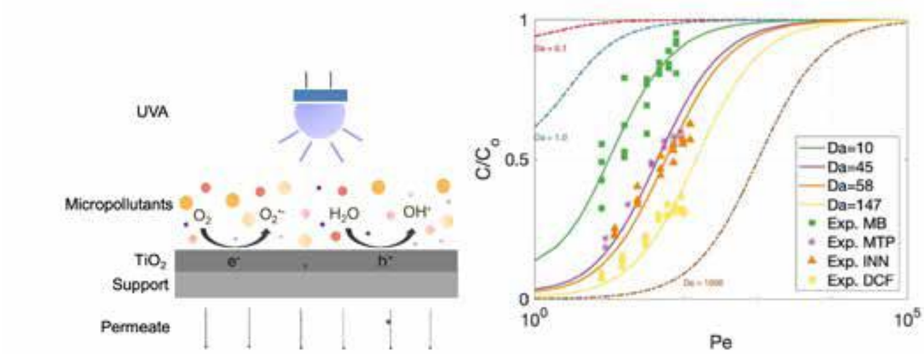
Start of the project
2017

Information
Rob Lammertink
r.g.h.lammertink@utwente.nl

Project title
Photocatalytic membranes for water purification

Project aim
This research seeks to elucidate the synergy between membrane separation and photocatalytic oxidation. The aim is to combine membrane and catalyst functionality within a single material. The beneficial synergy expected from having these two functions present at a single location will be studied experimentally and with a transport model. A reactive membrane is expected to reduce the concentration polarization and biofouling layers via the chemical conversion of reactants. Furthermore, photocatalytic degradation of contaminants in water is considered a viable method to remove micropollutants and inactivate viruses.

Progress
Photocatalytically active membranes were fabricated and tested to refine a 1D transport model. The combination of membrane filtration and photocatalytic oxidation showed a remarkable reduction of the concentration polarization. The model contains the membrane function (rejection, □) and the photocatalytic degradation (reaction, Da) including light distribution at different filtration rates (Pe). Different micropollutants were tested individually: methylene blue (MB), metoprolol (MTP), iopamidol (INN), and diclofenac (DCF).



<table>
<tr><th colspan="2">UT – Science and Technology – Soft Matter, Fluidics and Interfaces</th></tr>
<tr><td>

<u>Project leaders</u>
Prof.dr.ir. R.G.H. Lammertink, Dr. J.A. Wood

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Vanda Liadinskaia

<u>Funded by</u>
ARLA Foods, Dairy Industries International

<u>Funded %</u>
Industry 100%

<u>Start of the project</u>
2020

<u>Information</u>
Rob Lammertink
r.g.h.lammertink@utwente.nl

</td><td>

<u>Project title</u>
ARLA project - 3-year post-doctoral fellowship interfacial aspects of whey protein separation using artificial

<u>Project aim</u>
Postdoc research. In the ARLA project a combination of experimental and theoretical studies will be performed that will analyze and determine the underlying dominant mechanisms controlling transport. The intended outcome of this project is to possess a higher degree of insight into these mechanisms and to be able to then utilize this understanding in order to enhance separation efficiency in realistic industrial processes. These goals will be accomplished via a combination of experimental and theoretical investigations of various systems of interest, as described by ARLA Foods, Dairy Industries International.

<u>Progress</u>
During the 1st year work on the project, several milestones were achieved. A comprehensive analysis of relevant scientific literature was conducted, which allowed to design, manufacture, and assembly a special lab scale filtration setup required to perform necessary bench top experimental work. Preliminary results were obtained in order to optimize the experimental design for the next project phase. UHPLC analytical protocol was developed for protein content determination. Optical Coherence Tomography was successfully implemented as an alternative noninvasive technique for membrane fouling monitoring and visualization.

</td></tr>
</table>

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Prof.dr.ir. R.G.H. Lammertink Research theme Complex dynamics of fluids Participants Moritz Junker Funded by TKI HTSM, Oasen, NX Filtration Funded % NWO 75 % Industry 25% Start of the project 2019 Information Rob Lammertink r.g.h.lammertink@utwente.nl	Project title Micro-pollutant removal by salt permeating polyelectrolyte multilayer based nanofiltration membranes Project aim The rising concentration of aquatic contaminants also referred to as micropollutants, in surface and drinking water is a current environmental issue of concern. Since current waste-water treatment technologies are not capable of retaining these molecules, advanced separation technologies need to be developed. One promising technique is nanofiltration. Applying the novel Layerby- Layer method to make Polyelectrolyte Multilayer membranes, allows for finetuning of membrane properties. The goal of this project is to gain a fundamental understanding of transport processes inside these novel membranes to predict and optimize the filtration process. Progress A transport model for a Polyelectrolyte Multilayer nanofiltration membrane was implemented based on the extended Nernst-Planck equation. This model takes into account potential inhomogeneous charge distributions throughout the separation layer. The model was extended to account for distribution in pore size of the membrane, which was then studied for different membranes. In addition, mass transfer within the hollow fibre geometry of these membranes was studied and accounted for in the model description. The model was applied in various studies on the influence of process conditions (pressure, pH, ionic strength, feed composition) on membrane performance.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders prof.dr.ir. R.G.H. Lammertink Research theme Complex dynamics of fluids Participants Xiuqin Wang Funded by Xiamen University, China Funded % Scholarships 100% Start of the project 2018 Information Rob Lammertink r.g.h.lammertink@utwente.nl	Project title Novel cross-linked ion exchange membranes Project aim Crosslinked anion exchange membranes by multi-cation crosslinker are one of the attractive candidates for water electrolysis applications, because they enable high ionic conductivity to take place under high ion exchange capacity and low swelling, thereby minimizing the overall cell voltage for water splitting. Progress Ionic liquids with dense functional ionic groups are used as cross-linking agents, which can increase the IEC while limiting the swelling. The cross-linked film display very good dimensional stability and flexibility. The membranes were not 100% cross-linked but form additional side chain structures. The combination of cross-linked structure and side chain structure promotes the formation of ion clusters with different size as confirmed by AFM and SAXS. Using these membranes the electrochemical performance for alkaline water electrolysis is investigated.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Prof.dr.ir. R.G.H. Lammertink, Dr. J.A. Wood Research theme Complex dynamics of fluids Participants Burak Akdeniz Cooperation MCEC Office, University Utrecht Funded by NWO Funded % NWO 100% Start of the project 2019 Information Rob Lammertink r.g.h.lammertink@utwente.nl	Project title Making use of diffusiophoresis for enhanced mass transports of spatially inhomogeneous catalysts Project aim One of the essential components of the chemical industry are the catalytic materials which are mostly suffered from insufficient transport in their bulk. In this project, we propose to improve hydrodynamic transport away from the surface of the inhomogeneous catalysis by making use of diffusiophoretic effect. The main aim of this project is to (i) quantitative describe this process by comparing controlled experiments and simulations in 2D, (ii) to optimize the pattern of the catalyst to achieve optimal flow with as small catalytic region as possible and (iii) to apply this concept also to 3D catalyst (porous media). Progress It is essential to understand the underlying mechanism of diffusiophoresis/ diffusiosmosis phenomena before moving to more advanced systems. For this reason, fundamental objectives with electrolyte and non-electrolyte cases are considered in a dead-end pore setup. With performed experiments, we have a better understanding of the surface-induced flow, and we analyzed the influence on the particle migration through the dead-end pore. Moreover, we worked on identifying the actual driving force of the diffusiophoresis experimentally. In the electrolyte case, we got an indication that particle velocity is driven by the relative gradient, as theoretically suggested before.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces

<table>
<tr>
<td>
<u>Project leaders</u>
Dr. J.A. Wood , Prof.dr.ir. R.G.H. Lammertink

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Arputha Paul

<u>Funded by</u>
NWO

<u>Funded %</u>
NWO 100%

<u>Start of the project</u>
2017

<u>Information</u>
Rob Lammertink
r.g.h.lammertink@utwente.nl
</td>
<td>
<u>Project title</u>
Stirring the boundary layer with electric fields

<u>Project aim</u>
The optimisation of most of the chemical-process technologies involving solid-fluid interfaces, is often limited by the boundary layers that are formed near the interface, since it determines the rate of transport of different species across these interfaces. One innovative way to reduce this boundary layer is by creating a self-induced mixing within the layer i.e. velocities being generated only near the interface, instead of the conventional way of using energy dissipating techniques that stirs the whole bulk fluid. This is possible by creating gradients along the interfaces that induce fluid velocity near the interfaces that might reduce the boundary layer thickness. Thus, the aim of the project is to control and enhance the transport of chemical species across the boundary layer, externally, using electric fields.

<u>Progress</u>
An electrodialysis set-up was chosen to study the effect of the stirring caused by electric fields. Consequently, experiments with surface modified spacers that can bend the electric field are being carried out and results will be compared with the electrodialysis using normal spacers. The experimental and numerical outcomes will be used to understand the different electro-kinetic effects across interfaces. And eventually the effect of the stirring caused by voltage as a gradient, if any, will be quantified and reported.
</td>
</tr>
</table>

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Prof.dr.ir. R.G.H. Lammertink Dr. J.A. Wood Research theme Complex dynamics of fluids Participants Harm Wiegerinck Funded by NWO Funded % NWO 100% Start of the project 2020 Information R.G.H. Lammertink r.g.h.lammertink@utwente.nl	Project title Project number OCEN W.KLEIN .124 / budget number: 9250:L Ion Transport Phenomena in Structured Colloidal Networks Project aim In this project ion-selective 3D colloidal particle networks will be used to mimic ion-exchange membranes in a millifluidic chip. Because these particles networks are tunable in size and material and can be deposited by drying into different patch geometries. Using this platform the effect of the topology of heterogeneous membranes on electrodialysis performance will be tested by using electrical, fluid velocity and ion concentration characterizations. This fundamental knowledge could potentially lead to design rules for improved ionexchange membranes for desalination and osmotic energy harvesting. Progress Built millifluidic chip for electrodialysis with colloidal particle networks. Found a method to successfully deposit colloidal particles in the millifluidic chip by drying colloidal dispersions. Started “membrane” potential measurements with the millifluidic chip setup. Built a basic Shock electrodialysis model in Comsol based on literature and extended it by including particle charge regulation and temperature effects due to Joule heating.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Prof.dr.ir. R.G.H. Lammertink Research theme Complex dynamics of fluids Participants Hans David Wendt Cooperation Oasen, Nijhuis Water, KWR, 4 water boards, Saxion, NX Filtration, STOWA Funded by NWO and industry Funded % NWO 75 % Industry 25 % Start of the project 2020 Information R.G.H. Lammertink r.g.h.lammertink@utwente.nl	Project title Organic micro-pollutant removal from municipal effluents by polyelectrolyte multilayer based hollow fibre nanofiltration membranes Project aim The presence of organic micro-pollutants (OMPs) in surface water leads to growing concern. This work proposes to extend a wastewater treatment plant (WWTP) with a nanofiltration step to prevent OMPs from entering surface waters. The aim is to concentrate the OMPs from the effluent and treat them further in the WWTP by recycling the concentrate. The goal is to understand the separation and degradation performance of the hybrid process. Specific membrane characteristics need to be included within the process scheme to accurately predict accumulation of OMPs and salinity, which necessitates understanding and describing OMP and ion transport through these membranes. Progress Accumulation is predicted by state of the art and a simple mass balance model for the full process. The state-of-the-art models can predict negative retentions for monovalent ions, which is validated by experimental results for ion mixtures. Furthermore, all models show that salts with low solubility (e.g. $CaSO_4$) might exceed the solubility limit, creating the potential issue of scaling. Solutions are suggested to mitigate this issue. If a first-order rate is assumed for OMP degradation in the bioreactor, the process is promising to increase the overall removal of OMPs from 30 to 80-90%.

UT – Science and Technology – Soft Matter, Fluidics and Interfaces	
Project leaders Dr. J.A. Wood Prof.dr.ir. R.G.H. Lammertink Research theme Complex dynamics of fluids Participants Angela Mary Thomas Funded by Evonik Industries Funded % Industry 100 % Start of the project 2020 Information R.G.H. Lammertink r.g.h.lammertink@utwente.nl	Project title Computational Study of Membrane & Modules (COSMOS). 2-year post-doctoral position on the modeling of gas separation membrane processes Project aim Postdoc research: In this project a theoretical/numerical analysis will be performed that will analyze the potential of Mixed Matrix Membranes (MMMs) as membrane materials and compare them to both conventional and facilitated transport membranes. This will be accomplished through modeling and simulation of the relevant transport phenomena in these systems, in order to understand the microscopic mass transport limitations within the separation layer and macroscopic mass transport limitation at a module level. The outcome of this project will give insights in the capabilities of MMMs, the efficacy of readily prepared membranes and their viability within processes. Progress Under the COSMOS project in 2020, material based theoretical calculations were conducted to assess the feasibility of MMMs for gas separation. By analyzing the relative flux using COMSOL through various combinations of the membrane components (the matrix and the filler), the effect of the filler and matrix selectivities and permeabilities on the separation efficiency of MMM was noted. It has been found that a filler-matrix combination with comparable permeabilities can achieve significant shift in MMM performance. Several theoretical studies were also implemented to describe possible defects in MMMs.

UT – Engineering Technology - Engineering Fluid Dynamics

Prof.dr.ir. CH Venner

Modern engineering challenges concern the design and development of processes and equipment with a high performance and minimal environmental impact, i.e. effective use of resources, energy and minimal generation of waste products and noise. The Engineering Fluid Dynamics group aims to contribute to innovative solutions for practically relevant societal, environmental and industrial challenges by bringing “fundamental” physics to actual applications on many scales, ranging from large size high power such as aircraft parts, compressors, and turbines, to small-scale thin-layer free surface flows in lubrication and bearing applications. The research is both experimental and theoretical, including numerical simulations with in house development of accurate numerical codes and multilevel/multigrid computational methodologies. The research focuses on the following themes:

Fluid Mechanics of Rotating Flow Machines
The flow in centrifugal pumps, compressors, and around wind turbine blades. The research involves optimization of the functional aspects (blade/impeller geometry, cavitation characteristics, efficiency, active flow control) as well as the minimization of non-drag related energy losses in the lubrication and transition layers, and minimization of environmental aspects such as the reduction of vibrational and (aeroacoustic) noise. For this purpose, an aeroacoustic test facility is used: A silent closed circuit wind tunnel with a (0.7x0.9 m2) free-jet test-section (maximum velocity 65 m/s) which is enclosed by a 6x6x4m3 anechoic chamber.

Multi-phase flows and wave phenomena
Flows with phase transition occur in many engineering applications such as flow of oil/water/gas mixtures in hydrocarbon transport lines, ice accretion on aircraft wings in flight, flows with cavitation, separation of mixtures, and dense-phase fluid particle flows in dredging applications. The group develops computational methods for specific applications aimed at actual design and prototyping and also carries out fundamental studies, on the mechanisms of e.g. condensation, in multiphase systems. Research is also carried out aimed at identifying the acoustic signature of the flow.

Computational aerodynamics algorithm design
Practical applications in engineering involve the occurrence of phenomena on largely different scales in almost any application. In such cases both high order accuracy as well as computational efficiency are of the utmost importance. The group develops and tests numerical algorithms for simulation and optimization , and validates predictions for actual applications ranging from Navier Stokes and Euler equations to potential flows, and reduced systems such as lubrication flows with combined elasticity on nano-scale. Aspects of development are high order compact schemes, multigrid/multilevel computational methodologies and gradient based adjoint optimization.

Bio-physical flows
This research deals with the flow in (bio)medical and natural systems, Projects include flow in lungs (aerosol deposition), medical sprays, and separation of specific cell rich flows. Research is aimed at developing new (computational) diagnostic and therapeutic tools. Research in nature-inspired flows is aimed at the development of robot-birds and minimizing the impact of technology on the natural environment.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H. Venner
E.T.A. van der Weide
H. Ozdemir (ECN)

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
A. Koodly Ravishankara (PhD),
E.T.A. van der Weide
H. Ozdemir and
C.H. Venner

Cooperation
ECN

Funded by
ECN

Funding %
ECN (TNO) 100 %

Start of the project
2017

Information
E.T.A. van der Weide
e.t.a.vanderweide@utwente.nl

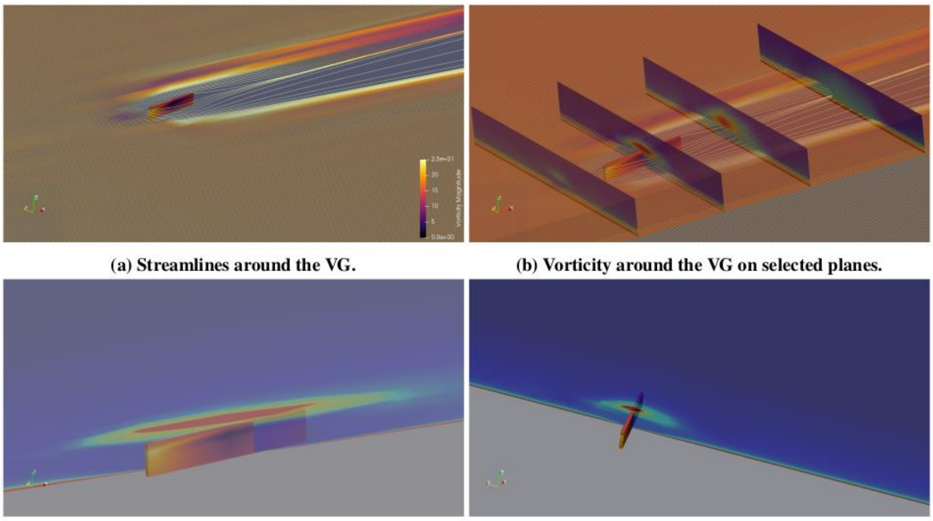
Project title
Numerical Methods for Wind Turbine Aerodynamics

Project aim
Current wind turbines have very large rotors and many use flow enhancement devices like vortex generators. Traditional low-fidelity turbine design and analysis tools are either incapable of handling such complexity or give very inaccurate results. This research focuses on developing high-fidelity tools to design and analyze current and future wind turbines. Additionally, the high-fidelity models can also be used to analyze wind farms and tune the lower fidelity tools. The new models will be implemented in the open source CFD code SU2.

Progress
The pressure based incompressible solver of SU2 has been tested for several test cases, including a case with vortex generators. Solver applied to relevant cases.

2020 Publications
Implementation of a pressure based incompressible flow solver in SU2 for wind turbine applications. AIAA-paper 2020-0992, January 2020

Ravishankara, A.K., Bakhmet, I., Ozdemir, H., 2020, Estimation of roughness effects on wind turbine blades with vortex generators, Journal of Physics, Conference Series, V. 1618, 052031.



(a) Streamlines around the VG. **(b) Vorticity around the VG on selected planes.**

(c) Extent of vorticity along $x-$ axis. **(d) Extent of vorticity along $y-$ axis.**

Vorticity profiles in a turbulent boundary layer around a vortex generator.

UT – Engineering Technology - Engineering Fluid Dynamics

<u>Project leaders</u>
C.H. Venner
E.T.A. van der Weide
B. Soemarwoto (NLR)

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
J.S. Fischer (PhD),
E.T.A. van der Weide
B. Soemarwoto and
C.H. Venner

<u>Cooperation</u>
NLR

<u>Funded by</u>
NLR

<u>Funding %</u>
NLR 100 %

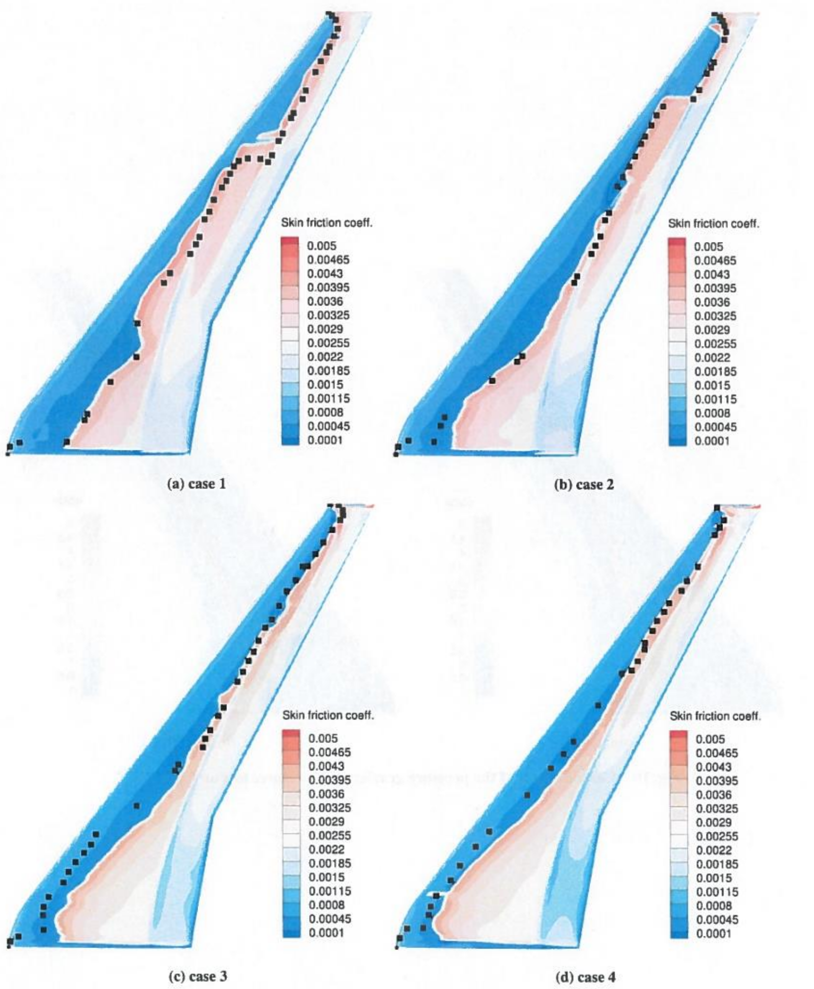
<u>Start of the project</u>
2018

<u>Information</u>
E.T.A. van der Weide
e.t.a.vanderweide@utwente.nl

<u>Project title</u>
Laminar wing design

<u>Project aim</u>
Natural laminar flow (NLF) technology has been identified as a promising candidate to achieve fuel burn savings on small and mid-sized aircraft in the order of 10%. This PhD project is dedicated to extend the scope of the three-dimensional optimization for wings making use of transition information. Use will be made of existing methods for the aerodynamics (ENSOLV RANS solver), transition modeling (correlation based, stability analysis) and optimization (gradient based in combination with adjoints). The main work of this project is to integrate these methods into a practical design tool that allows for optimization of aircraft wings including transition.

<u>Progress</u>
The coupling between the flow solver (ENSOLV) and the linear stability solver (COSALX) has been finalized and tested extensively for the Sickle wing of the TU Braunschweig and the NASA CRM. The results for these test cases are very encouraging and show the potential of the method. Furthermore, a start has been made with the adjoint version of the coupled solver.



Skin friction coefficient contour on the NASA CRM upper surface for cases 1, 2, 3 and 4 resulting from RANS-LST coupling and indicated transition points from experiment.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H. Venner
E.T.A. van der Weide

Research theme

Mathematical and computational methods for fluid flow analysis

Participants
E. Shehadi (PhD),
E.T.A. van der Weide and
C.H. Venner

Cooperation
GE Germany, Technical University of Munich, Karlsruhe Institute of Technology, University of Cambridge, ARMINES Paris Tech, CERFACS, Safran Tech, Safran Helicopter Engines, ANSYS

Funded by
EU

Funding %

University	25 %
EU	75 %

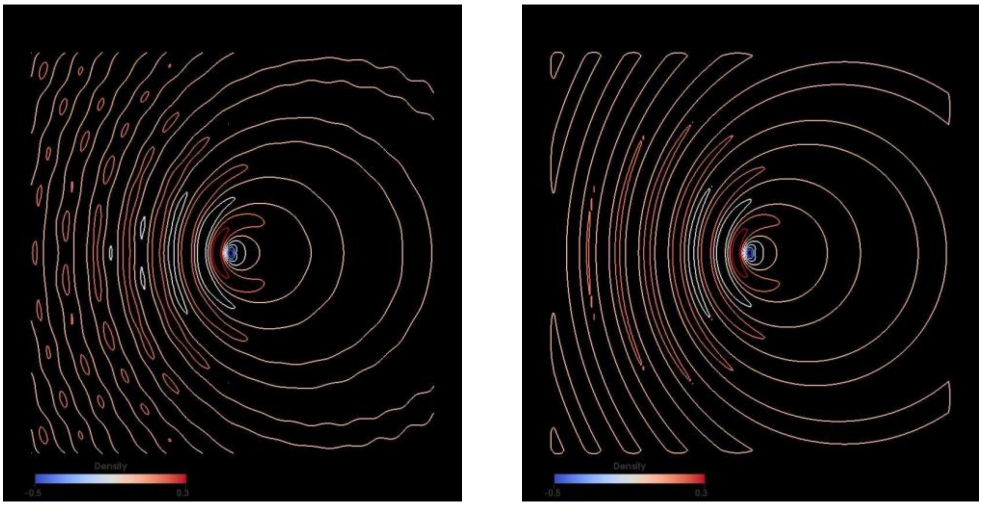
Start of the project
2018

Information
E.T.A. van der Weide
e.t.a.vanderweide@utwente.nl

Project title
LES of Compressible Turbulent Flow through Combustor Liner and Dilution holes

Project aim
The EU project MAGISTER addresses the problem of thermo-acoustic instabilities in combustion chambers of aircraft engines (which occur when cleaner combustion processes are aimed for) by means of machine learning (ML). The role of this PhD project is to deliver simulation data for the turbulent flow through combustor liner and dilution holes that will act as training data for the ML algorithms. For this purpose the high order discretization in the open source code SU2 will be used and this project will focus on non-reflecting boundary conditions using the Perfectly Matched Layer approach, which enables the LES simulation of turbulent flows in the above-mentioned geometries.

Progress
Due to the demanding computational cost in Matlab, prototyping has shifted to raw C++, for high-performance purposes. A two-dimensional multi-zonal structured solver has been developed in C++ from scratch to tackle how effective as well as compare different non-reflective boundary conditions in conjunction with the high-order discontinuous Galerkin method. Non-reflecting methods tested are the perfectly matched layer (PML) and the Navier-Stokes characteristic boundary condition (NSCBC). The discontinuous Galerkin variant used is of a nodal type. The physics studied ranges from the linearized Euler equations to the Navier-Stokes, with the non-linear Euler a special case.



Density contours for the characteristic boundary condition treatment (left) vs. perfectly matched layer approach (right) for a pulsating Gaussian source solved with the Euler equations.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H. Venner
E.T.A. van der Weide

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
E.T.A. van der Weide and
C.H. Venner

Cooperation
NLR, DNW, DeHarde, ALTRAN, ASDE, Leonardo

Funded by
EU

Funding %
EU 100 %

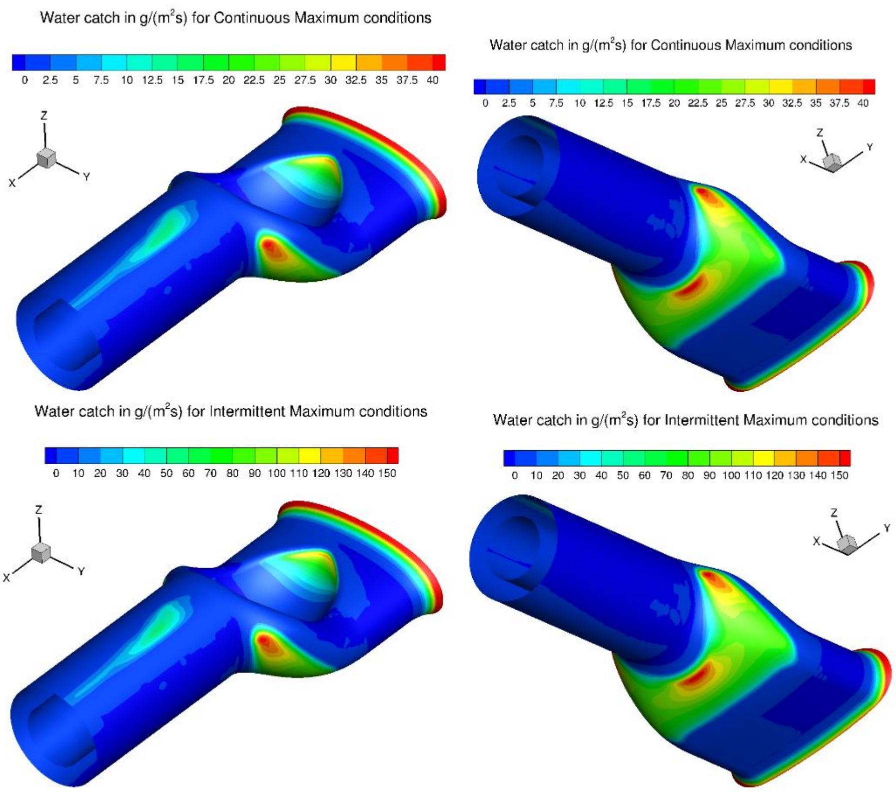
Start of the project
2018

Information
E.T.A. van der Weide
e.t.a.vanderweide@utwente.nl

Project title
Tilt Rotor Inlet Innovative Design And Testing (TRINIDAT)

Project aim
The goal of the EU project TRINIDAT is to optimize and experimentally validate the design of the inlet of the turboprop engines for the Next Generation Civil Tilt Rotor configuration that is being developed by Leonardo. Within this project the University of Twente is responsible for the icing analysis of the inlet, for which the in-house code MB-Ice, developed in the EU project HAIC is used.

Progress
The icing analysis of the first inlet design have been carried out using the flow solution from NLR. The water catch has been computed for both Continuous Maximum and Intermittent Maximum conditions. The analysis shows that certain parts of the inlet are rather sensitive for ice accretion, while other parts hardly show any catching of water.



Computed water catch for Continuous and Intermittent Maximum conditions.

UT – Engineering Technology - Engineering Fluid Dynamics	
Project leaders Y.H. Wijnant C.H.Venner Research theme Mathematical and computational methods for fluid flow analysis Participants Msc. H. Faghanpourgan Cooperation Comfoor Funded by Comfoor EFRO PDEng Cluster Smart Industries Oost Nederland Funding % Industrty 50% EU 50% Start of the project 2018 Information Dr. Ir. Y.H. Wijnant y.h.wijnant@utwente.nl	Project title Development of a flat spectrum filter for 3D printed custom-made hearing protection Project aim This PDEng Project aims to develop an adaptive hearing protection device. As opposed to hearing aids, which could also be used to attenuate sound instead of amplify sound, the focus is on developing an active component which only increases transmission loss when needed. This increases the users comfort as there is no need to remove the hearing protection device in silent surroundings. Progress A mathematical model to simulate sound propagation through the hearing protection device has been implemented and associated hardware has been manufactured. A design has been developed, successfully tested and validated. 2020 Dissertations Faghanpourganji, H., 2020, Development of a flat spectrum filter for 3D printed custom-made hearing protection. PDENG Thesis, University of Twente. 

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
Y.H. Wijnant
C.H.Venner

Research theme

Mathematical and computational methods for fluid flow analysis

Participants
Msc. J. Mahajan

Cooperation
Comfoor

Funded by
EFRO PDEng Cluster Smart Industries Oost Nederland

Funding %
University 50%

Start of the project
2018

Information
Dr. Ir. Y.H. Wijnant
y.h.wijnant@utwente.nl

Project title
Development of an energy-efficient adaptive hearing protection device

Project aim
This PDEng Project aims to develop an adaptive hearing protection device. As opposed to hearing aids, which could also be used to attenuate sound instead of amplify sound, the focus is on developing an active component which only increases transmission loss when needed. This increases the users comfort as there is no need to remove the hearing protection device in silent surroundings.

Progress
A mathematical model to simulate sound propagation through the hearing protection device has been implemented and ways to actively attenuate the sound have been identified. We further focussed on the actual design and implementation.

UT – Engineering Technology - Engineering Fluid Dynamics

<u>Project leaders</u>
Y.H. Wijnant
C.H. Venner

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
Ir. Niels Consten
Prof. A. de Boer

<u>Cooperation</u>
Soundinsight

<u>Funded by</u>
Soundinsight

<u>Funding %</u>
Industry 100%

<u>Start of the project</u>
2018

<u>Information</u>
Dr. Ir. Y.H. Wijnant
y.h.wijnant@utwente.nl

<u>Project title</u>
On the measurement of intensity and absorption using a spherical microphone array.

<u>Project aim</u>
The Project aims to develop and validate the possibility to measure sound intensity and sound absorption using a spherical array of microphones. It includes the development of a suitable calibration procedure. In addition, the statistically attainable accuracy should be investigated.

<u>Progress</u>
A mathematical model has been implemented and hardware is available to measure sound intensity. A first calibration procedure has been implemented and is investigated using statistical data analysis.

<u>2020 Publications</u>
Niels Consten, Theo Campmans, Stéphanie Bertet, Ysbrand Wijnant. On the measurements of sound power using a cubical arrangement of microphones in a small rigid sphere. Conference Proceedings 45th DAGA Rostock, 2020.

-

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
L.D. de Santana
M. Tuinstra
C.H. Venner

Research theme
Complex dynamics of fluids

Participants
J.Biesheuvel (PhD)
L.D. de Santana
M. Tuinstra
C.H. Venner

Cooperation
Embraer, DNW, NLR

Funded by
TKI

Funding %
TKI 100%

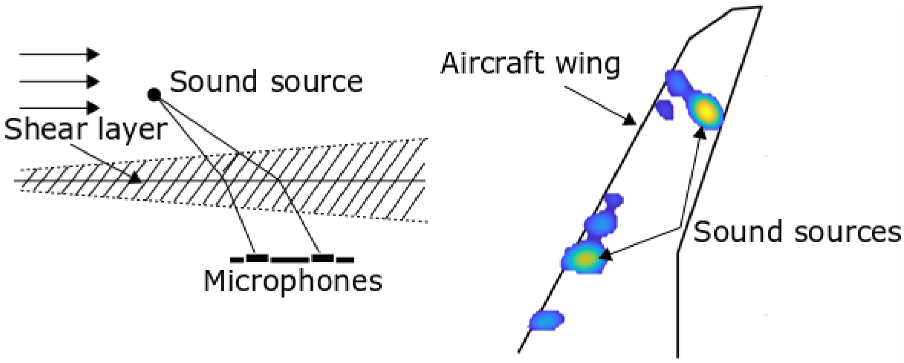
Start of the project
2018

Information
Dr. L. D. de Santana
leandro.desantana@utwente.nl

Project title
Advancing Aircraft Silent Design by Wind Tunnel Test Uncertainty Reduction: Measurement uncertainties in Aeroacoustic wind tunnels

Project aim
The Project aims at reducing uncertainties in noise measurements in wind tunnel experiments, allowing aircraft OEM to reduce their wind tunnel testing costs and decrease design lead times. A key uncertainty is the decorrelation of sound traveling through the shear layer of an open jet wind tunnel. This leads to reduced microphone cross-powers, and lower Sound Pressure Levels when a beamforming algorithm is applied to localize and quantify acoustic sources. The priority is to understand the process that leads to coherence loss and to robustly model the phenomenon. Correction methods will be proposed reducing measurement/design uncertainty.

Progress
A microphone coherence loss model was developed and evaluated using measurements in NLR's Small Anechoic Wind tunnel (KAT). The model was applied to assess the effect on beamforming measurements in closed test-sections on an industrial scale (DNW-LLF, figure on the right). Aerodynamic & acoustic measurements were performed in the Large Low-speed Facility of the German-Dutch Wind tunnels: Noise measurements on a full aircraft model (open-jet), and calibrated speaker tests to study the interaction of the acoustic waves with a turbulent shear layer (figure on the left).



UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H. Venner
L.D. de Santana

Research theme
Complex dynamics of fluids

Participants
M.P.J. Sanders (PhD)
J. Biesheuvel (PhD)
L.D. de Santana
C.H. Venner

Cooperation
Embraer, DNW, NLR

Funded by
TKI

Funding %
TKI 100 %

Start of the project
2018

Information
L. D. de Santana
leandro.desantana@utwente.nl

Project title
Advancing Aircraft Silent Design by Wind Tunnel Test Uncertainty Reduction: High Lift Device Sound Sources

Project aim
Noise generated by the high-lift devices (i.e. wings) is a major contributor to the sound production of an airplane. The flow mechanisms associated with this noise source are complex and require small-scale wind tunnel testing that enable the measurement and understanding of essential flow quantities related to the noise production. By flow field characterization and coupling with far-field noise measurements, we investigate the restraints and uncertainties encountered in small-scale wind tunnel testing of high-lift devices. Based on these findings, more physics-based modeling of high-lift device noise can be developed which can be used in silent aircraft design.

Progress
The research is initiated with a renovation of the wind tunnel test sections at the University of Twente. The necessary sound localization and quantification tools have been developed in-house and benchmarked. Airfoil noise prediction models have been implemented which form the cornerstone to modeling of the high-lift device noise. Aeroacoustic measurements have been conducted in an open-jet (Figure 2) and hard-wall test section wind tunnel and are compared with LBM simulations (Figure 1). Acoustic corrections still need to be applied to the measurement data. The dataset is used to propose extended high-lift device noise prediction models.

2020 Publications
Sanders, P.J., de Santana, L., Venner, C.H. 2020, The Sweep Angle Effect on Slat Noise Characteristics of the 30P30N HighLift Model in an Open Jet Wind tunnel, AIAA Aviation Forum, June 15-19 2020.

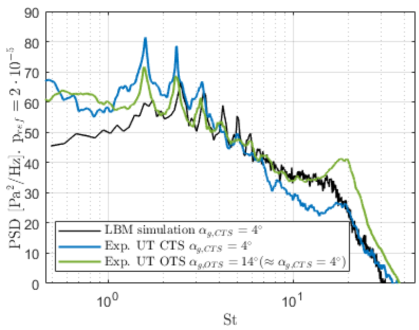


Figure 1 - Sound Pressure Level comparison of the noise generated by the high-lift device based on LBM simulations, and open-jet and hard wall test section measurements.

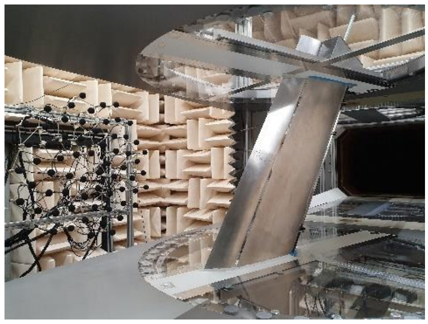


Figure 2 - Experimental setup at the University of Twente with the high-lift device in swept wing configuration.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
L. de Santana,
C.H.Venner

Research theme
Complex dynamics of fluids

Participants
F. dos Santos

Cooperation
TNO

Funded by
University

Funding %
University 100 %

Start of the project
2019

Information
Dr. L. D. de Santana
leandro.desantana@utwente.nl

Project title
Influence of tripping devices in hastening transition in zero and favorable pressure gradients

Project aim
Tripping devices are commonly used in wind tunnel testing to hasten the laminar-turbulent transition to mimic the aerodynamic effects present in the full-scale application. Depending on the tripping device characteristics and flow conditions, the development of a fully developed turbulent boundary layer is challenging. Therefore, this projected aimed to investigate experimentally the effectiveness of two types of tripping devices, e.g., zigzag strips and randomly distributed grits of varied sizes, in hastening the transition and generating a fully developed turbulent boundary layer in a flat plate. Flows of zero and favourable pressure gradients were also considered.

Progress
An experimental setup was developed where hot-wire measurements were performed to quantify the velocity and turbulence intensity profiles in the streamwise direction. The research demonstrates that all trips hastened the transition from laminar to turbulent flow. Zigzag strips introduced stronger disturbances inside the boundary layer compared to grits, consequently, requiring a more significant recovery length. The grits with height of 59% of the boundary layer thickness presented the most consistent results for both pressure gradients since this trip did not overstimulate the boundary layer and developed a turbulent flow faster than the other trips.

2020 Publications
F. L. dos Santos, M. P. Sanders, L. D. de Santana, C. H. Venner, Influence of tripping devices in hastening transition in a flat plate submitted to zero and favourable pressure gradients. AIAA Scitech 2020, AIAA 2020-0046 (2020). DOI: 10.2514/6.2020-0046.

Left: Flat plate with the grits elements distributed near the leading edge.
Right: Top view of the grit particles placed on the flat plate surface

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
L. de Santana
C.H.Venner

Research theme
Complex dynamics of fluids

Participants
F. dos Santos

Cooperation
TNO, MARIN

Funded by
DMO

Funding %
Industry 100 %

Start of the project
2019

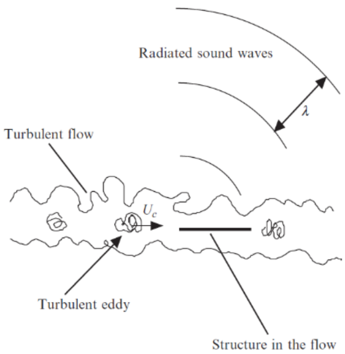
Information
Name Dr. L. D. de Santana
leandro.desantana@utwente.nl

Project title
Aeroacoustic noise prediction of ship propellers

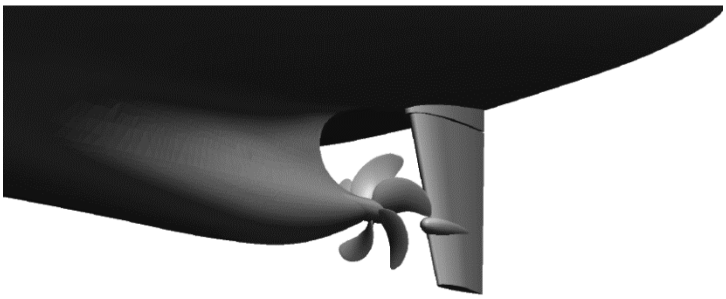
Project aim
Modeling the turbulence solid-body interaction is crucial for noise control. The Amiet noise prediction theory considers the noise radiated by a thin flat plate in an isotropic turbulent stream. These conditions are invalid for ship propellers since the inflow is disturbed by the ship hull, and the blades have a more complex geometry and rotational movement. Hence, this research aims to investigate experimentally the near-field phenomena at the leading edge of propeller blades and its relation to the far-field noise production. Based on the experimental results, improvements to the existing noise prediction models and/or new fast turn-around models will be proposed.

Progress
Literature review have been done regarding the aeroacoustic noise sources in propellers and the models used to predict far-field noise production. The next steps of the research are:
Develop an experimental setup to study the inflow turbulence characteristics and its influence in the far-field noise generation of well-known geometries, such as flat plate and airfoils, and compare the results with the Amiet leading edge noise prediction model;

- Investigate experimentally the influence in the far-field noise of a propeller geometry and its rotation movement in a turbulent stream.
- Experimental investigation of surface elements on pressure fluctuations and nose
- Experimental investigation dissipation range of Von Karman turbulence spectrum



Turbulent flow incident on a structure that radiates sound waves



Propeller and partial ship hull image.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
R. Hagmeijer, C.H. Venner

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
R. Hagmeijer (UT), H.C. Venner (UT), H.B. ter Woord (UT)

Cooperation
Vrouw-Kind Centrum & Medical School Twente, Medisch Spectrum Twente

Funded by
TKI HTSM, Medisch Spectrum Twente

Funding %
University 15%
NWO 70%

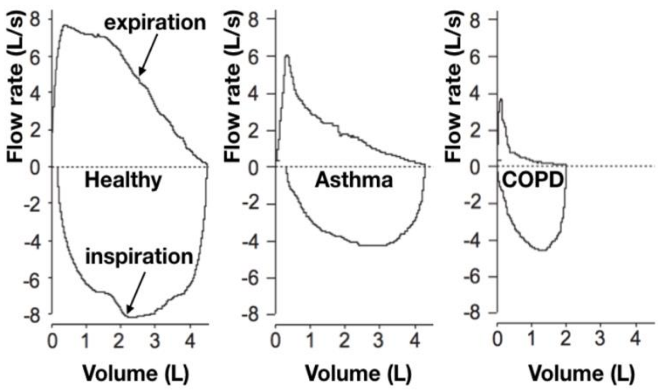
Start of the project
2019

Information
R. Hagmeijer
r.hagmeijer@utwente.nl

Project title
Towards a clinical decision support system in high-flow ventilation therapy: integrated respiration monitoring

Project aim
High flow nasal cannula (HFNC) therapy is widely used to treat critically ill patients with acute, severe respiratory disorders. However, criteria to support clinical decisions with respect to (a) initiation of HFNC therapy, (b) titration of HFNC therapy, and (c) control of airway pressure are lacking. We propose to develop criteria based on using the HFNC-device itself as monitoring device, supported by an optimized and validated theoretical lung model in the background.

Progress
Started with thorough review of the literature on existing pulmonary models. Some work has been done on developing flow in lung tubes and on the current state of the art on alveoli-modeling. Especially the role of surface tension in the alveoli is investigated.



Typical examples of flow-volume loops obtained from spirometry applied to a healthy patient and patients suffering from pulmonary diseases. Miller et al. , Standardisation of spirometry, Eur. Respir. J. 2005, 26: 319–338.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
R. Hagmeijer, C.H. Venner

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
R. Hagmeijer (UT), H.C. Venner (UT, M. Duiverman (UMCG), P.Wijkstra (UMCG), R. Hebbink (UT), K. Jain (UT), E. Groot Jebbink (UT)

Cooperation
Universitair Medisch Centrum Groningen

Funded by
TKI Longfonds

Funding %
University 100 % (TKI)

Start of the project
2018

Information
R. Hagmeijer
r.hagmeijer@utwente.nl

Project title
Nasal High-Flow Therapy to treat COPD exacerbations: a matter of monitoring and controlling settings?

Project aim
COPD is a disease with high morbidity and mortality worldwide. COPD exacerbations are *the* important contributor to disease deterioration and decrease in health-related quality of life (HRQoL). Therapeutic options to treat exacerbations effectively are limited. Many patients have persistent loss of vital functioning and suffer from frequent re-hospitalisations. Nasal high flow therapy (nHFT) is an innovative therapy that provides humidified and heated air through a nasal cannula. We aim to prove efficacy of nHFT in enhancing recovery exacerbations by developing new technologies to control and monitor the effect of nHFT and by providing background for optimal settings of nHFT.

Progress
We have worked on a new way of monitoring nHFT by using the therapeutic device itself, which seems quite successful. Furthermore we are in the course of doing large-scale Lattice-Boltzmann (LBM) simulations to investigate the refreshment rate (“wash-out”) of the nasal cavity, i.e., the removal of CO_2-rich air from the nasal cavity by means of the nHFT-jets. Development of new model of head and realization of experiments. Contribution to COVID research

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
G.W. Römer, R. Hagmeijer, C.H. Venner

Research theme
Complex dynamics of fluids

Participants
G.W. Römer, S. van der Linden, R. Hagmeijer,

Cooperation
EU and industry

Funded by
EU

Funding %
Industry 35%
EU 65%

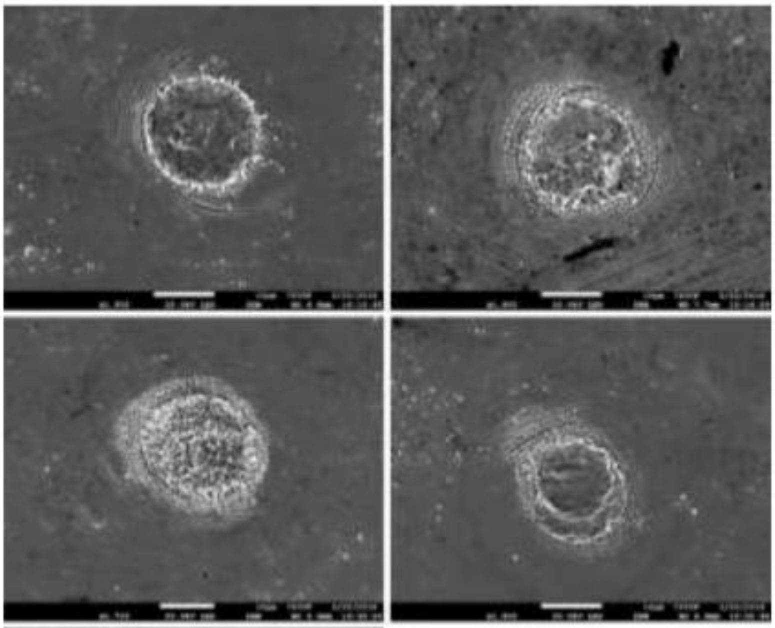
Start of the project
2016

Information
R. Hagmeijer
r.hagmeijer@utwente.nl

Project title
Safe and Amplified Industrial Laser Processing (SAILPRO)

Project aim
In this German-Dutch Interreg multi-partners project knowledge and expertise in on the generation & application of nanoparticles is combined with the knowledge and expertise in machining & application of surface textures. Both fields of application share the benefits of the same laser process: ultra-short laser processing under a water film. Partners: microTEC Gesellschaft für Mikrotechnologie mbH, University of Duisburg-Es- sen (UDE), Veld Laser (VL), PM Bearings (PMB), Laser Appli- cation Cen- ter (LAC), Particle Metrix GmbH (PM).

Progress
The required liquid layer height in under water laser ablation is typically realized by pouring a predefined amount of liquid on the surface. Surface tension causes the air-liquid interface at the boundaries of the domain to deviate from a planar interface. An experimental set-up is proposed which circumvents the issues of a curved free surface. Next, a 7 picosecond pulsed laser source at a wavelength of 515nm was used to study the efficiency of laser ablation of stainless steel for a range of liquid layer heights. Our findings provide a more detailed quantification of crater depth as a function of liquid layer height.



UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
N. P. Kruyt

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Chaofa Zhao, Niels P. Kruyt

Cooperation
University of Twente and IRSTEA (France) and Università dell'Aquila (Italy)

Funded by
EU

Funding %
EU 100%

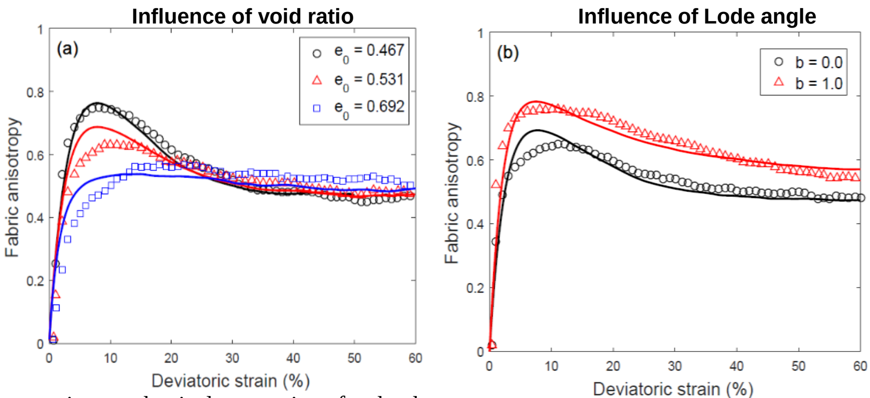
Start of the project
2019

Information
Niels P. Kruyt
n.p.kruyt@utwente.nl

Project title
Higher-order constitutive relations for granular materials: a multi-scale approach

Project aim
This Project aims to (i) propose micromechanical expressions for the three-dimensional higher-order strain and stress tensors for granular materials; (ii) investigate free-energy and dissipation potentials from the micromechanical viewpoint; (iii) develop thermodynamically consistent higher-order constitutive relations for rate-independent granular materials; (iv) validate the constructed higher-order constitutive relations, using available data from laboratory tests on granular materials; (v) apply the developed higher-order continuum-mechanical constitutive relations to large-deformation problems in geotechnical engineering.

Progress
An evolution law for fabric anisotropy of granular materials has been developed, that is based on observations from results of experiments and three-dimensional DEM simulations from literature.



Current micromechanical expressions for the three-dimensional higher-order strain and strain tensors for granular materials have been analyzed. Skills of DEM simulations of granular materials have been acquired. We are working on improving and proposing new micromechanical expressions for the three-dimensional higher-order strain and stress tensors for granular materials.

2020 Publications
Chao-Fa Zhao, Niels P. Kruyt. 2020, An evolution law for fabric anisotropy and its application in micromechanical modelling of granular materials, International Journal of Solids and Structures. 196-197, 35-66.
Chao-FA Zhao, Niels P. Kruyt, Millet, O., 2020, Capillary bridges between spherical particles under suction control: Rubture distances and capillary forces, Powder Technology, 360, 622-634. https://doi.org/10.1016/j.powtec.2019.09.093

UT – Engineering Technology - Engineering Fluid Dynamics

<u>Project leaders</u>
N.P Kruyt, C.H. Venner

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
Jie Wang, N.P Kruyt, C.H. Venner, E.T.A van der Weide, L.D. de Santana

<u>Cooperation</u>
University of Twente and Howden

<u>Funded by</u>
CSC & Howden

<u>Funding %</u>
Industry 100%

<u>Start of the project</u>
2017

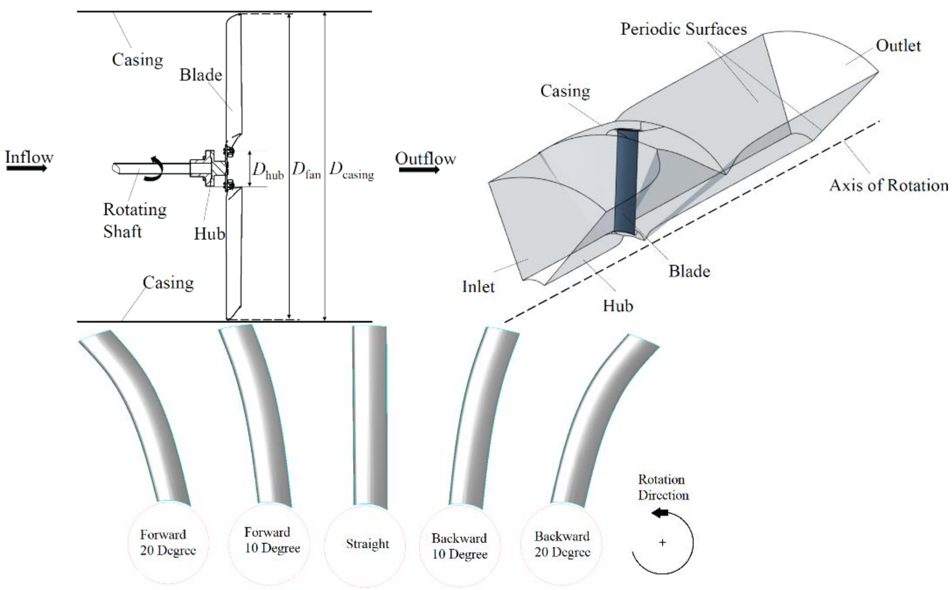
<u>Information</u>
N.P Kruyt
n.p.kruyt@utwente.nl

<u>Project title</u>
Design for High Efficiency of Low-pressure Axial Fans: Use of Blade Sweep and Vortex Distribution

<u>Project aim</u>
The effects of blade sweep (in axial and circumferential direction) and of the vortex distribution (i.e. the spanwise variation of the “blade loading” or “mean-swirl distribution”) on the aerodynamic performance of low-pressure axial fans are investigated parametrically by using CFD. Based on these results, optimized design methodologies considering blade sweep and vortex distribution will be formulated for obtaining low-pressure axial fans with high efficiency.

<u>Progress</u>
The strategy of CFD simulation of low-pressure axial fans with low hub-to-tip ratio (lower than 0.2) has been reported, good agreements with experimental data are found. Based on the strategy, simulation results of swept blade have been updated and investigated, and reported. Effects of vortex distribution on the axial fan performance, as well as the couple of vortex distribution and blade sweep.

<u>2020 Publications</u>
Wang, J., and Kruyt, N.P., 2020, Validation of CFD Simulations of aerodynamic performance of low speed axial fans with low hub-to-tip ratio. AIAA Scitex 2020, https://doi.org/10.2514/6.2020-0345



UT – Engineering Technology - Engineering Fluid Dynamics

<table>
<tr>
<td>

<u>Project leaders</u>
C.W. Visser
C.H. Venner

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Jieke Jiang

<u>Cooperation</u>
IamFluidics
Aquamarijn
Bether Encapsulates

<u>Funded by</u>
European Regional Development Fund

<u>Funding %</u>
University 60%
EU 40%

<u>Start of the project</u>
2017

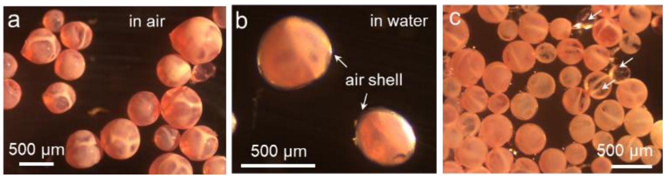
<u>Information</u>
Jieke Jiang
j.jiang-1@utwente.nl

</td>
<td>

<u>Project title</u>
In-air microfluidics, better and faster encapsulation

<u>Project aim</u>
• To identify the different materials for microcapsules
• To observe the formation of air-shell outside the capsules
• To identify how the flow rate will influence the formation of capsules

<u>Progress</u>
By screening varying combinations of polymers/nanoparticles and solvents, capsules were made from a liquid jet at high speed (>1 ml/min) (figure a); A layer of air formed outside the capsules (figure b); An existing problem is some of the capsules will release the liquid core when contact with water (indicated by arrows in figure c). We will try to solve this problem by adjusting the diameter of liquid jet and the ratio of polymer and nanoparticles.



<u>2020 Publications</u>
2020, Jiang, J., Shea, G., Rastogi, P., Kamperman, T., Venner, C.H., Visser, C.W. Continuous High Throughput Fabrication of Architected Micromaterials via In-Air Pohtopolymerization. Advanced Materials, dec. 2020 https://doi.org/10.1002/adma.202006336

</td>
</tr>
</table>

<table>
<tr><th colspan="2">UT – Engineering Technology - Engineering Fluid Dynamics</th></tr>
<tr><td>
<u>Project leaders</u>
C.H.Venner

<u>Research theme</u>
Mathematical and computational methods for fluid flow analysis

<u>Participants</u>
C.H.Venner, H.Boffy, P.M.Lugt(SKF), J.Wang (Qingdao), A.A. Lubrecht (INSA), J.H.Snoeijer (UT-PoF)

<u>Cooperation</u>
UT-TNW (PoF), INSA de Lyon, France, Qingdao Technological University, PR. China, SKF ERC, Netherlands, UvA, Leibniz University Hannover, FZG University of Munich.

<u>Funded by</u>
UT/SKF

<u>Funding %</u>
University 50%
Industry 50%

<u>Start of the project</u>
2007

<u>Information</u>
C.H.Venner
c.h.venner@utwente.nl
</td><td>
<u>Project title</u>
Thin Layer Flow

<u>Project aim</u>
Development of accurate thin film/layer flow models and numerical simulation algorithms for the prediction of lubricant film formation capacity and lubrication life in rolling element bearing contacts in relation to operating conditions, lubricant rheology (oil-grease), supply conditions (starved-flooded), and material properties, and fundamental analysis of physical phenomena.

<u>Progress</u>
Continuation of research on fundamental aspects of film formation fundamentals, and friction prediction.

<u>2020 Publications</u>
Liu, H.C., Zhang, B.B., Bader, Nl, Poll, G., 2020, Scale and contact geometry effects on friction in thermal EHL: twin-disc versus ball-on-disc. Tribology International, doi:10.1016/j.triboint.2020.10669
</td></tr>
</table>

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H. Venner

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Binbin Zhang, Armando Felix

Cooperation
SKF ERC, Nieuwegein, INSA de Lyon, France

Funded by
CSC, SKF

Funding %
University 10%
Scholarships 90%

Start of the project
2016

Information
Prof.dr.ir. C.H.Venner
c.h.venner@utwente.nl

Project title

Landscaping the Subsurface for Tribological Longevity

Project aim

Development of optimally efficient and computational methods for advanced computational diagnostics and optimization of the effect of 3D topology and structural heterogeneity in the subsurface of bearing materials on the service life in contact mechanics and lubrication. The methods will allow “design” of the required local topological mechanical and thermal properties such that fatigue life is maximized, whilst maintaining lubrication life, as well as quick assessment of risk of reduced “lubrication life” from tomographic maps of actual material samples.

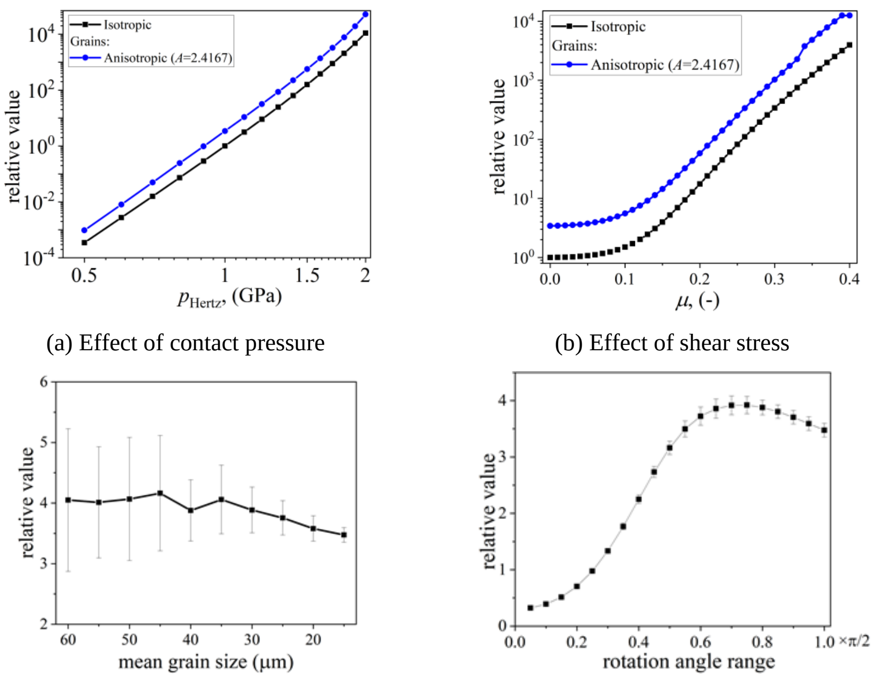
Progress

Extensive simulations for anistropic materials with grain structure, writing of PhD Thesis.

2020 Publications

Zhang, B., Felix Quinonez, A., and Venner, C.H., 2020, Effect of Material Anisotropy on Rolling Contact Fatigue under Dry and Lubricated Contact Conditions: A Numerical Study. Tribology International, in review. preprint published online https://doi.org/10.1016/j.triboint.2020.106584.

Zhang, B., Liu, H., Felix Quinonez, A., and Venner, C.H., 2020, Effects of 3D anisotropic heterogeneous subsurface topology on film thickness, pressure, and subsurface stresses in an elasto-hydrodynamically lubricated point contact., Tribology International, V. 151, 106471. https://doi.org/10.1016/j.triboint.2020.106471



(a) Effect of contact pressure (b) Effect of shear stress

(c) Effect of grain size (d) Effect of rotation angle

Effect of contact pressure, shear stress, grain size and rotation angle on the relative stress integral value for heterogeneous anisotropic and isotropic material (p_h=1 GPa except 1(a), R=0.02 m, a_{hertz}=138.78 µm, average grain diameter 15 µm except 1(c) rotation angel range 0~π/2 except 1(d))

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H.Venner

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Yan Zhao
C.H.Venner
G.E. Morales-Espejel

Cooperation
SKF

Funded by
SKF/CSC

Funding %
University 10%
Scholarships 90%

Start of the project
2018

Information
C.H.Venner
c.h.venner@utwente.nl

Project title

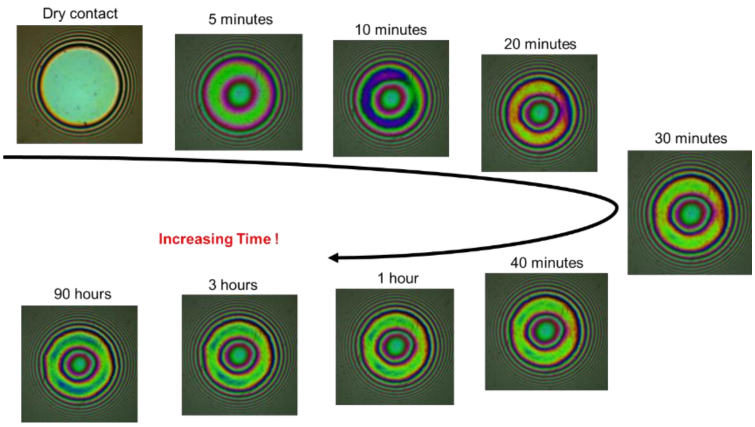
Viscoelastic Layer Modeling for Contact Mechanics with Heterogeneous Materials and (Mixed) Lubrication Layer Islands Interface.

Project aim

Develop a new approach in which the lubricant film is replaced by a locally varying viscoelastic layer which can be tuned to mimic a fluid like layer, or a more complex polymer or grease thickener layer, which is combined with a dry contact modeling. Using optical interferometry methods with thin polymer layers on disc to validate the model. The developed model and computational method will be combined with extremely efficient Multigrid/Multilevel computational methods which allow to simulate full 3D heterogeneous, granular, and anisotropic material behaviour yielding a novel and efficient way to analyze and optimize lubricated contacts as transitional interfaces.

Progress

Extensive investigation of numerical simulations of viscoelastic contact problems. New algorithm developed avoiding expensive time integration. Application to squeeze and rolling problems.



The recorded interferogram showing the variation of the film shape in the impact experiments. (F = 20 N, P_h=0.526 GPa, T = 23 ℃)

UT – Engineering Technology - Engineering Fluid Dynamics

<u>Project leaders</u>
C.H.Venner
H.W.M.Hoeijmakers

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
E.T.A. van der Weide, S. de Maag, F. Segerink, H.L. Offerhaus.

<u>Cooperation</u>
UT Optical Sciences

<u>Funded by</u>
UT

<u>Funding %</u>
University 100 %

<u>Start of the project</u>
2015

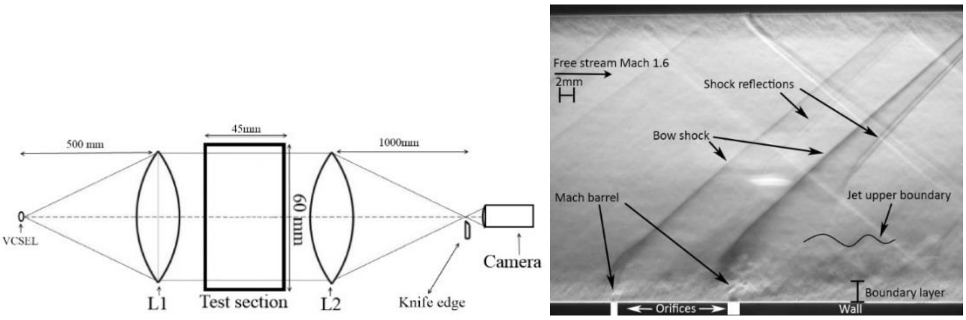
<u>Information</u>
C.H.Venner
c.h.venner@utwente.nl

<u>Project title</u>
Illuminating aspects of supersonic flow

<u>Project aim</u>
Development of short time pulse driven methodology, electronical and optical components for e.g. highly time accurate Schlieren imaging, and use of these methods to study fundamental aspects and time varying phenomena and structures in supersonic flows.

<u>Progress</u>
An innovative Power VCSEL driven Schlieren has been developed and used to visualize a system for a cascaded injection in a supersonic cross flow. The flow observed in the supersonic wind tunnel of the University of Twente, is a cascaded dual tandem air injection transverse to a stream of Mach 1.6. Continuation of work with higher accuracy experiments and larger J ratios.

<u>2020 Publications</u>
de Maag, S., Hoeijmakers, H.W.M., Venner, C.H., Segerink, F.B., and Offerhaus, H.L., Power VCSEL driven Schlieren visualisation for cascaded injection in supersonic flow. AIAA SCITEX, Orlando, jan 2020, AIAA 2020-0040. DOI 10.2514/6.2020-0040.



Left: Schematic of Schlieren set-up. Right: Example of Schlieren image obtained for Mach = 1.6, diameter orifices 1 mm (upstream) and 2 mm (downstream). Visualised are the tandem jets 20 mm apart with a momentum ratio of J=1.37, each featuring a Mach barrel, from the orifices, the bow shocks induced by the jets, the boundary layer along the walls and their interaction with the shocks. Also visible are Mach waves originating from small slope discontinuities of the walls.

UT – Engineering Technology - Engineering Fluid Dynamics

Project leaders
C.H.Venner
H.W.M.Hoeijmakers

Research theme
Complex dynamics of fluids

Participants
L. de Santana, L Groot Koerkamp, S. Stramigioli, G.A. Folkertsma.

Cooperation
UT Robotics and Mechatronics, Clear Flight Solutions.

Funded by
UT

Funding %
University 100%

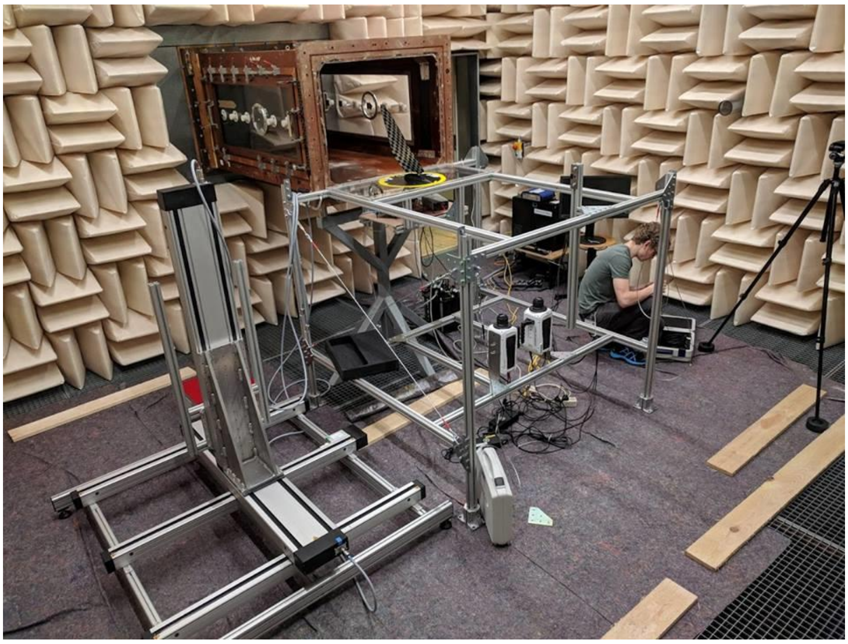
Start of the project
2012

Information
CH Venner
c.h.venner@utwente.nl

Project title
Fluid Dynamics of Nature Inspired Configurations

Project aim
Investigate experimentally and numerically the flow about nature-inspired configurations, such as Robot Birds. These nature-inspired copies of real birds combine lift and propulsion by flapping wings. The peregrine falcon Robird is a nature and animal friendly means of bird control around airports with many other possible sustainable applications. In the project (scale-models of) this robotic bird, and other flapping flight configurations are investigated in the wind tunnel and numerically to unveil natures secrets.

Progress
Started development of computational model based on lifting line theory. Setup of new PIV experiments. Extending collaborations.



Setup of flexible robird flapping wing in windtunnel for PIV measurements

UT – Engineering Technology - Thermal Engineering

Prof.dr.ir. G Brem
Prof.dr.ir. ThH van der Meer (em.)

The research activities of the laboratory of Thermal Engineering concentrate on thermal processes for heat and power generation in industrial and domestic applications from the disciplines thermodynamics, fluid mechanics, heat transfer, chemistry and acoustics. The research aims at increasing share of the use of renewable energy, and a more efficient and clean utilization of fossil fuels. The projects are organized around the themes turbulent combustion, thermo-acoustics and transient heat transfer.

The research theme Turbulent Combustion and Thermo-acoustics is related to issues on ignition, extinction, flame stability, pollutant formation (NOx and soot), combustion noise and its interaction with the combustion chamber structure. Numerical models are developed for turbulent combustion including compressibility and heat transfer. These models are implemented in commercially available software (CFX) and in academic code (ALYA). Experimental research is performed in atmospheric and in pressurized combustors using as Laser Induced Fluorescence and Raman/Rayleigh spectroscopy for in-flame measurements of temperature and species concentrations, acoustic measurements applied to Flame Transfer Functions. For particulate emissions (soot) a system is available that can measure particle size distributions of 2-200 nm. Applications are: gas turbine engines, boilers and furnaces.

The research theme Instationary Heat Transfer is related to heat transfer in piston compressors, heat transfer and chemical conversion in pulsed compression reactors and new materials for enhanced heat transfer in regenerators and heat exchangers. Applications are: thermo-acoustic heat pumps and engines and magneto-caloric heat pumps and coolers. Numerical models are developed for the multi-physics phenomena in these systems supported by experimental research.

UT – Engineering Technology - Thermal Engineering

Project leaders
Jim Kok

Research theme
Complex dynamics of fluids

Participants
J.B.W. Kok, S. Navarro Arredondo

Cooperation
General Electric, Karlsruher Institut für Technologie Cambridge University, AMINES, ANSYS, Technische Universität München, CERFACS and Safran HE

Funded by
Marie Skłodowska-Curie Actions (MSCA) Innovative Training Networks (ITN) H2020-MSCA-ITN-2017

Funded %
EU 100 %

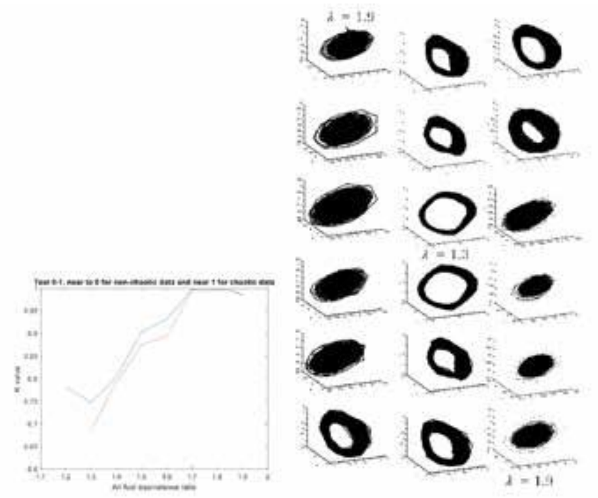
Start of the project
2017

Information
J.B.W. Kok
j.b.w.kok@utwente.nl

Project title
MAGISTER: Machine learning for advanced gas turbine injection systems to enhance combustor performance. Characterization of acoustically (un)forced spray flames at elevated pressure and preheated air

Project aim
Through experiments in two premix combustors of different characteristics (atmospheric pressure, open end, gas fuel and high pressure, preheated air, liquid fuel), acoustic characterization, qualitative of the flame and heat release are look upon from combustion under different parameters, looking towards thermoacoustic instabilities and their nonlinear behaviour is studied.

Progress
Experiments in an atmospheric, open-end combustor and nonlinear analysis through three phase portraits, chaos analysis. Implementation of the UT burner. Preparing a high pressure, liquid fuel combustor for operation to characterize its behaviour in the lean regime going towards a richer combustion.



Chaos test for the atmospheric pressure combustor from maximum air to fuel equivalence ratio to minimum and back. Three phase portraits generated corresponding to each point from maximum to minimum and back.

UT – Engineering Technology - Thermal Engineering

Project leaders
J.B.W. Kok

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Universiteit Twente (UT), General Electric Deutschland (GEDE), Technische Universitaet Muenchen (TUM), Karlsruher institut fuer Technologie (KIT), University of Cambridge (UCAM), CERFACS, ARMINES Paris Tech, Safran (ST), Safran Helicopter Engines (SHE), ANSYS France SAS (ANSYS)

Cooperation
GE, Rolls Royce, KLM, Stanford University, Georgia Institute of Technology, FDX Fluid Dynamix, Shell Research

Funded by
Marie Skłodowska-Curie Actions (MSCA) Innovative Training Networks
(ITN) H2020-MSCA-ITN-2017

Funded %
EU 100 %

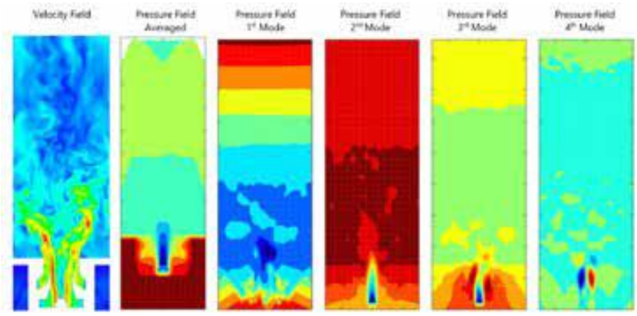
Start of the project
2017

Information
J.B.W. Kok
j.b.w.kok@utwente.nl

Project title
MAGISTER: Machine learning for advanced gas turbine injection systems to enhance combustor performance

Project aim
LES of Spray flames for the in-house designed swirl burner. Application of ML techniques such as POD and DMD in characterizing combustion dynamics.

Progress
Design of the UT burner. Preparing a computational mesh and case setup for LES of the burner. Applying POD and DMD to characterize the dominant flow patterns and frequencies.



Study of droplet distributions and comparison to experimental results forbatomization process. Using POD to define a proper droplet distribution.

UT – Engineering Technology – Multiscale Mechanics

Prof.dr.rer.nat. S Luding
Prof.dr. AR Thornton

The Multi-Scale Mechanics group (MSM) is part of the department Thermal and Fluids Engineering (TFE) in the Faculty of Engineering Technology (ET) at the University of Twente, as well as member of the research institute MESA+. The group studies the static and dynamic properties of dry and wet granular materials, as well as fluids and solids in general.

The main research areas include: flowing particles as well as sticky powders; segregation versus mixing; non-Newtonian fluids and rheology; macro-molecules; self-healing materials; wave-propagation in disordered media like soil; composite solids like concrete or asphalt; and avalanche flows of grains, soil or snow. A wide range of length and time scales characterises the relevant physical processes in these systems. Multi-scale mechanics means that at each length scale, the question arises how the mechanics at that level is determined by the properties of the underlying level, and how, in turn, the current level affects the next level(s).

Generally the above systems are modelled on three levels:

- Microscopic where interactions between individual grains or atoms are considered. Here, the deformation behaviour of the granule/atom/molecule with contact/interaction mechanics and physics on the nano-meter scale determines the dynamics and statics of the many particles.
- Mescoscopic where collections of upscaled grains or coarse-grained atoms with effective properties are modelled. At this level the discrete nature is still retained but ‘pseudo-particles’ approximate groups of grains or atoms. Mesoscopic models use the small-scale information to formulate effective contact laws. By decreasing the degrees of freedom, larger systems and longer times can be simulated allowing the study or large/slower processes like shear localisation, structure formation, self-assembly of patchy colloidal particles and proteins, or asphalt on the stone-bitumen scale of long-term use to be studied.
- Macroscopic (or bulk) where the materials are approximated by continuum models with an associated rheological model. On this level a discrete, granular material can behave like a complex fluid involving anisotropy and non-Newtonian rheological features.

The group employs a combination of theory, experiments and advanced numerical simulations to understand the multiple scales and their intricate couplings; considerable work is undertaken on accurate micro-macro methods to traverse the different levels of the modelling hierarchy. This powerful combination allows the group to develop a comprehensive fundamental understanding of the studied systems. Finally, the MSM group is a leader in open-source development and leads the open-source code particle solver: mercuryDPM (mercuryDPM.org). Additionally, it uses and contributions to many other packages like the opensource FEM solver oomph-lib. This development work in 2015 led to the foundation of a spin-off company mercuryLab.org whose aim is to put these powerful open-source software packages into the hands of industry.

UT – Engineering Technology - Multiscale Mechanics

Project leaders
A.R. Thornton, W.K. den Otter

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
R.S. Sitlapersad

Cooperation
Utrecht University
Jawaharlal Nehru Centre for Advanced Scientific Research (Bangalore, India)

Funded by
NWO

Funded %
NWO 100 %

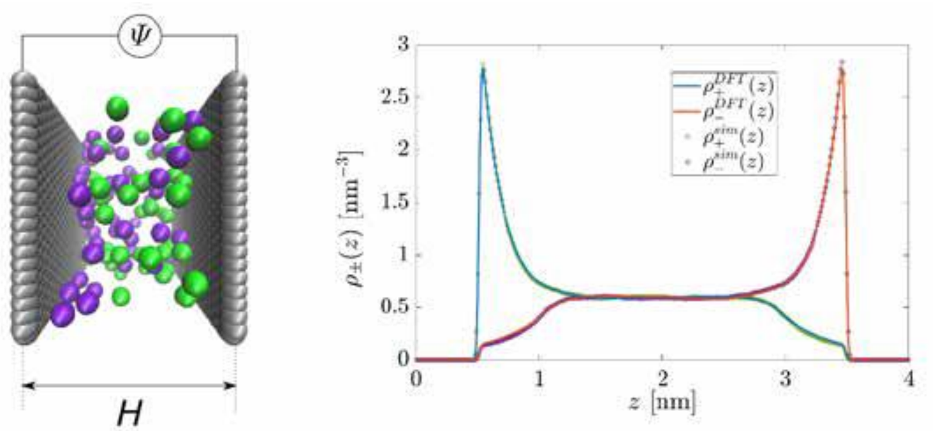
Start of the project
2018

Information
Ranisha Sitlapersad
r.s.sitlapersad@utwente.nl

Project title
Optimized energy storage using ionic electrolytes

Project aim
Energy storage is of the utmost importance to stabilize power grids relying on intermittent renewable energy sources. A promising alternative to current commercial energy storage devices is devices based on a new class of electrolytes, namely ionic liquids. The aim of this project is to explore and optimize ionic liquid energy storage technologies through molecular dynamics (MD) simulations.

Progress
We investigated the electric double layers formed by an aqueous electrolytes in a capacitor, see the figure, using Brownian Dynamics (BD) simulations of the ions. We explored the effects of the ion concentrations, ion size, ion valencies, electrode potentials and the slid width on the distributions of the ions and the (differential) capacitance. An excellent agreement was found with the density functional theory (DFT) calculations by our collaborators in Utrecht, thereby establishing that the latter can be applied to these systems.



Left: An aqueous electrolyte, with cations in purple and ions in green, between two electrodes, in grey, at different potentials. Right: the ion density profiles obtained with BD and DFT.

UT – Engineering Technology - Multiscale Mechanics

Project leaders
T. Weinhart

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
H. Cheng

Funded by
STW-NWO-AES VIDI

Funded %
NWO 100%

Start of the project
2019

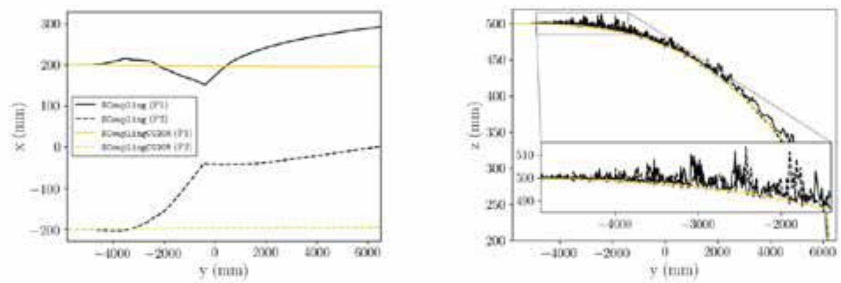
Information
H Cheng
h.cheng@utwente.nl

Project title
Virtual prototyping of particulate processes (ViPr) – Design and optimization via multiscale modeling and rapid prototyping

Project aim
The behaviour of granular materials and their interactions with deformable structures are difficult to predict and highly relevant to many fields including civil, geotechnical, and mechanical engineering. Classically, the bulk of granular particles are modelled as continua where a constitutive equation is essential to predicting the bulk behaviour. We aim to implement a concurrent multi-scale framework by coupling discrete particle (DPM) and finite element (FEM) submodels. Two approaches are developed, namely surface coupling and volume couple using Coarse-Graining (CG) for micro-macro transition.

Progress
The simplest test case that examines the full capability of surface coupling is a single particle rolling over an elastic cantilever. During this dynamic process, both the smoothness of particle-convex/concave wall interaction and the surface coupling algorithm are tested. The benefit of a CG-enriched formulation for FEM-DEM surface coupling is illustrated by comparing the trajectories of the particles when they slide on the elastic cantilever. The figures a and b respectively show the trajectories of the two particles in the x-y and y-z plane. The results show that with a CG width of 20R (5dX), not only is the symmetry of the particle trajectories in the x-y plane recovered, but also the oscillation in both particle trajectories in the z direction is removed entirely.

2020 Publications
Cheng, H. , Luding, S., Saitoh, K. , & Magnanimo, V. (2020). Elastic wave propagation in dry granular media: effects of probing characteristics and stress history. International journal of solids and structures, 187, 85-99. https://doi.org/10.1016/j.ijsolstr.2019.03.030.



Trajectories of the two particles sliding on the cantilever under gravity; (a) Particle trajectories in the x-y plane (b) Particle trajectories in the y-z plane

UT – Engineering Technology - Multiscale Mechanics

Project leaders
A.R. Thornton, T. Weinhart,
I. Ostanin

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
S. Pourandi

Cooperation
TU/e, TU Delft

Funded by
NWO

Funded %
NWO 100%

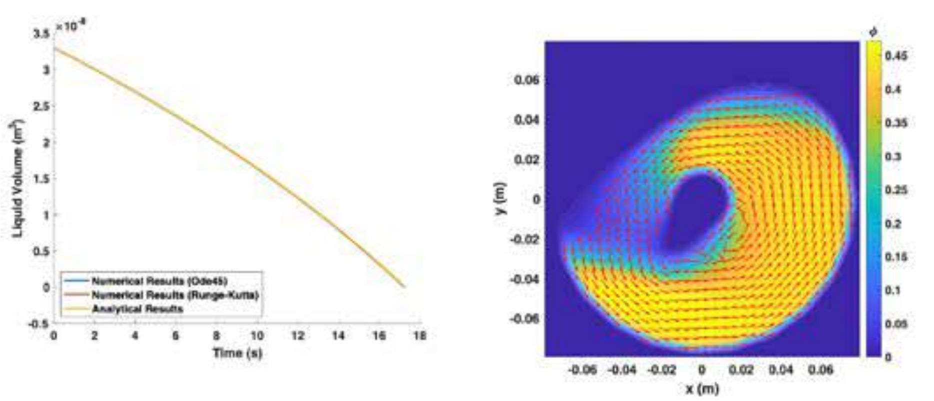
Start of the project
2020

Information
Sahar Pourandi
s.pourandi@utwente.nl

Project title
CFD -DEM study of powder mixing in systems with complex particle interactions in moving devices below the fluidization limit WdO: Simulation of powder mixing under industrial conditions

Project aim
This project contributes to the Industrial Dense Granular Flow consortium, in a collaboration with Delft and Eindhoven. We will perform coupled CFDDEM simulations, using open-source software MercuryDPM and FoxBerry, to study powder filling and mixing in complex industrial systems. To achieve this, we will couple the momentum, heat, and liquid content equations. Optimizing the code's parallelism by load balancing will allow large-scale simulations. Experimental results from our partners will be used to validate the model. We will study mixing quality, dynamics, and power consumption.

Progress
The focus is on developing MercuryDPM and simulating industrial processes with the CFD-DEM coupled software. In the first couple of months of this project, we have implemented a new class of particles in MercuryDPM which have both liquid and thermal properties; this is required for the coupled CFD-DEM code. We also implemented a new evaporation model in MercuryDPM and validated the model against analytic results. We have applied MercuryCG to an industrial process and extracted simple continuum fields.



(Left) Comparing numerical and analytical results for the evolution of moisture content in a particle. (Right) The coarse-grained solid volume fraction (colour map) and flow field (arrows) in an industrial process.

UT – Engineering Technology - Multiscale Mechanics

Project leaders
W. K. den Otter, S. Luding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
T. Hulikal Chakrapani, A. Jarray

Cooperation
Canon, TU/e

Funded by
NWO/Canon

Funded %
NWO 50%
Industry 50%

Start of the project
2017

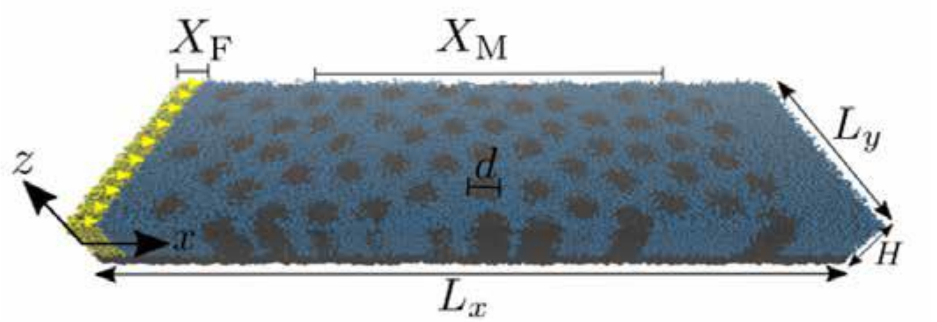
Information
Thejas Hulikal Chakrapani
t.hulikalchakrapani@utwente.nl

Project title
Mesoscopic modeling of ink imbibition in paper

Project aim
Water-based ink for ink-jet printing on paper contains several components in addition to pigment particles. With the water content of the droplet gradually decreasing through evaporation and imbibition into the paper, a solidifying deposit is left at the surface of (and partly inside) the paper. A good ink produces a well-defined and lasting deposition of pigment particles. The aim of the project is to study the evolution of ink droplets deposited on paper at the mesoscopic level. The project entails modelling both the ink, a complex liquid, and the paper, a complex solid.

Progress
Studying imbibition dynamics in paper is a challenging task due to the complexities of the substrate. We studied the permeability of porous media composed of irregularly arranged cylindrical pillars confined between two flat walls, see the figure. Multi-body Dissipative Particle Dynamics simulations yielded velocity profiles in agreement with Brinkman's theory – showing that this theory works well, though the fit parameters deviate from their expected values. The calculated permeabilities of four pillar configurations, ranging in porosity from 55% to 85%, agreed well with those measured using microfluidic chips etched with the same pillar configurations.

2020 Publications
T. Hulikal Chakrapani and W.K. den Otter, Capillary imbibition of binary fluid mixtures in nanochannels, Langmuir 2020, 36, 12712–12722.



Snapshot of the 3D simulation set-up for flow through a wall-bounded irregular array of pillars, at a porosity of 55%. All pillars (black) have the same diameter and span the height between the two flat walls (not shown) bounding the flow cell at the top and bottom. The system is periodic in both in-plane directions. The flow of the fluid (blue) is maintained by a body force in the yellow region, resulting a pressure drop across the array of pillars.

UT – Engineering Technology - Multiscale Mechanics

Project leaders
T. Weinhart, S. Luding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
T. Plath

Cooperation
Bayer AG

Funded by
NWO

Funded %
NWO 100%

Start of the project
2020

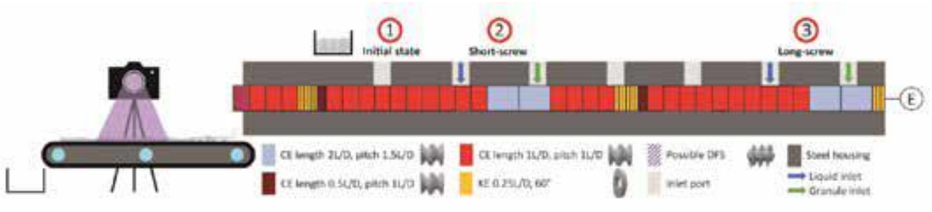
Information
Timo Plath
t.plath@utwente.nl

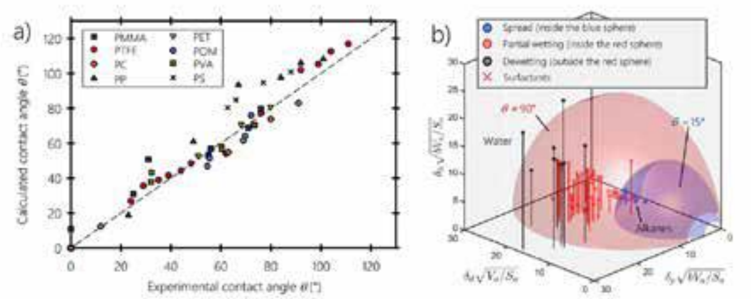
Project title
Heterogeneous multiscale modeling and rapid prototyping of twin-screw wet granulation processes

Project aim

- Enhance understanding of twin-screw wet granulation processes via experimental studies.
- Implement novel algorithms to describe the granulation behaviour in a wet granulation process.
- Develop a PBM-DEM heterogeneous multiscale model to rapidly optimize twin-screw designs for specific applications.
- Develop an additive manufactured lab-scale experimental setup for twinscrew wet granulation.

Progress
To gain a better understanding of twin-screw wet granulation, a parametric study was undertaken using a highly accurate experimental setup, see the figure. The results show that residence time distributions can be used to detect process instabilities. Current research focuses on implementing novel algorithms to describe the granulation. MercuryDPM is utilized for discrete particle simulations while a population balance model will be solved via a novel quadrature method of moments. In future work, we will combine both models into a coupled multiscale model and develop a purely additive-manufactured experimental setup for twin-screw wet granulation.



UT – Engineering Technology - Multiscale Mechanics	
Project leaders S. Luding, W.K. den Otter, H. Wijshoff, A. A. Darhuber Research theme Mathematical and computational methods for fluid flow analysis Participants A. Jarray Cooperation TU/e, TU Delft Funded by NWO Funded % NWO 100% Start of the project 2017 Information A. Jarray a.jarray@utwente.nl	Project title Systematic approach to predicting wettability using molecular dynamics simulation Project aim Wettability of ink on paper is a key phenomenon underlying the inkjet printing process. The goal of this project is to establish a fast and reliable method to predict the degree of wettability of liquids on substrates. Progress We developed a fast and efficient approach to predict the wettability and spreading of liquids on polymeric substrates. First, a molecular dynamics parameterization is proposed for the calculation of the solubility parameter for 74 compounds including surfactants typically used in inkjet printing. Then, we introduce a molecular geometrical factor to relate the solubility parameter to the surface tension, obtaining estimates in remarkable agreement with experiments. By using a modified Young–Fowkes equation, the contact angles of liquids on various polymeric substrates are determined and their dependence on the hydrogen bonding, dispersion and polar contribution of the solubility parameter are investigated. We find that wetting properties are obtained with a good accuracy when taking into account the hydrogen-bonding and polar interactions in the geometric sum of the solubility parameter. Based on these findings, a 3D wetting space is proposed to evaluate liquids wettability and judge their suitability for specific substrates. This will enable easy formulation of liquids with wettability tailored for a particular surface and application. 2020 Publications 1. Jarray A., Wijshoff H., Jurriaan L. A., and den Otter W. K., Systematic approach for wettability prediction using molecular dynamics simulations, Soft Matter16, 4299-4310 (2020). 2. Faasen D.P., Jarray A., Zandvliet H.J.W., Kooij E.S., and Kwiecinski W., Hansen solubility parameters obtained via molecular dynamics simulations as a route to predict siloxane surfactant adsorption, Journal of Colloid and Interface Science 575, 326–336 (2020).  *a) Simulation-based versus experimental contact angle of several liquids on various polymeric substrates. PMMA: polymethyl methacrylate, PTFE: polytetrafluoroethylene, PP: polypropylene, PC: polycarbonate, PET: polyethylene terephthalate, POM: polyoxymethylene, PS: polystyrene and PVA: polyvinyl alcohol.* *b) Wettability spheres with liquids, represented by dots, inside the blue (red) sphere forming contact angles of less than 15° (90°) on PTFE.*

UT – Engineering Technology - Multiscale Mechanics

Project leaders
A.R. Thornton, T. Weinhart, S. Luding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M.J. Post

Funded by
NWO-STW

Funded %
NWO 100%

Start of the project
2017

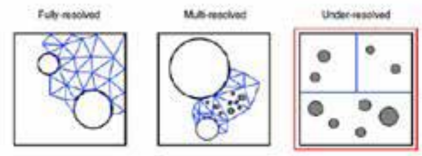
Information
M.J. Post
m.j.post@utwente.nl

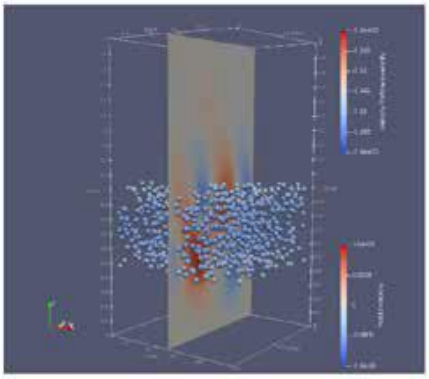
Project title
Multi-scale modeling of agglomeration – Application to selective laser sintering

Project aim
Powder agglomeration is a widely encountered phenomenon in many industries such as pharmaceutics, additive manufacturing, food processing and so forth. A fast and accurate numerical model of agglomeration is therefore highly desired. We have developed a multiscale approach to extract the macroscopic bulk behaviour of powders from the microscopic particle-particle interactions. The aim of this project is to extend this model to also include particle-fluid interactions. The extended model will specifically be used for applications that involve high pressure powder agglomeration, a.k.a. tableting.

Progress
The particle-fluid coupling is incorporated using two open-source codes: MercuryDPM as the particle solver and oomph-lib as the finite element method (FEM) fluid solver. The current work focusses on under-resolved coupling, using the Anderson-Jackson formulation, with the particle volume fraction extracted from discrete particle simulation; multi-resolved coupling is planned later, see the left figure. This will be applied to the die-filling process and powder compaction to study the tableting process. In the current state of the code, 3D and two-way coupled, the behaviour of a particle-fluid system can be studied for example in particle settling, see the right figure.

2020 Publications
T. Weinhart, L. Orefice, M. Post, M.P. van Schrojenstein Lantman, I.F.C. Denissen, D.R. Tunuguntla, J.M.F. Tsang, H. Cheng, M.Y. Shaheen, H. Shi, P. Rapino, E. Grannonio, N. Losacco, J. Barbosa, L. Jing, J.E. Alvarez Naranjo, S. Roy, W.K. den Otter and A.R. Thornton, Fast, flexible particle simulations – an introduction to MercuryDPM, Comput. Phys. Comm. 249, 107129 (2020).





Multi-resolution coupling for polydisperse packings Two-way coupled simulation of particles settling in a fluid

UT – Engineering Technology - Multiscale Mechanics

Project leaders
T. Weinhart, S. Luding

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M.Y. Shaheen

Funded by
STW

Funded %
NWO 100%

Start of the project
2017

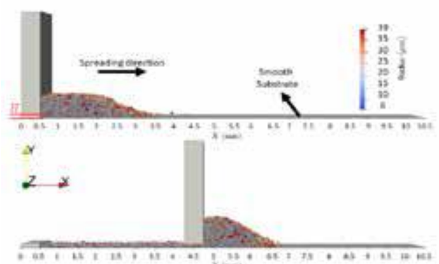
Information
M.Y. Shaheen
m.y.shaheen@utwente.nl

Project title
Multi-scale modeling of agglomeration - Application to selective laser sintering and tableting

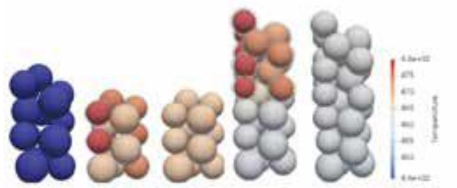
Project aim
The aim of this project is to quantitatively predict agglomeration of fine-grained material through compression and/or heating. This requires the development and calibration of new multi-scale particle models for fine powders and their subsequent application to processes in, among others, additive manufacturing and the production of pharmaceutics. Selective laser sintering (SLS) and tableting are chosen as the prototype processes to which the new techniques are applied first. Especially challenging and novel aspects are the process dynamics, both the kinetics and rate-dependence, as well as the coupling between the macro and micro scales.

Progress
Laser Powder Bed Fusion (LPBF) is an additive manufacturing technology. One of the main challenges is the optimization of process parameters to achieve high-quality products, currently carried out by costly experimental trials. We are developing a computational model that will reduce the number of trials, thereby reducing manufacturing costs and allowing processability predictions for new materials. This is achieved by comparing simulations of individual production steps against experimental data on the same step, to advance the development of accurate predictive models. Two steps are illustrated below, powder spreading in the upper figure and contact melting in the lower figure.

2020 Publications
T. Weinhart, L. Orefice, M. Post, M.P. van Schrojenstein Lantman, I.F.C. Denissen, D.R. Tunuguntla, J.M.F. Tsang, H. Cheng, M.Y. Shaheen, H. Shi, P. Rapino, E. Grannonio, N. Losacco, J. Barbosa, L. Jing, J.E. Alvarez Naranjo, S. Roy, W.K. den Otter and A.R. Thornton, Fast, flexible particle simulations – an introduction to MercuryDPM, Comput. Phys. Comm. 249, 107129 (2020).



Simulation of powder spreading in a powder-based 3D printer



Simulation snapshots of laser sintering in a powder-based 3D printer. From left to right: deposition of a layer of particles, partial melting during laser heating, the formation of bonds during cooling, deposition and heating of a second layer of particles, and bonding during cooling.

UT – Engineering Technology - Multiscale Mechanics

Project leaders
V. Magnanimo, S. Luding

Research theme
Complex mechanics of granules and powders

Participants
H. Shi (UT), A. Jarray (UT)

Cooperation
Tech. Univ. Braunschweig, Univ. of Edinburgh, Univ. of Greenwich, Univ. of Liége

Funded by
EU ITN T-MAPPP

Funded %
EU 100 %

Start of the project
2014

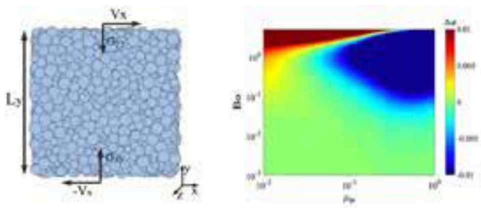
Information
Hao Shi
h.shi-1@utwente.nl

Project title
Discrete element method simulations and experimental validation of tableting processes

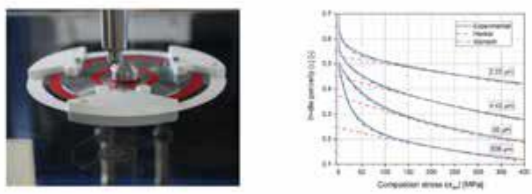
Project aim
- Development of rheological model of granular and powder flow using DEM simulation;
- Investigate multi-scale deformation behaviour of tableting/powder compaction process;
- Identify the flow regimes of powder/granular systems.

Progress
The work is finalized in line with the objectives described above. We have achieved the first objective by conducting simple shear and split bottom shear DEM simulations and enriched the existing rheological model by including influences from both macro and microscales, i.e., shear rate, softness, friction, and cohesivity. A systematic experimental study on powder compaction is carried out and various models are tested to fulfill the second objective. For the third objective, we have conducted powder deformation/flow tests that cover quasi-static to dynamic regimes and made a first attempt in linking powder bulk friction across these flow regimes.

2020 Publications
1. Shi, H., Roy, S., Weinhart, T., Magnanimo, V., & Luding, S. (2020). Steady state rheology of homogeneous and inhomogeneous cohesive granular materials. Granular matter, 22(1), 1-20.
2. Cabiscol, R., Shi, H., Wünsch, I., Magnanimo, V., Finke, J. H., Luding, S., & Kwade, A. (2020). Effect of particle size on powder compaction and tablet strength using limestone. Advanced Powder Technology, 31(3), 1280-1289.
3. Shi, H., Lumay, G., & Luding, S. (2020). Stretching the limits of dynamic and quasi-static flow testing on cohesive limestone powders. Powder technology,
367, 183-191.



(left) Simple shear DEM setup. (right) Phase diagram on volume fraction influenced by micro friction and cohesion.



(left) Styl'One Compaction Simulator. (right) Porosity evolution during compation

UT – Civil Engineering - Water Engineering and Management

Prof.dr. SJMH Hulscher
Prof.dr. KM Wijnberg

The work of the Water Engineering and Management (WEM) group was originally mainly devoted to the modelling of sand waves on the seabed. First it was shown, that the seabed patterns in the North Sea can be explained as free instabilities of the seabed. Subsequently, the modelling of sand waves was extended and refined. The group has worked on a scientific and practical tool for fully nonlinear modelling of sand waves. Over the last five years, the offshore morphodynamic work has broadened to rivers, coasts and blue-ice. Alternate bars in a flume were compared with (Ginzburg-)Landau-type models.

The group was the first to explore data assimilation for morphodynamic predictions. Further work concentrated on using data assimilation to combine field data with sand wave amplitude models for maintenance dredging management of navigation channels and sand wave-related pipeline problems. Also, North Sea data were analysed and a new bed mode, called long bedwaves, was discovered. The origin of nearshore bars was addressed. A method was developed for modelling human interferences in a morphodynamic setting. This has opened perspectives for a new approach towards modelling large-scale sand mining in shallow seas. A project for developing tools for evaluation of human interference in the North Sea for optimal management of the seabed started recently and sediment transport concerning near-shore sand pits is being investigated. Since 2000 the group has studied the use of morphodynamical models in a societal context. Recently, a method for decision making based on quantitative information including uncertainties was developed in the multidisciplinary project Flyland, which opens the field of designing an assessment framework for appropriate modelling.

UT – Civil Engineering – Water Engineering and Management

Project leaders
Prof dr. S.J.M.H. Hulscher
Dr. ir. T.M. Duong

Research theme
Complex dynamics of fluids

Participants
Ir. R.W.A. Siemes

Cooperation
University of Twente, Rijkswaterstaat, Van Oord, Boskalis, Deltares, NOIZ

Funded by
NWO

Funded %
University 10 %
NWO 50 %
Industry 30 %
Scholarships 10 %

Start of the project
2020

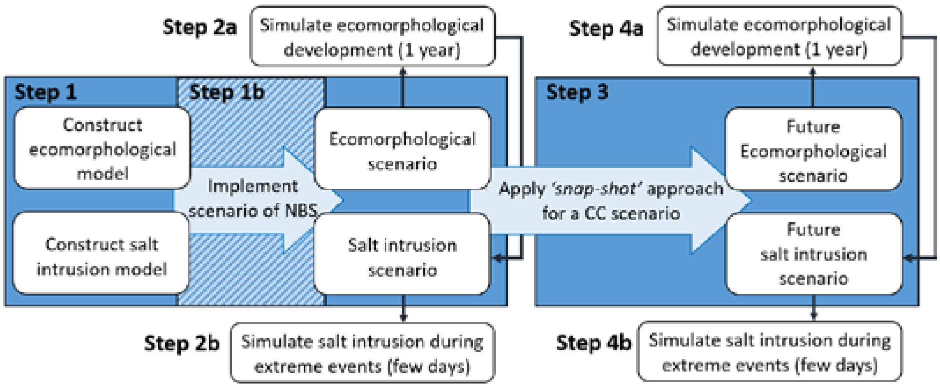
Information
Rutger Siemes
r.w.a.siemes@utwente.nl

Project title
Efficient modelling of long-term nature-based solutions for estuarine salt intrusion

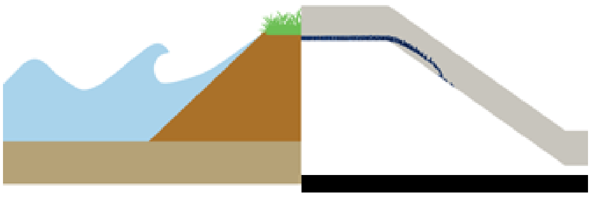
Project aim
The objective is to develop a computational efficient modelling framework that is able to evaluate NBS to mitigate estuarine salt intrusion under long-term ecomorphological development, taking into account CC up to 2100.

Progress
Final development phase of idealized models in which the said framework is applied.

2020 Publications
Siemes, R. W., Borsje, B. W., Daggenvoorde, R. J., & Hulscher, S. J. (2020). Artificial Structures Steer Morphological Development of Salt Marshes: A Model Study. Journal of Marine Science and Engineering, 8(5), 326.



The proposed modelling framework, from constructing the models (step 1-1b), running scenarios for the present (step 2), ajusting the models for a climate change scenario using a 'snap-shot' approach (step 3) and again running the models for the future (step 4)

UT – Civil Engineering – Water Engineering and Management	
Project leaders prof. dr. S.J.M.H. Hulscher dr. J.J. Warmink Research theme Mathematical and computational methods for fluid flow analysis Participants V.M. van Bergeijk, MSc Cooperation Rijkswaterstaat, Ministerie van Infrastructuur en Water, Deltares, HKV, Royal Haskoning DHV, Arcadis, Witteveen + Bos, HillBlocks, Vechtstromen Funded by NWO TTW (Perspectief) Funded % NWO 100% Start of the project 2017 Information Vera van Bergeijk v.m.vanbergeijk@utwente.nl	Project title All-Risk: Modeling the effects of transitions on wave overtopping flow and dike cover erosion Project aim During storms, waves overtop the dikes and cause erosion at the landward side of the dike. The erosion due to wave overtopping can lead to a dike breach. Wave overtopping is one of the main cause of dike failure and transitions are potential weak spots in the dike cover. However, there is still a lot unknown about the wave overtopping process and its erosional effects. The goal of the project is to quantify and model the effect of transitions on overtopping flow and dike cover erosion. Progress We developed a numerical model to simulate the overtopping flow over the crest and the landward slope of grass-covered dike. A paper was published on the validation of the numerical model where the load on the dike cover was investigated. The effect of damages in the dike cover on the wave overtopping erosion is included in the analytical model. The analytical model was extended to a probabilistic framework to study the vulnerability of damaged dike profiles for overtopping. 2020 Publications 1. Verdonk, V.A., Van Bergeijk, V.M., Warmink, J.J., & Hulscher, S.J.M.H. (2020). Beyond inner slope instability: a method to quantify the residual dike strength by wave overtopping. Abstract from the 1st IAHR Young Professionals Congress. 2. Warmink, J. J., Van Bergeijk, V. M., Frankena, M., Van Steeg, P., & Hulscher, S. J. M. H. (2020). Modelling Transitions in Grass Covers to Quantify Wave Overtopping Erosion. In Coastal Engineering Proceedings 2020 (pp. 1–8) https://doi.org/10.9753/icce.v36v.papers.39 3. Van Bergeijk, V. M., Warmink, J.J., & Hulscher, S. J. M. H. (2020). Modelling of Wave Overtopping Flow over Complex Dike Geometries: Case Study of the Afsluitdijk. In Coastal Engineering Proceedings 2020 https://doi.org/10.9753/icce.v36v.papers.52 4. Van Bergeijk, V.M., Warmink, J.J. & Hulscher, S.J.M.H. (2020). Modelling the Wave Overtopping Flow over the Crest and the Landward Slope of Grass-Covered Flood Defences. Journal of Marine Science and Engineering. 8, 489. https://doi.org/10.3390/jmse8070489 5. Van Bergeijk, V. M., Warmink, J. J., & Hulscher, S. J. M. H. (2020). Hydrodynamic modelling of wave overtopping flow over grass-covered dikes. 36-36. Abstract from NCK Days 2020, Texel, The Netherlands. 6. Peters, M., van Bergeijk, V. M., van Zuylen , J. A., Warmink, J. J., & Hulscher, S. J. M. H. (2020). The effect of the slope angle on the failure of the grass revetment due to wave impact. 84-84. Abstract from NCK Days 2020, Texel, The Netherlands. 

UT – Civil Engineering – Water Engineering and Management

Project leaders
S.J.M.H. Hulscher

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
R.M.J. Schielen
A. Bomers

Cooperation
University of Utrecht, Rijkswaterstaat, Deltares

Funded by
Rijkswaterstaat, Deltares

Funded %
NWO 85%
Industry 15%

Start of the project
2015

Information
Anouk Bomers
a.bomers@utwente.nl

Project title
Floods of the past – Design for the future

Project aim
We aim to contribute to improvement of the current estimates of discharge frequency relationships for the Lower Rhine, which is the primary tool for flood risk management in the lower Rhine delta and directly contributes to the first layer of protection. This study focusses on the parametrization of hydraulic characteristics of the river and floodplains of the various historical years and use this as input for the hydraulic models to reconstruct past flood magnitudes and hence extent the historic time series of measured discharges.

Progress
In 2020, a hydraulic paleo-model was developed with which it was possible to gain insight in the flow patterns of the Rhine river before human interventions. Furthermore, insight was gained in the most extreme flood magnitudes that has occurred in the past.

2020 Publications
1. Bomers, A., Schielen, R. M. J., Hulscher, S. J. M. H. (2020) Reply to “Consequences of dike breaches and dike over flow in a bifurcating river system”. Natural Hazards, 103(2), 1633:1637. DOI:10.1007/s11069-020-04162-x
2. Dierx, J. R., Bomers, A., Cohen, K. M., Hulscher, S. J. M. H. (2020). Modelling erosion of pre-embankment breaches in coversand ridges in the IJssel floodplain due to major flood events in medieval times. Proceedings of the 1st IAHR Young Professionals Congress. ISBN: 978-90-82484-6-63.
3. Bilgili, E., Bomers, A., Van Lente, G. J. W., Huthoff, F., Hulscher, S. J. M. H. (2020). The influence of a grid structure on hydraulic river modelling outcomes of schematised river meanders. Proceedings of the 1st IAHR Young Professionals Congress. ISBN: 978-90-82484-6-63.
4. Bomers, A., Hulscher, S.J.M.H., Van der Meulen, B., Schielen, R.M.J. (2020). Historic flood reconstructions for a safer future: The use of three types of surrogate models. River Flow 2020: Proceedings of the 10th Conference on Fluvial Hydraulics, Delft, The Netherlands.

UT – Civil Engineering – Water Engineering and Management

Project leaders
Dr. ir. E.M. Horstman

Research theme
Complex dynamics of fluids

Participants
Dr. ir. E.M. Horstman, Dr. ir. P.W.J.M. Willemsen, Ir. R. Gijsman

Cooperation
NIWA New Zealand, NUS Singapore

Funded by
NWO VENI

Funded %
NWO 100%

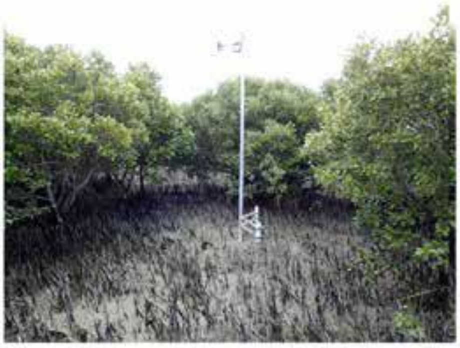
Start of the project
2018

Information
E.M. Horstman
e.m.horstman@utwente.nl

Project title
Mangrove-RESCUE : Mangrove Resilience for Enhanced Safety of Coastal Urbanisations and Environments

Project aim
Mangrove ecosystems shelter tropical and subtropical shorelines. Their natural resilience allows them to recover from erosion events and to adapt to changing conditions. Mangrove-RESCUE aims to acquire a mechanistic understanding of mangrove resilience and the contribution of mangroves to coastal safety. To obtain the required quantitative insights into how various factors affect biophysical interactions in mangroves, this project combines field measurements and numerical simulations of the processes that influence this resilience. The resulting model should enable the long-term prediction and protection of mangrove development and their contribution to coastal safety.

Progress
Field studies at mangrove sites in Singapore and New Zealand are ongoing, monitoring bed level changes and concurrent tides and waves. Additional characteristics of the bed and vegetation at these sites have been quantified in previous field visits. These combined observations will be used to analyze and explain erosion and deposition patterns through time and across the different mangrove sites. Newly obtained insights and data will be used for the development of a numerical model that can simulate the long-term bio-morphodynamic development of mangroves in different settings and under changing conditions.



UT – Civil Engineering – Water Engineering and Management

Project leaders Prof. dr. S.J.M.H. Hulscher, Prof. dr. T.J. Bouma Research theme Complex dynamics of fluids Participants Ir. P.W.J.M. Willemsen (PhD candidate), Dr. ir. B.W. Borsje (Daily supervisor) Cooperation NIOZ Yerseke, TuDelft, Deltares Funded by Deltares, Rijkswaterstaat, Van Oord, Boskalis, WNF, Hogeschool Zeeland, NWO-ALW, NIOZ, University of Twente. Funded % University 40% NWO 60% Start of the project 2015 Information Pim W.J.M. Willemsen p.willemsen@utwente.nl	Project title BE SAFE : Bio-Engineering for Safety using vegetated foreshores: long-term biogeomorphology Project aim We aim to develop new methods to assess how, and how much vegetated foreshores can contribute to flood risk reduction. The project will lead to a better understanding of (uncertainties in) the functioning and stability of these ecosystems and the development of novel governance arrangements. This requires integration of knowledge from ecology, biogeomorphology, hydraulic engineering, and governance. Progress Several achievements have been obtained in 2020: • Contributing to two peer-reviewed published journal papers (see below). • Finalizing a 2D model for modelling hydrodynamics, morphodynamics and vegetation development to simulate the development of the intertidal area. • Finalizing a manuscript on “Modelling decadal salt marsh development: variability of the salt marsh edge under influence of waves and sediment availability”, which is under review. • Writing a manuscript on “Modelling the influence of short-term high waves on the long-term stability of artificial salt marshes”. • Performing collaborative fieldwork in Singapore to measure hydrodynamics, morphodynamics and vegetation growth in mangroves. Measurements are continuously conducted. • Finalizing dissertation (see below) • Finalizing BE-SAFE project. 2020 Publications 1. M Hendriks, P Dankers, J Cleveringa, L Sittoni, P Willemsen, Kelly Elschot, MEB van Puijenbroek, MJ Baptist (2020). A large-scaled field experiment on salt marsh construction, NCK-Days 2020. 2. Pim W.J.M. Willemsen, Bas W. Borsje, Vincent Vuik, Tjeerd J. Bouma, Suzanne J.M.H. Hulscher (2020). Data presented in the paper: “Field-based decadal wave attenuating capacity of combined tidal flats and salt marshes”, 4TU. Centre for Research Data, https://doi.org/10.4121/uuid:4c25347f-f71e-466bbe3c-1fe8f8d8784c.

UT – Civil Engineering – Water Engineering and Management

Project leaders
Prof. dr. ir. S.J.M.H. Hulscher
Dr. ir. P.C. Roos
Dr. ir. B.W. Borsje

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
Dr.ir. J.H. Damveld

Cooperation
Delft University of Technology, NIOZ Yerseke, Boskalis, IMARES, RBINS OD Nature, Dienst der Hydrografie, ACRB, Deltares, Rijkswaterstaat
Dr. F. Heins (independent)

Funded by
NWO-ALW, Boskalis Westminster N.V.

Funded %
NWO 90 %
Industry 10%

Start of the project
2015

Information
Dr.ir. Johan H. Damveld
j.h.damveld@utwente.nl

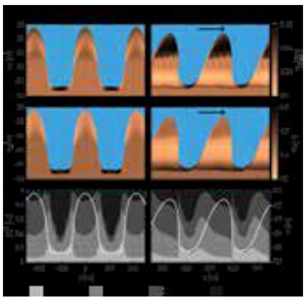
Project title
SAND BOX – Smart and sustainable design for offshore operations in a sandy seabed

Project aim
We aim to develop a coupled biogeomorphologic model to increase the understanding of the effects of benthic organisms on the behaviour of coastal bedforms. Using this model we intend to study the spatiotemporal evolution after anthropogenic interventions in shallow coastal seas, together with implementing the concept of ecological landscaping in offshore engineering. This requires integration of knowledge from ecology (collaboration with NIOZ Yerseke), fine sediment dynamics (collaboration with TU Delft) and geomorphology (this subproject).

Progress
Using Delft3D, the coupled feedbacks between sand waves and patches of the tube-building worm Lanice conchilega were studied. Also using Delft3D, sediment sorting processes over sand waves were studied. It turns out that both the organisms and fine sediments are more abundantly present in sand wave troughs, compared to the crests. These model results agree qualitatively with field observations. Conversely, both the presence of organisms and sediment sorting processes affect the morphology of sand waves. These aspects decrease the wave height and may change the wavelength. These findings were published in journal articles, and in a dissertation.

Dissertations
Damveld, J. H. (2020). The feedbacks among tidal sand waves, benthic organisms and sediment sorting processes. University of Twente. https://doi.org/10.3990/1.9789036550000.

2020 Publications
1. Damveld, J. H., Borsje, B. W., Roos, P. C., & Hulscher, S. J. M. H. (2020). Biogeomorphology in the marine landscape: modelling the feedbacks between patches of the polychaete worm Lanice conchilega and tidal sand waves. Earth surface processes and landforms, 45(11), 2572-2587. https://doi.org/10.1002/esp.4914.
2. Damveld, J. H., Borsje, B. W., Roos, P. C., & Hulscher, S. J. M. H. (2020). Horizontal and vertical sediment sorting in tidal sand waves: modeling the finite-amplitude stage. Journal of Geophysical Research: Earth Surface, 125(10), [e2019JF005430]. https://doi.org/10.1029/2019JF005430.
3. Damveld, J., Borsje, B. W., Roos, P., & Hulscher, S. (2020). Sediment sorting in tidal sand waves fields: the internal structure revealed?. Abstract from EGU-General Assembly 2020, Vienna, Austria. https://doi.org/10.5194/egusphere-egu2020-13679.
4. Damveld, J. H. (2020). The use of cameras for seabed monitoring: A North Sea example. ConcepTueel, 29(3), 22-25. https://issuu.com/concept27/docs/concepttueel_edition_3_2020_web_version_2.
5. Damveld, J. H., Roos, P. C., Borsje, B. W., & Hulscher, S. J. M. H. (2020). Tidal sand waves and benthic organisms: observations and modelling of biophysical interactions. 15-15. Abstract from NCK Days 2020, Texel, Netherlands. https://www.nck-web.org/boa-2020/319-tidal-sand-waves-and-benthicorganisms-
observations-and-modelling-of-bio-physical-interactions.



UT – Civil Engineering – Water Engineering and Management	
Project leaders Prof. dr. ir. S.J.M.H. Hulscher Dr. ir. P.C. Roos Research theme Mathematical and computational methods for fluid flow analysis Participants Ir. W.M. van der Sande Cooperation University of Twente, Rijkswaterstaat Van Oord, Boskalis, Deltares, Wageningen University & Research Funded by NWO Funded % University 5% NWO 60% Industry 35% Start of the project 2019 Information Wessel van der Sande w.m.vandersande@utwente.nl	Project title Idealized modeling of estuarine sand dunes: understanding natural and human-induced dynamics Project aim We will develop a process-based morphodynamic model of large-scale bedforms in estuaries to explain their formation and dynamic equilibrium configuration as a function of environmental parameters. Progress Development of process-based idealized model which aims to show the effect of the gravitational circulation on estuarine bedforms. 2020 Publications van der Sande, W. M., Roos, P. C., & Hulscher, S. J. (2020). Modeling the influence of the gravitational circulation on estuarine sand dunes. In EGUGeneral Assembly 2020: Sharing Geoscience Online.

UT – Civil Engineering – Water Engineering and Management

Project leaders
Prof. dr. K.M. Wijnberg, Prof. dr. D. Van der Wal, Dr. ir. E.M. Horstman

Research theme
Complex dynamics of fluids

Participants
Ir. R. Gijsman

Cooperation
NIWA New Zealand
NUS Singapore

Funded by
University of Twente

Funded %
University 100%

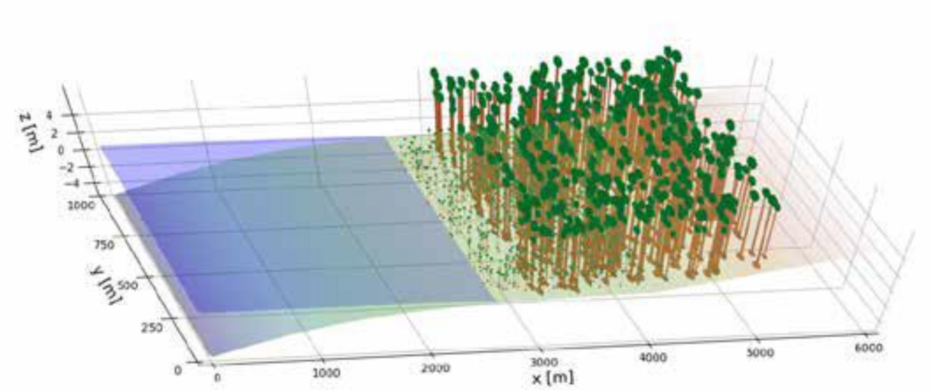
Start of the project
2019

Information
Ir. R. Gijsman
r.gijsman@utwente.nl

Project title
Biophysical responses of mangrove forests to variations in hydrodynamic forcing

Project aim
We aim to measure, analyze and model sedimentation/erosion and tree growth/decline in mangrove forest fringes in response to yearly variations in hydrodynamic forcing, to assess mangrove forests as an adaptive coastal protection measure. Field measurements will be conducted in contrasting mangrove forest fringes in New Zealand (collaboration with NIWA) and Singapore (collaboration with NUS). These measurements span timescales of up to a year to capture variations in hydrodynamic forcing.

Progress
The literature review has been extended to provide an overview of existing knowledge on the value of mangroves for coastal protection, as well as the present monitoring techniques, modelling approaches and guidelines that are available. The installed measurement frames in a mangrove forest fringe in the Firth of Thames estuary in New Zealand and Sungei Buloh Wetland Reserve in Singapore have been collecting hydrodynamic and morphological data. These measurements will be used to calibrate and validate a developed numerical process-based biophysical model (see the figure). Initially, short-term morphodynamics and their effect on the survival of young mangrove seedlings are investigated. Afterwards, the model timescales are extended to study long-term development of mangroves and their value for coastal protection. These long-term developments will be validated with a remote-sensing analysis using satellite observations and areal imagery of the Firth of Thames estuary.



UT – Civil Engineering – Water Engineering and Management

Project leaders
prof. dr. S.J.M.H. Hulscher
dr. J.J. van der Werf

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
ir. J.W.M. Kranenborg

Cooperation
Delft University of Technology
Deltares
Arcadis
Royal HaskoningDHV
Svašek Hydraulics
Rijkswaterstaat
Hoogheemraadschap Hollands
-Noorderkwartier, Marin

Funded by
NWO-TTW, Deltares

Funded %
NWO 90 %
Industry 10%

Start of the project
2018

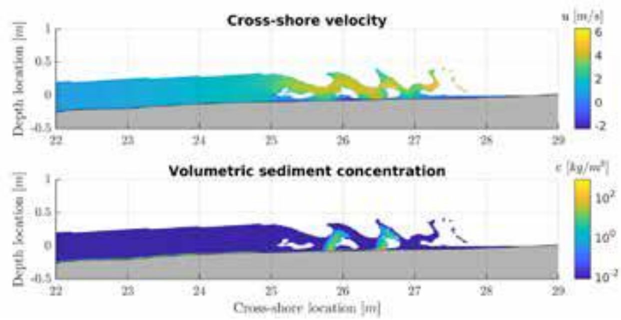
Information
Joost Kranenborg
j.w.m.kranenborg@utwente.nl

Project title
Shaping the beach: Numerical modeling of swash zone processes

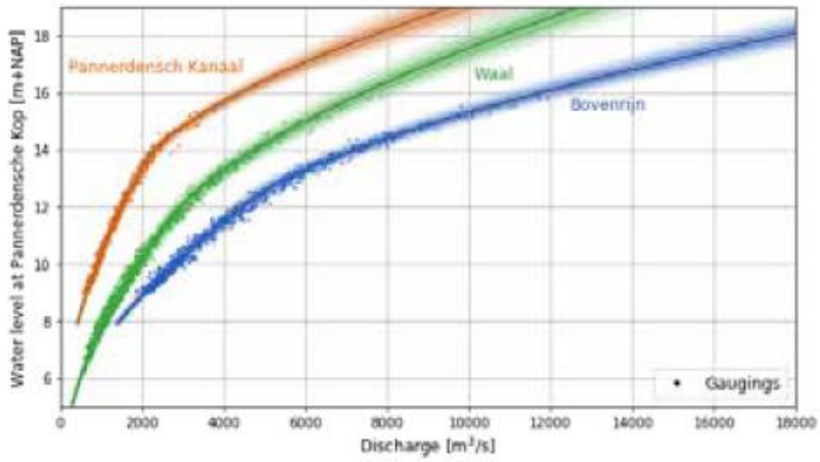
Project aim
Numerical models are an important tool to study the nearshore environment. However, these tools still have difficulty predicting the dynamics on the boundary between the dry part of the beach and the surf zone, called the swash zone. In this project, a new type of model will be applied to study swash zone processes that affect sediment transport and morphodynamics. These new insights in swash processes will be used to build novel parametrizations that can be used by coastal engineers to better predict the evolution of the coastal environment.

Progress
During the year the focus has been on developing and improving the depth-resolving model for swash hydrodynamics and sediment transport. Special focus has been on stabilizing numerical aspects and improving the morphodynamic performance of the model. Some initial model results can be seen in the figure. Apart from the model, an analysis of an experimental campaign concerning pore pressures in the swash has started.

2020 Publications
1. Kranenborg, J. W. M., Pauli, T. , Dionisio Antonio, S. , Campmans, G. H. P., Jacobsen, N. G. , van der Werf, J. J., Reniers, A. , & Hulscher, S. J. M. H. (2021). Large-scale laboratory measurements of the pore pressure response to bichromatic waves in the swash zone. 31. Abstract from NCK Days 2021, Online Event, Netherlands. https://www.nck-web.org/content/documents/2021/Book_of_Abstracts_2021.pdf
2. Kranenborg, J. , Campmans, G., Jacobsen, N. G. , Werf, J. V. D., McCall, R., Reniers, A. , & Hulscher, S. (2020). Rans modelling of cross-shore sediment transport and morphodynamics in the Swash-zone. In P. Lynett (Ed.), Proceedings of virtual Conference on Coastal Engineering, 2020 (International Conference Proceedings of Coastal Engineering; Vol. 36). https://doi.org/10.9753/icce.v36v.sediment.14
3. Kranenborg, J. W. M. , Campmans, G. H. P., Jacobsen, N. G. , van der Werf, J.J., Reniers, A. J. H. M. , & Hulscher, S. J. M. H. (2020). A depth-resolving model for intraswash hydrodynamics and sediment transport. 55-55. Abstract from NCK Days 2020, Texel, Netherlands.



Cross-shore velocity and sediment concentrations in the swash. The sediment plumes generated by the vortices from the broken bore are clearly visible.

UT – Civil Engineering – Water Engineering and Management	
Project leaders prof. dr. S.J.M.H. Hulscher dr. J.J. Warmink Research theme Mathematical and computational methods for fluid flow analysis Participants ir. M.R.A. Gensen Cooperation Rijkswaterstaat, STOWA, Deltares, HKV Lijn in Water, Natuurmonumenten, Hollands Noorderkwartier, Noorderzijlvest, It Fryske Gea, Vechtstromen Funded by NWO-TTW Funded % NWO 100% Start of the project 2017 Information Matthijs Gensen m.r.a.gensen@utwente.nl	Project title AllRisk B3: Large-scale uncertainties in river water levels Project aim The goal of this project is to quantify and possibly reduce river water level uncertainties in a bifurcating river system. In this very complex and interactive system uncertainties are propagated throughout the entire system. Dominant sources of water level uncertainties are regulation structures at bifurcation points, river bed forms and large-scale river interventions. In this project these uncertainty sources are studied in a model with dimensions similar to the river Rhine. This work will give insight into the combined effect of natural processes and human river interventions to improve river maintenance strategies. Progress Using a 31 year long record of discharge estimates based on ADCP measurements, rating curves were constructed for the Rhine bifurcations. Standard rating curves do not show a closing water balance, with more discharge in the main river than in the distributaries. By incorporating water balance closure in the construction of rating curves, more accurate rating curves were obtained. Using the rating curves, we estimated water level and discharge uncertainty for the Rhine distributaries. 2020 Publications 1. Gensen, M.R.A., Warmink J.J., & Hulscher, S.J.M.H., 2020. Water level uncertainties due to uncertain bedform dynamics in the Dutch Rhine system. In Kalinowska, M.B., Mrokowska, M.M., Rowínski, P.M. (Eds.): Recent trends in Environmental Hydraulics: 38th International School of Hydraulics. 2. Gensen, M.R.A., Warmink, J.J., Huthoff, F., & Hulscher, S.J.M.H., 2020. Uncertain amount of impact of a river intervention in a bifurcating river system. 80-81. Abstract from NCR Days 2020, Nijmegen, Netherlands. 3. Gensen, M. R. A., Warmink, J. J., Hulscher, S. J. M. H., & Huthoff, F. (2020). Effects of a river intervention on water levels in a bifurcating river system. In W. Uijttewaal, M. J. Franca, D. Valero, V. Chavarrias, C. Ylla Arbos, R. Schielen, & A. Crosato (Eds.), River Flow 2020: Proceedings of the 10th Conference on Fluvial Hydraulics (Delft, Netherlands, 7-10 July 2020) (pp. 1890-1898). London: CRC Press/Balkema. 4. Gensen, M. R. A., Warmink, J. J., Huthoff, F., & Hulscher, S. J. M. H. (2020). Feedback Mechanism in Bifurcating River Systems: the Effect on Water-Level Sensitivity. Water, 12(7), [1915]. https://doi.org/10.3390/w12071915. 

UT – Civil Engineering – Water Engineering and Management

Project leaders
prof. dr. S.J.M.H. Hulscher
prof. dr. K.W. Wijnberg
dr. G.H.P Campmans

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
P. Pourteimouri, MSc

Cooperation
NWO
Delft University of Technology

Funded by
NWO and co-funded by RWS and HHNK

Funded %
University 15 %
NWO 85 %

Start of the project
2018

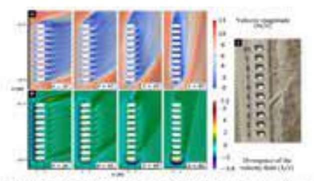
Information
Paran Pourteimouri
06 85180426
p.pourteimouri@utwente.nl

Project title
The impact of buildings at beaches on airflow and aeolian sediment transport patterns across beach-dune topography, using CFD modeling

Project aim
Coastal zones have always attracted a large number of people because of the resources and the recreations that they provide. This highly increasing population living along the coastline, increases the demands for construction of restaurants, beach houses and pavilions at the beach-dune interface. These structures block the wind flow and change the airflow patterns which in turn alter the sediment transport pathways and influence the aeolian sand dunes.

Progress
During the last year, the PhD candidate, P. Pourteimouri, developed her aerodynamic model using OpenFOAM which is an open source CFD solver. She modelled the airflow patterns around an isolated building and multiple buildings. The numerical model predictions of the flow velocity were validated with wind-tunnel measurements provided by a meteorological institute at Hamburg University. The comparisons show reasonable agreements between the numerical model results and the experimental data. Next, the PhD candidate studied the impacts of building dimensions, specifically building length, width and height, as well as wind direction on airflow patterns.

2020 Publications
1. Pourteimouri, P., Campmans, G. H. P., Wijnberg, K. M., & Hulscher, S. J. M. H. (2020). Modelling airflow patterns around beach houses using computational fluid dynamics. 47-47. Abstract from NCK Days 2020, Texel, Netherlands.
2. Pourteimouri, P. (2020). The impact of buildings at beaches on airflow and aeolian sediment transport patterns across beach-dune topography, using CFD modelling: Literature report. (CE&M Research report; No. 2020R-002/WEM-002). Enschede: University of Twente.
3. Pourteimouri, P., Campmans, G., Wijnberg, K., & Hulscher, S. (2020). The impact of buildings' characteristics on airflow patterns and bed morphology at beaches, using CFD modelling. Coastal Engineering Proceedings, (36v), 4-4.
6. Wijnberg K. M., Poppema D., Mulder J., Bergen van J., Campmans G., Galifornia-Silva F., Hulscher S., Pourtemouri P. (2020). Beach-dune modelling in support of Building with Nature for an integrated spatial design of urbanized sandy shores. Research in Urbanism Series (RiUS) 7.



UT – Civil Engineering – Water Engineering and Management

<u>Project leaders</u>
Prof. Dr. S.J.M.H. Hulscher, Dr. J.J. Warmink, Prof. Dr. M.R.A. van Gent

<u>Research theme</u>
Complex dynamics of fluids

<u>Participants</u>
Weiqiu Chen

<u>Cooperation</u>
Deltares, Rijkswaterstaat, Ministerie van Infrastructuur en Water, HKV, Royal Haskoning DHV, Arcadis, Witteveen + Bos, HillBlocks, Vechtstromen

<u>Funded by</u>
China Scholarship Council

<u>Funded %</u>
Scholarships 100 %

<u>Start of the project</u>
2017

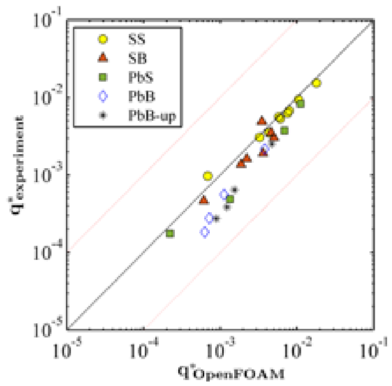
<u>Information</u>
Weiqiu Chen
06 16510831

<u>Project title</u>
The influence of berms, roughness and oblique waves on wave overtopping processes at dikes

<u>Project aim</u>
Due to climate change and sea level rise, the possibility of wave overtopping at dikes increases significantly and large overtopping discharge might cause dike breaching. Thus, a reliable estimation of average overtopping is essential for dike design and safety assessment. Berms and roughness elements are widely used to decrease overtopping discharge at dikes. Besides, the direction of incident waves is often oblique relative to dikes. Therefore, this Project aims to improve estimates of berm, roughness and oblique wave influence on wave overtopping thereby improving predictive accuracy of average overtopping discharge at dikes.

<u>Progress</u>
A 2DV numerical model was built based on the framework of OpenFOAM to predict the overtopping discharge at dikes. The numerical model is capable of predicting the average overtopping discharge accurately.
The 2DV numerical was also validated for overtopping flow velocity and layer thickness at the waterside edge of dike crest.
A 3D OpenFOAM model was developed to include the oblique waves. Model results verified the dependency of the influence of oblique waves on
berm width.

<u>2020 Publications</u>
1. Chen, W., van Gent, M. R. A., Warmink, J. J., & Hulscher, S. J. M. H. (2020). The influence of a berm and roughness on the wave overtopping at dikes. Coastal engineering, 156, [103613]. https://doi.org/10.1016/j.coastaleng.2019.103613.
2. Chen, W., Marconi, A., van Gent, M. R. A., Warmink, J. J., & Hulscher, S. J. M. H. (2020). Experimental Study on the Influence of Berms and Roughness on Wave Overtopping at Rock-Armoured Dikes. Journal of marine science and engineering, 8(6), 1-21. [446]. https://doi.org/10.3390/jmse8060446.
3. Chen, W., Warmink, J., van Gent, M., & Hulscher, S. (2020). Modelling of wave overtopping at dikes using openfoam. In Coastal Engineering 2020 (36v ed.). (International Conference Proceedings of Coastal Engineering; Vol. 36). https://doi.org/10.9753/icce.v36v.papers.27.



Deposition patterns around two scale models of buildings

UT – Civil Engineering – Water Engineering and Management

Project leaders
Prof. dr. K.M. Wijnberg
Prof. dr. S.J.M.H. Hulscher

Research theme
Complex dynamics of fluids

Participants
Ir. D.W. Poppema

Cooperation
University of Delft

Funded by
NWO, Hoogheemraadschap Hollands Noorderkwartier, Rijkswaterstaat, Deltares, Witeveen&Bos, H+N+S

Funded %
NWO 80%
Industry 20%

Start of the project
2017

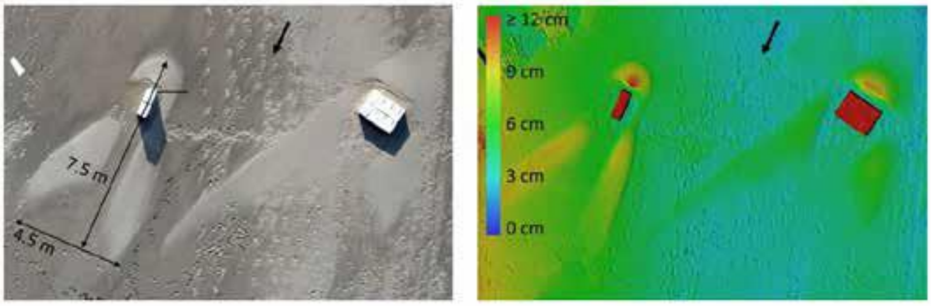
Information
Daan Poppema
d.w.poppema@utwente.nl

Project title
Experiment-supported modelling of beach-dune evolution in interaction with the built environment

Project aim
Dunes protect the Netherlands and many other places against flooding. Simultaneously, the beach-dune area is very attractive for recreation, resulting in the development of buildings like beach pavilions and holiday homes. These buildings influence Aeolian sediment transport patterns and thereby shape dune development. Therefore, this research aims to understand and model how (configurations of) buildings at the beach-dune interface affect Aeolian sediment flows and long-term dune development. It is part of the ShoreScape project.

Progress
Based on earlier field experiments, we determined how the size and shape of buildings at the beach affect the size of patterns of sand deposition around these buildings, described this in a paper for Coastal Engineering and presented this work virtually on the vICCE conference. In addition, we analysed the effect of building orientation and the distance between buildings on deposition patterns, based on field experiments performed earlier and a new full-size experiment performed on the beach together with TU Delft. With these results, we have answered part of the research questions of the thesis.

2020 Publications
1. Poppema, D.W. (2020). The effect of buildings on the morphological development of the beach-dune system: Literature report. (CE&M Research report; No. 2020R-001/WEM-001). Enschede: University of Twente.
2. Poppema, D. , Wijnberg, K., Mulder, J. , & Hulscher, S. (2020). The effect of building geometry on aeolian deposition and erosion patterns: a field experiment. In P. Lynett (Ed.), Proceedings of virtual Conference on Coastal Engineering, 2020 (International Conference Proceedings of Coastal Engineering; Vol. 36). https://doi.org/10.9753/icce.v36v.sediment.21
3. Poppema, D. W. , Wijnberg, K. M., Mulder, J. , & Hulscher, S. J. M. H. (2020). The effect of building geometry on aeolian deposition patterns: a field experiment. 21-21. Abstract from NCK Days 2020, Texel, Netherlands.



UT – System Analysis and Computational Science – Mathematics of Computational Science

Prof.dr.ir. JWW van der Vegt

The research in the Mathematics of Computational Science group in the Department of Applied Mathematics of the University of Twente concentrates on two main topics:

1. The development, analysis and application of numerical algorithms for the (adaptive) solution of partial differential equations for problems originating from the physical and technical sciences, in particular (discontinuous Galerkin) finite element methods.
2. Mathematical modeling of complex physical problems to make them accessible for computation, in particular for turbulence and geophysical problems. In order to support these activities a significant research effort is directed towards the development of hpGEM, an object oriented toolkit for finite element methods, written in C++, and suitable for high performance parallel computers. Important applications are in the fields of gas dynamics, wet chemical etching of microstructures, fluid structure interaction, two phase flows both dispersed and with free surfaces, water waves, large eddy simulation of turbulent flows, geophysical flows and computational electromagnetics. Many of these projects are conducted in close collaboration with groups in physics and chemical technology, large technological research institutes (NLR, MARIN, WL Delft Hydraulics, KNMI), and industry (DSM, AKZO and Shell).

The research is conducted in the research institute IMPACT and the research in two-phase flows is part of the UT spearhead program “Dispersed multiphase flows”. The NACM group participates in the 3TU Centre of Excellence for Multiscale Phenomena.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation

Prof.dr.ir. BJ Geurts

The research of this group focusses on computational modeling of multiscale problems in multiphase flows, environmental flows and flows in complex domains. Applications are selected from the field of energy and biofluid mechanics. Novel algorithms, their parallel implementation and analysis are at the core of the research, with an emphasis on error quantification, immersed boundary methods and time-parallel integration.

UT – System Analysis and Computational Science - Multiscale Modeling and Simulation

Project leaders
Prof. B.J. Geurts

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
D. Alblas
J. Mikhal

Cooperation
Dr. J. Boogaarts – Radboud Medical Center

Funded by
UT, Radboud

Funding %
University 100%

Start of the project
2016

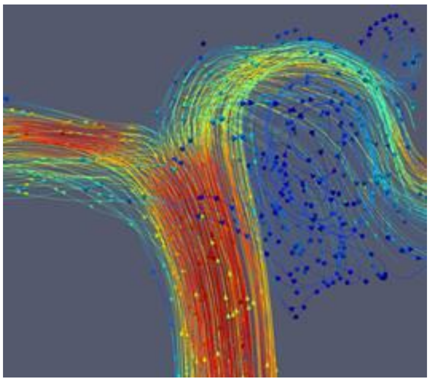
Information
Bernard Geurts
b.j.geurts@utwente.nl

Project title
Precision stenting for cerebral aneurysms and stenosis

Project aim
The objective of this research is to develop computational models for the prediction of pulsatile flow in diseased cerebral vessels displaying an aneurysm or in the carotid artery, showing stenosis. Starting from medical imagery of an individual patient, methods are developed to automatically construct the flow domain corresponding to the diseased region. This implies translation of a surface representation (STL format) to a volume representation with which the incompressible Navier-Stokes equations can be treated using an immersed boundary method. The simulation platform is implemented in OpenFOAM to allow easy transfer of methods to other groups in the field. Next to pulsatile flow analysis, the inclusion of flow-diverting stents is studied with the aim to predict location, size and shape of aneurysm that would be best suited for the patient. Moreover, the dynamics of thrombus formation and multiscale shape remodeling is included to simulate the long-term healing process.

Progress
The flow inside a side-wall aneurysm and a bifurcation aneurysm was simulated successfully. The required spatial resolution for accurate prediction of the flow was determined and a systematic upper- and lower bounding solution was obtained from geometry variations, showing the sensitivity of the solution to input uncertainties. For the first time a full flow-diverting stent was simulated at appropriate resolution – the requirements are considerably stricter as the very fine structure of the stent needs to be resolved in order to capture the flow in its smallest detail.

2020 Publications
D. Alblas, Computational fluid dynamics of stented aneurysms, Thesis MSc Applied Mathematics, University of Twente, 2020.



Flow in a bifurcation aneurysm showing extended residence time because of the diseased vessel geometry.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation	
Project leaders Prof. B.J. Geurts, Prof. D.D. Holm Research theme Mathematical and computational methods for fluid flow analysis Participants E. Luesink Cooperation Prof. C. Cotter (ICL) Funded by IC London, EPSRC Funding % NWO 100 % Start of the project 2017 Information Erwin Luesink e.luesink16@imperial.ac.uk	Project title Stochastic rotating geophysical models Project aim Flow in the upper ocean and the lower atmosphere requires solving the equations of fluid dynamics. However, our incomplete understanding of turbulence, the chaotic nature of these equations as well as the changing climate are several factors that make solving these equations incredibly difficult. By means of introducing stochastic transport noise in the shallow water equations under rotation we arrive at qualitative models with which we may study the structures in the flow and mathematical properties of geophysical fluid dynamics. This is aimed at contributing to improved weather and ocean mixing forecasting and provide an estimate of the uncertainty in the forecasts. Progress A detailed investigation of stochastic dynamics in the Lorenz 63 framework was undertaken. Both analytical and numerical methods were developed to understand the role of explicit stochasticity on the structure of the solutions. The effects of adding stochasticity on the well-known dynamical attractor of Lorenz 63 were considered and the changes in the Lyapunov exponent were quantified. A Lie-Poisson solver was developed enabling the conservation of energy and Casimirs of the dynamical system – an extension to stochastic dynamics is included to assess the reliability of the modeled predictions.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation

Project leaders
Prof. B.J. Geurts, Prof. D.D. Holm

Researchtheme
Mathematical and computational methods for fluid flow analysis

Participants
S. Ephraty
A. Franken

Funded by
NWO TOP 1 Grant

Funding %
NWO 100 %

Start of the project
2017

Information
Erwin Luesink
e.luesink16@imperial.ac.uk

Project title
Stochastic structure-preserving models for geophysical fluid dynamics

Project aim
Geophysical fluid dynamics is at the heart of studies into weather predictions, upper ocean mixing and long-term climate change. We aim to develop structure-preserving stochastic models with which first principles of the flow are retained while allowing for significant coarsening of the computational models. Through data-assimilation further connections to observed mixing behaviour can be established.

Progress
A solver for the rotating shallow water equations was developed and the wave interactions associated with flow over a 2D ridge were simulated. A detailed comparison of predictions obtained with a Discontinuous Galerkin (DG) method was prepared to quantify the resolution requirements for accurate direct numerical simulation. The flow structures were analyzed using empirical eigenfunctions.

Numerical solution of the KdV equation is pursued with different numerical methods, addressing challenges due to high order dispersion.

The use of Empirical Orthogonal Functions (EOFs) for the effective modeling of dynamical behaviour at multiple time and length scales was studied and applied to stochastic differential equations. This established the relevance of including temporal correlation next to coloured noise as stochastic model.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation	
Project leaders Prof. B.J. Geurts Research theme Mathematical and computational methods for fluid flow analysis Participants Dr. James-Michael Leahy Funded by NWO Top 1 Grant Funding % NWO 90% Industry 10% Start of the project 2020 Information b.j.geurts@utwente.nl	Project title Fundamental properties of stochastic dynamics for ocean modeling Project aim Coarsening flow models for upper ocean circulation will require both spatial and temporal regularization as well as the inclusion of structure-preserving stochastic forcing. This is aimed at capturing the main aspects of turbulent motion while reducing computational costs such that accurate simulations are possible on planetary scales, and for extended periods of time. Progress Work on Rotating Rayleigh-Benard convection has started with the development of a suitable 2D formulation in which backscatter is investigated next to stochastic forcing based on empirical eigenfunctions. The Lorenz 96 model was studied to explicitly account for closure of the slow dynamics in terms of machine learned subgrid modeling. The machine learning approach was extended to Rayleigh-Benard convection.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation	
Project leaders Prof. B.J. Geurts Research theme Mathematical and computational methods for fluid flow analysis Participants P. Cifani A. Franken Cooperation Imperial College London Funded by NWO Top 1 Grant Funding % NWO 90% Industry 10% Start of the project 2019 Information b.j.geurts@utwente.nl	Project title Stochastic structure-preserving simulation methods for turbulent flow Project aim The simulation of high-Reynolds number turbulence as occurs in geophysical flow problems requires computational models that consistently reflect the underlying structure of the equations. We develop new structure-preserving high-performance computational models for ocean-atmosphere models and incorporate stochastic dynamics. Progress A new solver for 2D Rotating Rayleigh-Benard convection was developed and tested. First steps toward PDF modeling of the coarsened dynamics were made. Lie-Poisson and geometric integration were developed for the Euler equations and the sine-Euler equations – the relevance of preserving energy and Casimirs was investigated in the context of PDEs. DNS of Rayleigh-Benard convection in 2d was conducted to create a point of reference for 'ultimate turbulence' investigations using machine learning.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation	
Project leaders Prof. B.J. Geurts, Dr. P. Cifani Research theme Mathematical and computational methods for fluid flow analysis Participants S. Ephrati, A. Franken, J. Froehlich (TU Dresden) Funded by University of Twente Funding % NWO 90% Industry 10% Start of the project 2018 Information Bernard Geurts b.j.geurts@utwente.nl	Project title High-performance simulation methods for turbulence modulation in bubbly flows Project aim The objective of this research is to create simulation methods for direct numerical simulation of turbulent bubbly multiphase flow. Resolving all scales of motion of bubbles embedded in a liquid requires precise reconstruction and tracking of the bubble interfaces and the interaction with the surrounding liquid. This creates a basis for understanding the modulation of turbulence in multiphase flows, in particular drag reduction near solid walls. Progress A Volume-of-Fluid approach was adopted and a simulation method with excellent parallel performance was developed in cylindrical coordinates. Many deformable bubbles can be successfully simulated in a turbulent Taylor-Couette flow and drag reduction can be investigated. The method was also adopted to the fundamental study of a single bubble interacting with a solid wall and in collisions with other bubbles. The interaction of bubbles with walls and rotation was investigated numerically, using DNS as main simulation approach. These results were compared to experimental findings.

UT – System Analysis and Computational Science – Multiscale Modeling and Simulation	
Project leaders Prof. B.J. Geurts, Dr. A. Lyulin Research theme Mathematical and computational methods for fluid flow analysis Participants K. Redosado Leon M. Boomstra Funded by NWO – TTW - OTP Funding % NWO 90% Industry 10% Start of the project 2018 Information Bernard Geurts b.j.geurts@utwente.nl	Project title Development of composite phase change materials for heat storage Project aim The objective of this research is to create simulation methods to understand the heat transport in Paraffin-Graphene composite materials. The topology of the fast conducting Graphene network and its connection to the bulk phase changing Paraffin material are investigated in order to arrive at a compact, high-performance material for heat storage and recovering, employing latent heat associated with the phase change of Paraffin. Progress Spherical compound systems are described using the diffusion equation and the Stefan problem – this first geometry allows analytical solutions of the temperature response, serving also as a point of reference for the development of numerical simulation methods that explicitly account for the jumps in the temperature gradient at the various interfaces that form in the system.

university of groningen

Faculty of Science and Engineering

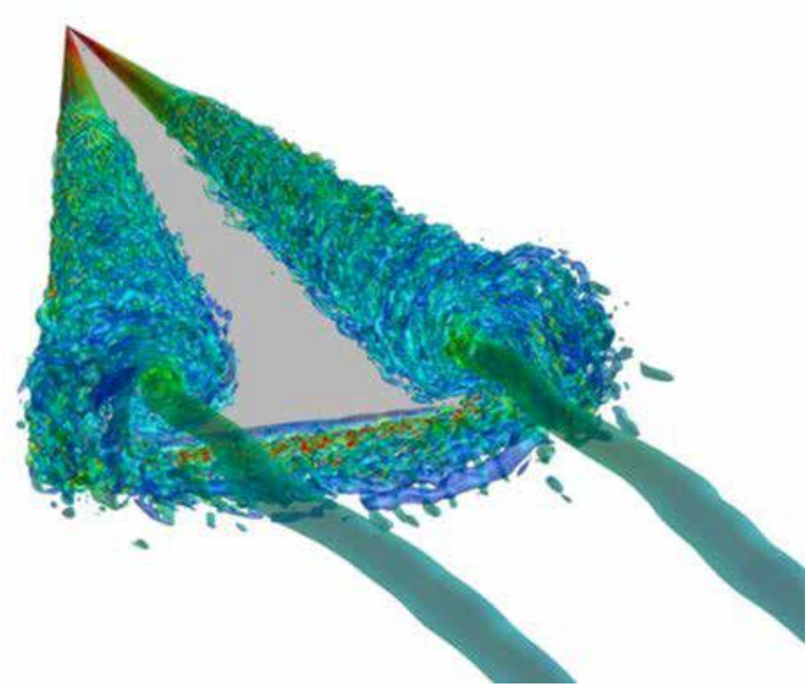
Computational and Numerical Mathematics

RUG – Science and Engineering - Computational and Numerical Mathematics

Prof.dr.ir. RWCP Verstappen
Prof.dr.ir. AEP Veldman

The group Computational Mechanics and Numerical Mathematics at the University of Groningen focuses on the development of numerical solution methods for partial differential equations in general, and for (aero and hydrodynamic) flow simulation in particular (CFD). Keywords for our algorithmic developments are symmetry-preserving discretization, Cartesian cut-cell approach, sharp-interface methods, efficient sparse-matrix solvers and large-scale continuation methods. It is our strategy to combine all algorithmic innovations from the individual research projects into one coherent CFD concept, such that all projects can profit from each other.

Application areas are direct and large-eddy simulation of turbulent flow, free-surface flow in aerospace (sloshing onboard spacecraft) and maritime engineering (hydrodynamic wave loading), oceanography (stability of the global ocean circulation), bio-medical fluid dynamics (hemodynamics) and heat transport (Rayleigh-Bénard flow). We plan to extend our research efforts towards multi-physics: fluid-structure interaction, two-phase flow, atmospheric flow and turbulent combustion. In the process of knowledge transfer, the in-house developed computer codes ComFlo and MRILU play an important role.



DNS of flow past a delta wing at Re=200,000.

RUG – Science and Engineering - Computational and Numerical Mathematics

Project leaders
A.E.P. Veldman

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
H Seubers, AEP Veldman (RUG), M Hosseini, X Chang, PR Wellens, RHM Huijsmans (TUD), J Helder, P van der Plas (MARIN), B Iwanowski (FORCE), M Borsboom (Deltares)

Cooperation
TU Delft, MARIN, FORCE Technology (Norway). Deltares, GustoMSC, Damen Shipyards, DNV-GL (Norway), Hyundai Heavy Industries (Korea)

Funded by
NWO

Funding %
NWO 50%
Industry 30%
Scholarships 20%

Start of the project
2014

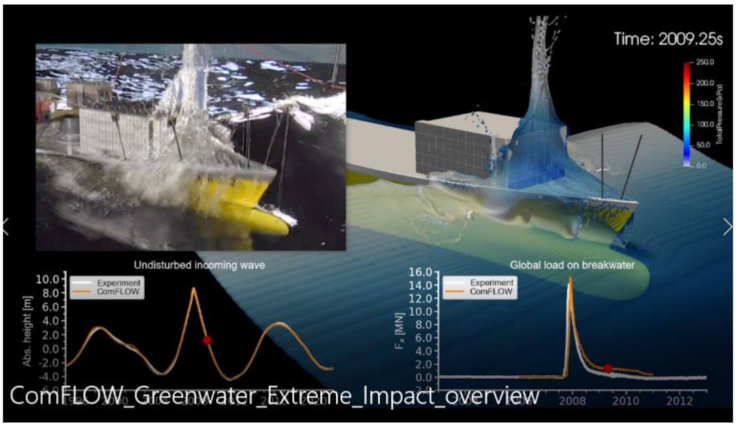
Information
AEP Veldman
a.e.p.veldman@rug.nl

Project title
ComMotion: computational methods for moving and deforming objects in extreme waves

Project aim
In the ComFLOW project, together with the maritime industry, concerns the design of numerical simulation methods for extreme waves and their impact on floating and moored constructions like offshore platforms and coastal protection systems. The most recent development phase concerns the interaction of extreme waves and floating and/or deforming bodies.

Progress
In the ComMotion project, extensions of the ComFLOW simulation method have been designed featuring moving and deforming objects. Special attention was paid to a quasi-simultaneous solid-fluid coupling method, and to absorbing boundary conditions in the presence of current. The free-surface description has been further refined, and implemented in ComFLOW release 4.2.3. The industry-funded User Group, coordinated by MARIN, has been formally installed. Applications are e.g. free-fall life boats, green-water events and floating fish farms.

2020 Publications
Arthur E.P. Veldman. Supra-conservative finite-volume methods for the simulation of subsonic flow. In: F. Chinesta, R. Abgrall, O. Allix and M. Kaliske (eds.) Proc. 14th World Congress on Computational Mechanics (WCCM) and ECCOMAS Congress 2020 (12 pages).



Comparison between experiment (MARIN) and numerical simulation (ComFLOW) of green water over the deck of a container ship.

RUG – Science and Engineering - Computational and Numerical Mathematics

<u>Project leaders</u> RWCP Verstappen <u>Research theme</u> Mathematical and computational methods for fluid flow analysis <u>Participants</u> MH Silvis, LB Streher, J Parekh, J. Sun, J. Merten, P. Cifani, J. Hidding, RA Remmerswaal, AEP Veldman, FX Trias (UPC), A Oliva (UPC), HJ Bae (Caltech) <u>Cooperation</u> Polytechnica University of Catalunya (UPC), eScience Center <u>Funded by</u> NWO, RUG <u>Funding % per money stream</u> University 20% NWO 60% Scholarships 20% <u>Start of the project</u> 1998 <u>Information</u> RWCP Verstappen r.w.c.p.verstappen@rug.nl	<u>Project title</u> Direct and Large-Eddy Simulation of Turbulence <u>Project aim</u> Our project concentrates on improving numerical techniques for direct numerical simulation (DNS) of turbulence, as well as on large-eddy simulation (LES). Finding a closure model represents the main difficulty to LES. Because turbulence is so far from being completely understood, there is a wide range of models, mostly based on heuristic arguments. The aim of the present project is to construct a class of LES-models that preserves (symmetry) properties of the Navier-Stokes equations, and ensures that the nonlinear dynamics is truncated properly, meaning that the formation of fine details is counterbalanced by the model. <u>Progress</u> In collaboration with UPC Barcelona, a mathematical methodology to combine eddy-viscosity models with the scale-similarity models in a consistent manner has been developed further. A two-layer approach was successfully tested for wall-bounded turbulent flows. An uncertainty quantification method has been implemented in OpenFOAM to approximate the uncertainty in large-eddy simulations resulting from uncertainties in the subgrid model. Low-dissipative simulation methods and models have been applied to subsonic turbulent flows. <u>2020 Dissertations</u> M.H. Silvis, Physics-based turbulence models for large-eddy simulation, Promotor: R.W.C.P. Verstappen, co-promotor: A.E.P. Veldman University of Groningen, October 9th, 2020. <u>2020 Publications</u> J. Parekh, R. Verstappen, Intrusive polynomial chaos for CFD using Open FOAM, Proceedings Int. Conf. on Computational Science, 677-691 (2020). W. Rozema, J.C. Kok, A.E.P. Veldman, R.W.C.P. Verstappen, Numerical simulation with low artificial dissipation of transitional flow over a delta wing, J. Comput. Phys. 405, 109182 (2020). W. Rozema, R.W.C.P. Verstappen, A.E.P. Veldman, J.C. Kok, Low-dissipation simulation methods and models for turbulent subsonic flow, Archives Comput. Meth. Engng. 27 (1) 299-330 (2020). R. Verstappen, Finite-volume filtering in large-eddy simulations using a minimum dissipation model. In Turbulence and Interactions, M. Deville et al (eds) Springer, 91-100 (2020).

RUG – Science and Engineering - Computational and Numerical Mathematics	
Project leaders Cristóbal Bertoglio Research theme Mathematical and computational methods for fluid flow analysis Participants Cristián Cárcamo Jeremías Garay Reidmen Aróstica Jorge Aguayo Giorgia Pase Cooperation University of Saint-Etienne, France Center for Biomedical Imaging, Chile Center of Mathematical Modeling, Chile Universidad de Concepción, Chile Funded by University of Groningen Chilean Research Agency (Conicyt) European Research Council Funding % University 25% EU 50% Scholarships 25% Start of the project 2016 Information Cristóbal Bertoglio c.a.bertoglio@rug.nl	Project title Inverse problems in blood flows from magnetic resonance imaging Project aim We develop improved means for quantifying flow features focusing on the cardiovascular system through parametrizing the models from magnetic resonance data. In particular, we aim that the methods are robust imaging artifacts (e.g. noise, aliasing, undersampling) Progress We made good progress on the mathematical analysis of different inverse problems from 4D Flow data: • We proved a priori error estimates for the pressure estimators from 3D velocity measurements with respect to their spatial resolution • We developed a new mathematical method to quantify *a posteriori* the quality of 4D Flow MRI • We performed existence analysis of minimizers for the problem of estimating the shape of obstacles from distributed velocity measurements 2020 Dissertations Inverse problems in elastography and displacement-flow MRI**.** Hugo Carrillo. *PhD thesis, Universidad de Chile & University of Groningen, 2020*.

RUG – Science and Engineering - Computational and Numerical Mathematics

Project leaders
F.W. Wubs

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
RA Remmerswaal, AEP Veldman (RUG), M Kaminski (TUD), H Bogaert (MARIN), L Brosset (GTT) + many more

Cooperation
MARIN, TU Delft, TU Eindhoven, Univ. Twente, CWI, GTT (France) + many more

Funded by
STW

Funded %
NWO 80%
Industry 20%

Start of the project
2016

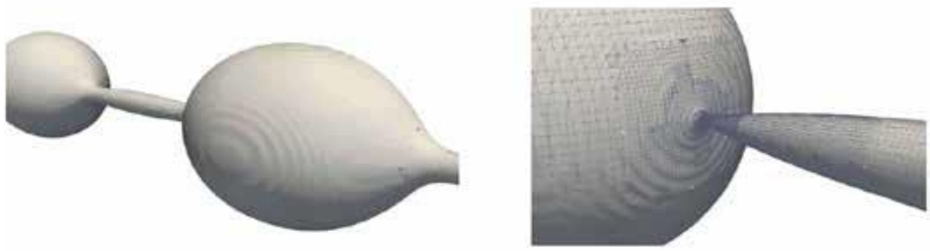
Information
AEP Veldman
a.e.p.veldman@rug.nl

Project title
SLING : Liquid sloshing in LNG tanks - Variability

Project aim
The SLING project studies liquid sloshing in cargo tanks for liquid natural gas (LNG). Of particular interest is the flow just before and during impact with the tank wall. In the RUG contribution, the variability of the free surface due to flow instabilities has to be modelled. The CFD approach is based on the ComFLOW simulation method developed in a parallel project.

Progress
The RUG contribution to the project focusses on the simulation of flow instabilities, like Kelvin-Helmholtz, Rayleigh-Taylor and Rayleigh-Plateau. Capillary forces and turbulence play a role. These delicate physical processes require high numerical accuracy. As the viscous shear layer between the two phases is hard to resolve, the velocity field and the pressure gradient are allowed to be discontinuous. This results in a sharp description of the interface. Its curvature is calculated from a parabolic reconstruction of the interface, which converges upon grid refinement.

2020 Publications
Ronald A. Remmerswaal and Arthur E.P. Veldman. Numerical modeling of twophase flow with contact discontinuities. In: F. Chinesta, R. Abgrall, O. Allix and M. Kaliske (eds.) Proc. 14th World Congress on Computational Mechanics (WCCM) and ECCOMAS Congress 2020 (12 pages).



Droplet pinch-off (Rayleigh-Plateau) from a long liquid filament. The right-hand picture shows the local grid refinement applied to resolve the flow details.

REPORTS OF INDIVIDUAL RESEARCH GROUPS

WAGENINGEN UNIVERSITY & RESEARCH

Department of Animal Sciences

Experimental Zoology

Department of Agrotechnology and Food Sciences

Food Process Engineering

Physical Chemistry and Soft Matter

WUR - Animal Sciences - Experimental Zoology

Prof.dr.ir. JL van Leeuwen

The mission of the Experimental Zoology Group is to unravel the relationships between form and function in zoological systems in a developmental and evolutionary context and to provide bioinspired solutions for technological and health problems. The current main research area of the Experimental Zoology Group is the biomechanics of motion systems in vertebrates and insects, with three research lines that profit from one another: (1) Biomechanics of animal flight, including the biofluid dynamics of avian and insect flight and in-flight host detection of malaria mosquitoes. (2) Biomechanics of fish swimming, including swimming and developmental mechanics in larval fish, fin propulsion, visuo-motor-system development and effects of a live-bearing reproductive strategy on swimming performance. This research line also includes developmental mechanics of bones and muscles linking bone remodelling to molecular regulation. (3) Bioinspired design solutions for human health, including development of steerable needles (inspired by the mechanics of the ovipositor in parasitic wasps), and construction of gentle grippers for delicate human tissues (inspired by wet adhesion of toe-pads in tree frogs). The Experimental Zoology Group participates also within the graduate school Wageningen Institute of Animal Sciences (WIAS).

WUR - Animal Sciences - Experimental Zoology

Project leaders
Dr. Ir. FT Muijres

Research theme
Complex dynamics of fluids

Participants
Prof. JL van Leeuwen

Cooperation
Prof. MH Dickinson (Caltech, USA)
Dr. GCHE de Croon (TU Delft)
Dr. BM Tomotani (NIOO)

Funded by
NWO ALW Veni
Wageningen University (WUR)

Funded %
University 50%
NWO 50%

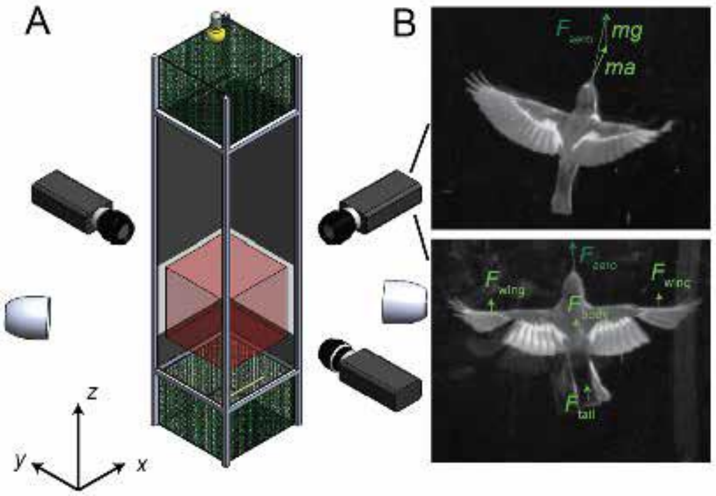
Start of the project
2012

Information
FT Muijres
florian.muijres@wur.nl

Project title
Aerodynamics of flight manoeuvres in natural flyers

Project aim
Flying animals such as birds, bats and insects are extremely maneuverable. Although the fluid dynamics of steady flight in animals is quite well understood, the dynamics that underlies manoeuvres was not yet well known. We study the aerodynamics and control dynamics of flying animals, with the aim to understand how animals use their flapping wings to manipulate aerodynamic forces and torques, in order to rapidly and precisely control flight stability and manoeuvrability.

Progress
We studied how birds escape by flying upwards [1,2]. We tested two conditions, birds with molt gaps and birds with added weights. To compensate for wing molt gaps, an upward escaping songbird actively reducing the molt gap by moving the adjacent feathers closer to each other, and by increasing the angle-of-attack of the wings during the downstroke movement [1]. Using a functional model, we quantified how the reduction in escape flight performance scales with added weight [2].



Experimental Zoology WUR (A) The flight arena for studying escape flights in birds, including a three-camera high-speed videography system. (B) video stills of Pied Flycatchers without and with wing molt gaps (top and bottom images, respectively), including the aerodynamic forces modelled (from [1]).

WUR - Animal Sciences - Experimental Zoology

Project leaders
JL van Leeuwen

Research theme
Complex dynamics of fluids

Participants
CJ Voesenek
MJM Lankheet (co-supervisor)
FT Muijres (co-supervisor)

Cooperation
G Li (Chiba University, Tokyo)
H Liu (Chiba University, Tokyo)
UK Müller (Univ. of Fresno)
GJ van Heijst (TUE)

Funded by
NWO/ALW

Funded %
University 20%
NWO 80%

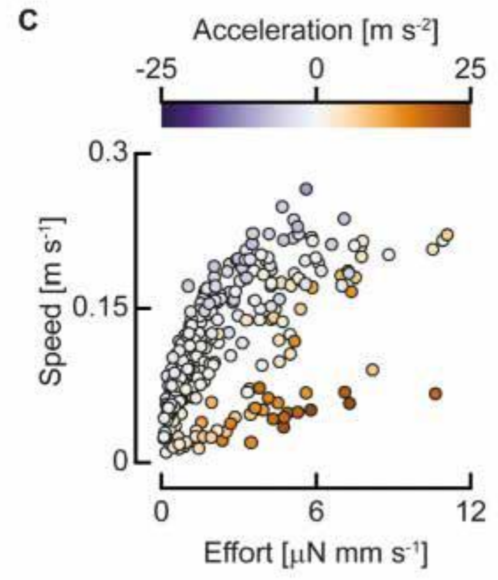
Start of the project
2012

Information
JL van Leeuwen
johan.vanleeuwen@wur.nl

Project title
Fluid-muscle interaction of free-swimming zebra fish larvae

Project aim
Zebrafish larvae start swimming within two days post fertilization (2 dpf) and develop rapidly over the next few days. We study how these developmental changes affect locomotory performance. To achieve this, we will create a numerical model of the larvae that accounts for the mechanics of the muscular system, the external fluid mechanics, and their mutual interactions. This approach allows us to unravel how muscle activation patterns lead to swimming motions and identify causes of changes in swimming performance across development.

Progress
We published a paper on the 3D fast-start performance of larval fish. To avoid predation, fish larvae need to generate high-enough escape speeds in a wide range of possible directions, in a short-enough time. We filmed the fish motions using five synchronised high-speed video cameras. Using an inverse dynamics analysis, we showed that larval fish can independently control their escape speed and escape direction. Furthermore, we prepared a paper to PLoS Biology about the internal bending moment distributions of swimming zebrafish larvae. This paper describes a novel methodology to compute bending moments for recorded arbitrary large amplitude 3D motions. The main biological conclusion of this paper is that fish larvae use a strikingly simple net actuation to control their swimming motions, which results in complex fluid-structure interactions.



Mean swimming speed of larval fish against effort (defined as the peak bending moment along the fish divided by the time of the half beat). Colour indicates the mean acceleration of the half tail beat. High speeds and high acceleration require effort. From: https://journals.plos.org/plosbiology/article?id=10.1371/journal.pbio.3000462

WUR - Animal Sciences - Experimental Zoology

Project leaders
FT Muijres

Research theme
Complex dynamics of fluids

Participants
A Cribellier (PhD candidate)
JL van Leeuwen (promotor)
MJM Lankheet (advisor)
A Hiscox (advisor)
J Spitzen (advisor)
W Takken (advisor)
RPM Pieters (technician)

Cooperation
J Casas (external advisor)
M Geier (external partner)
MH Dickinson (external advisor)
R Dudley (external advisor)

Funded by
Graduate School Wageningen
Institute of Animal Sciences (WIAS)

Funded %
University 100 %

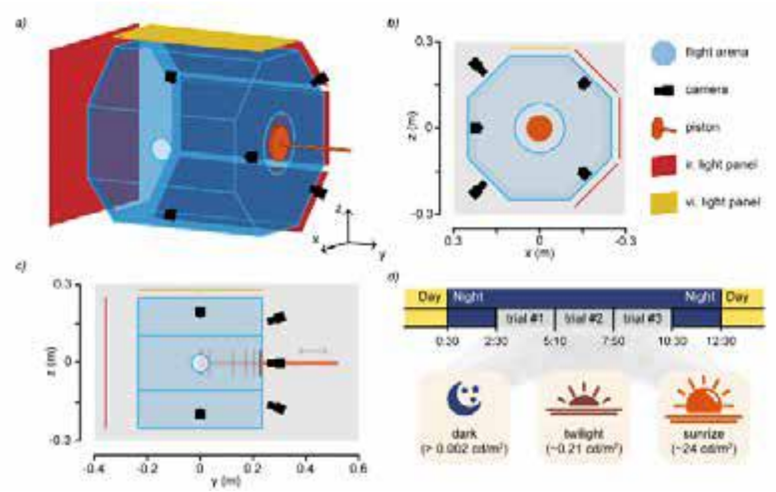
Start of the project
2016

Information
FT Muijres
florian.muijres@wur.nl

Project title
Biomechanics of the capture and escape of malaria mosquitoes

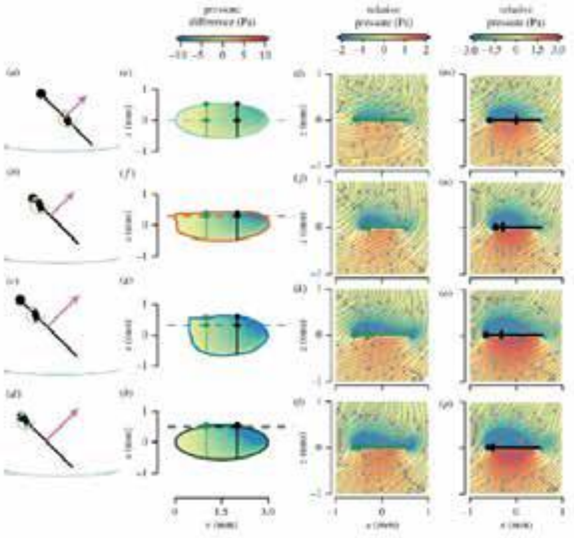
Project aim
We study the flight dynamics of the malaria mosquito (Anopheles coluzzii) with the applied goal to improve current and/or develop novel mosquito trap systems for vector control. We aim to reach this goal by quantifying mosquito flight dynamics during host searching with diverse attractive cues and mosquito escape manoeuvres when swatted. These should provide novel insights into mosquito flight behaviour and the mechanisms involved for odour and wind gust detection and response.

Progress
The building of our experimental setup (see the figure) was completed and experiments have been carried out successfully, and CFD simulations of the piston-induced airflow have been made. High-speed video recordings of mosquito escapes are currently being analyzed with the goal of estimating the kinematics of body and wing movements. All of these will be combined to better understand how mosquitoes detect a looming threat (the piston) and how good their escape performance in variable light conditions.



Flight arena with mechanical swatter and real-time mosquito tracking system. The tracking system consists of 5 Basler infrared-enhanced cameras at 90 fps. The piston (in orange) was triggered when a mosquito was predicted to enter a 5 cm radius sphere in the centre. (a) three-dimensional view, (b) front view and (c) side view of the flight arena. (d) Experimental protocol. Every day, the piston disk type was changed and 50 female mosquitoes were released into the arena. Then, during the night, three experimental trials were conducted with different light conditions (dark, twilight or sunrise).

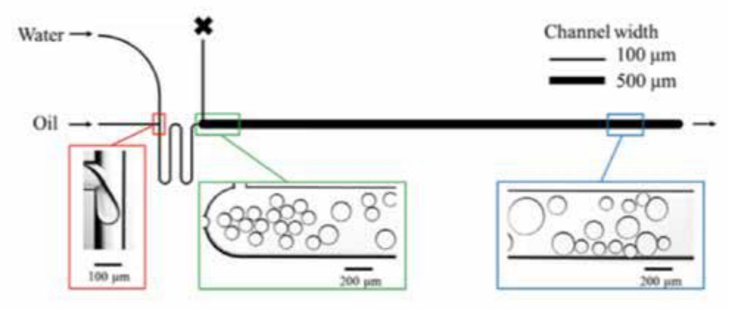
WUR - Animal Sciences - Experimental Zoology	
Project leaders FT Muijres Research theme Complex dynamics of fluids Participants P Goyal (PhD candidate) JL van Leeuwen (promotor) MJM Lankheet (advisor) RPM Pieters (technician) Cooperation G H C E de Croon (TU Delft) B W van Oudheusden (TU Delft) MH Dickinson (Caltech) Funded by NWO Funded % NWO 100 % Start of the project 2017 Information FT Muijres florian.muijres@wur.nl	Project title To be as nimble as a bee: A bio-inspired sensory-motor system for gust control of Micro Air Vehicles Project aim State-of-the-art drones have difficulty flying outdoors or in natural environments due to the presence of wind gusts. But, the insects navigate through complex environments in the presence of wind gusts very easily. The aim of this project is to study and understand the gust rejection capabilities of insects. Specifically, we aim to understand how bumblebees fly and land in various wind conditions including no wind condition. Progress We worked on the dataset obtained from the first set of experiments (different light conditions and landing patterns) and obtained the following major result: Bumblebees land rapidly by stepwise adjusting their setpoint of relative optical expansion rate Using a novel analysis method, we identify a previously unknown deceleration strategy that bumblebees exhibit during a landing approach. Specifically, we show that they decrease their velocity towards the landing platforms by holding the relative-rate-of-expansion constant within a wide range of set-points and collectively exhibit an increase in these set-points as distance from the platforms reduces. This increase in set-points with distance results in a discrete approximation of deceleration strategy of birds that use constant time-to-contact-rate to regulate their expansion rate with distance, thereby results in relatively fast landings. This robust landing strategy occurs in a wide range of luminance and with variable degrees of expansion cues, suggesting that bumblebees adequately controls landing by using neural summation.

WUR - Animal Sciences - Experimental Zoology	
Project leaders Dr. Ir. FT Muijres Research theme Complex dynamics of fluids Participants W.G. van Veen (PhD candidate) JL van Leeuwen (promotor) Cooperation MH Dickinson (Caltech, USA) Funded by NWO ALW Veni Wageningen University (WUR) Funded % University 50% NWO 50% Start of the project 2015 Information FT Muijres florian.muijres@wur.nl	Project title Aerodynamics of the high-frequency flapping wings in flying mosquitoes Project aim Mosquitos fly with exceptionally high wingbeat frequencies, which sets them apart from other flying insects. The goal of this project is to understand the aerodynamics behind high frequency flapping mosquito flight. Progress At the stroke reversal between upstroke and downstroke, insects wings rapidly pitch up, sometimes at more than 4000 rad per second. The unsteady aerodynamics that results from these fast wing pitch movements were not well understood, and thus we performed a computational fluid dynamics study on rapidly pitching insect wings. Between simulations, we systematically varied the wingbeat kinematics, including speed of the wingbeat and pitching motion, as well as the wing morphology. Based on the results, we developed an aerodynamic force model of pitching insect wings. Using this model, we showed that the so-called pitch-rate related aerodynamic forces are particularly high in the high-frequency wingbeat of mosquitoes.  *The aerodynamics of four rapidly upward pitching wings, including a mosquito wing (red outline) and fruit fly wing (blue outline). (a–d) Schematic of the aerofoil, where the dot indicates the leading edge and the diamond the pitch axis. Pink arrow illustrates the resultant aerodynamic force. (e–h) The distribution of pressure differences across the wing surface. (i-p) Pressure and flow field around the wings at two spanwise locations indicated in (e-h) by green lollipops for (i-l) and black lollipops for (m–p), respectively.*

WUR - Agrotechnology and Food Sciences - Food Process Engineering

Prof.dr.ir. CGPH Schroën

Microtechnology, membranes and modelling: M3 Membranes (and other microstructures) can be used for various purposes, such as separation, which is the traditional application for membranes, but also for formation of emulsions, foams, and sprays. Within the food microtechnology group, all these aspects are investigated (together with technical assistants Jos Sewalt and Maurice Strubel), and modelling is used as a tool to gain fundamental insight in the underlying mechanisms. For specific information on projects, please consult the pages indicated below, and the PhD thesis section which holds completed projects.

WUR - Agrotechnology and Food Sciences - Food Process Engineering	
Project leaders K Schroën, Bijoy Bera (post-doc) Research theme Complex dynamics of fluids Participants WU, TUD, UvA, Shell, Unilever Cooperation WU, TUD, UvA, Shell, Unilever Funded by ISPT/NWO Funded % NWO 100% Start of the project 2018 Information Karin Schroën karin.schroen@wur.nl	Project title Emulsion coalescence under extreme conditions Project aim Within the multi-phase flows program, emulsion formation and stability is investigated amongst others under extreme conditions, in order to thus generate data that are relevant for the industrial partners of the project. Within the project in Wageningen, microfluidic devices were developed that can be operated at elevated temperatures. Progress These microfluidic devices that are illustrated in the image below, were successfully used to generate data on emulsion droplet coalescence under a wide range of conditions, including variation of the amount of surfactant used. 

WUR - Agrotechnology and Food Sciences - Physical Chemistry and Soft Matter

Prof.dr.ir. J van de Gucht

Physical Chemistry and Soft Matter at Wageningen University & Research is interested in phenomena at the nanoscale, where creative chemistry is essential, complex physics is a rule rather than an exception and biology comes to life.

WUR - Agrotechnology and Food Sciences - Physical Chemistry and Soft Matter

Project leaders
Joris Sprakel

Research theme
Complex dynamics of fluids

Participants
Riccardo Antonelli,
Thomas Kodger

Cooperation
Canon production printing

Funded by
NWO

Funding %
NWO 100 %

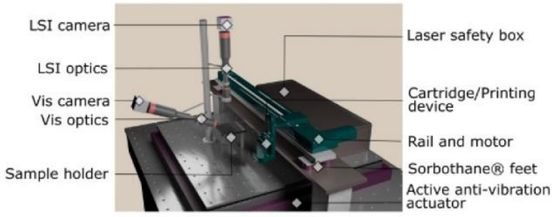
Start of the project
2018

Information
E-mail: office.pcc@wur.nl

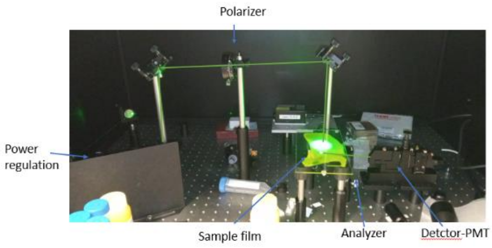
Project title
In Watching Ink Dry: unravelling drying and film formation in inkjet printing

Project aim
In this project, we will study inkjet printing with high spatiotemporal resolution, using a unique combination of a new imaging method, Laser Speckle Imaging, and a custom-made printing instrument. This will enable us to study in detail the drying, porous wicking, coalescence and film formation within picolitre droplets with micrometre and sub-millisecond resolution. In this approach, we aim to deepen our understanding of inkjet printing and study the effects of surface porosity, ink formulation and environmental conditions on the deposition process and the properties of the final coating.

Progress
For the development of this project two different setups have been built and are now operative. The main one (figure A) can print picolitre drops on demand and study their evolution using LSI technique. The second setup (figure B), more recent, can provide information on film formation dynamic. Next to the setup the project also aims for the production of a simplified ink version, as a reference to be compared with the commercial inks provided from the supplier.



A)



B)

WUR - Agrotechnology and Food Sciences - Physical Chemistry and Soft Matter

Project leaders
Prof. J. van der Gucht

Research theme
Complex dynamics of fluids

Cooperation
Unilever
Shell
Evodos

Funded by
NWO-ISPT

Funding %
NWO 100%

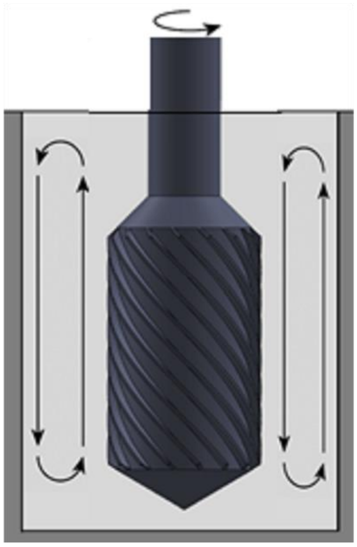
Start of the project
2020

Information
Akankshya Majhi
akankshya.majhi@wur.nl

Project title
Controlling multiphase flow: Local versus global rheology of dense emulsions

Project aim
The aim of this project is to better understand the relation between macroscopic behaviour of (complex) emulsions and phenomena at microscopic scale such as droplet wall interactions, droplet dynamics, interfacial properties and inter-droplet forces. This is achieved by combining bulk rheological measurements, microscopic non-invasive imaging techniques, microfluidic tools and microscopic force measurements. This will result in better models to describe (complex) emulsion behaviour.

Progress
Using simple 3D printed helical patterns on the wall, we investigated the wall-fluid interactions with an aim to control the bulk fluid flow behaviour. Flow field imaging using rheoMRI technique gave insights about the coupling of flows in two orthogonal directions. Currently, we are working on combining rheology measurements with Laser Speckle Imaging to measure wall stresses and to directly probe the heterogeneous (wall) flow properties of dense emulsions.



Angular wall patterns induce secondary flow

WUR - Agrotechnology and Food Sciences - Physical Chemistry and Soft Matter

Project leaders
Prof. J. van der Gucht

Research theme
Complex dynamics of fluids

Cooperation
Unilever
Evodos
Shell

Funded by
NWO-ISPT

Funding %
NWO 100%

Start of the project
2018

Information
Jesse Buijs
Jesse.buijs@wur.nl

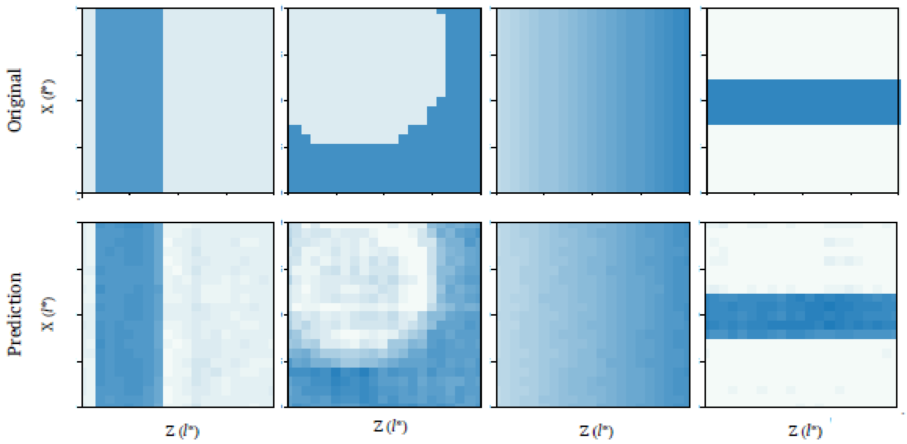
Project title
Controlling multiphase flow

Project aim
The aim of this project is to expand the non-invasive imaging tool Laser Speckle Imaging (LSI) from 2D to 3D. LSI quantifies dynamics such as flow or diffusion in non-transparent materials and is thus perfectly suited to study the behaviour of droplets in concentrated emulsions. By expanding the technique to yield 3D-information we can get new insights into this behaviour under changing conditions.

Progress
While the machine to perform 3D-LSI is still under construction, we started developing data analysis algorithms in parallel. First we simulate photons travelling through computer generated objects with certain flow patterns to obtain simulated experimental data. Then we use machine learning to predict the flow pattern from the simulated experimental data. In the image there are some example predictions which show that accurate depth information can be obtained from the experimental data and thus 3D-LSI is viable.

2020 Publications

Baij, L., Buijs, J., Hermans, J. J., Raven, L., Iedema, P. D., Keune, K., & Sprakel, J. (2020). Quantifying solvent action in oil paint using portable laser speckle imaging. *Scientific reports*, *10*(1), 1-12.



REPORTS OF INDIVIDUAL RESEARCH GROUPS

Institute for Meteorology and Oceanography (IMAU)

UU - Institute for Marine and Atmospheric Research Utrecht (IMAU)

Prof.dr. LRM Maas
Prof.dr.ir. HA Dijkstra

The research of this Institute concerns transport processes in the ocean and in the atmosphere. Some sections of IMAU participate in the research school Buys Ballot. However, the section Dynamical Oceanography also participates in the JM Burgerscentrum.

One of the main topics is the role of the ocean circulation in the variability of the climate system and the processes controlling the large-scale ocean circulation. In particular, attention is focused on the path changes of the oceanic western boundary currents (such as the Gulf Stream in the Atlantic Ocean, the Kuroshio in the Pacific Ocean, and the Agulhas Current near South Africa), the El Nino / Southern Oscillation phenomenon in the Pacific, and the North Atlantic Multidecadal variability. Both theory development and (high resolution) model simulation are used to better understand these phenomena and our favourite framework to analyse the complex behaviour of ocean flows is that provided by stochastic dynamical systems theory.

Another line of research focuses on ocean wave dynamics. Due to temperature and salinity differences, the ocean is stratified in density. This supports internal gravity waves, that have their maxima below the surface of the ocean. These waves are especially generated by surface tidal motions over topographic irregularities, like the continental shelf edge or Mid-Atlantic Ridge. When subject to friction at ocean boundaries, Reynolds-stresses induced by internal gravity waves are found to also drive horizontal mean flows. Another type of large-scale ocean wave is related to the rotation of the earth: the Rossby wave. This wave type exists due to variations in background potential vorticity, which represents the ratio of planetary vorticity (equal to twice the rotation rate of the fluid) and water depth. An explicit expression for the Green’s function, describing the response to an impulsive point source is uncovered.

UU - Institute for Marine and Atmospheric Research Utrecht (IMAU)

Project leaders
HA Dijkstra

Research theme
Complex dynamics of fluids

Participants
D. Castellana MSc

Cooperation
CRITICS ITN partners

Funded by
EU-H2020

Funded %
EU 100%

Start of the project
2016

Information
H.A. Dijkstra
h.a.dijkstra@uu.nl

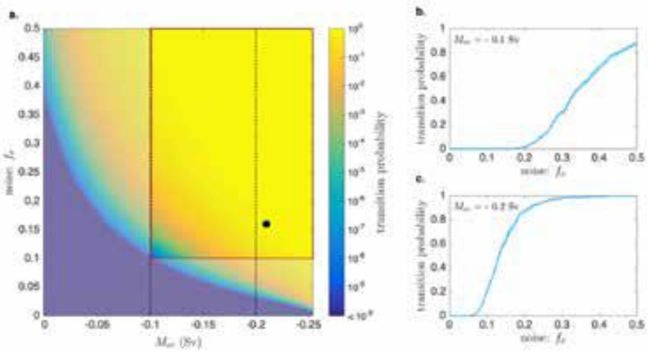
Project title
Critical Transitions in Complex Systems

Project aim
The Atlantic Ocean circulation, in particular its Meridional Overturning Circulation (MOC), is sensitive to freshwater anomalies. A tipping point may exist such that the present-day MOC will collapse if the northern North Atlantic freshwater forcing is gradually increased. In addition, if the MOC is in a multiple equilibrium regime it may undergo transitions due to the impact of noise. The aim of this project is to determine the probability of transitions of the MOC in a hierarchy of stochastic ocean-climate models. Both noise in the surface forcing as well as in the representation of turbulent mixing processes will be considered.

Progress
Using an idealized ocean model, we determined estimates of the transition probability of noise-induced transitions of the AMOC, within a certain time period, using a methodology from large deviation theory. We find that there are two types of transitions, with a partial or full collapse of the AMOC, having different transition probabilities. For the present-day climate state, we estimate the transition probability of the partial collapse over the next 100 years to be about 15%, with a high sensitivity of this probability to the surface freshwater noise amplitude. In a follow-up study, we determined the transition probabilities of the AMOC in state-of-the-art (CMIP5) climate models. The PhD project has finished in 2020 and Daniele Castellana defended his thesis on October 12, 2020.

Dissertations
Noise induced phenomena in the large-scale ocean circulation, PhD thesis Daniele Castellana, PhD defense 12-10-2020.

2020 Publications
Castellana, D. and H. A. Dijkstra, Noise-induced Transitions of the Atlantic Meridional Overturning Circulation in CMIP5 models, Nature Scientific Reports, 10: 20040 (2020).



(a) Transition probabilities of partial AMOC transitions (cessation of the downwelling) in 100 years, calculated for each couple of the parameters (Mov, fσ), representing the freshwater transport by the AMOC and the noise level of the freshwater forcing, respectively. (b-c) Transition probabilities as a function of fσ for two particular choices of Mov as indicated by the dashed lines in (a). The shaded areas represent the interquartile range for the probabilities.

UU - Institute for Marine and Atmospheric Research Utrecht (IMAU)

Project leaders
LRM Maas

Research theme
Mathematical and computational methods for fluid flow analysis

Participants
M Rovira-Navarro

Cooperation
B Vermeersen, W van der Wal, R van Ostayen (TUD) and T Gerkema (NIOZ)

Funded by
NWO

Funded %
NWO 100%

Start of the project
2018

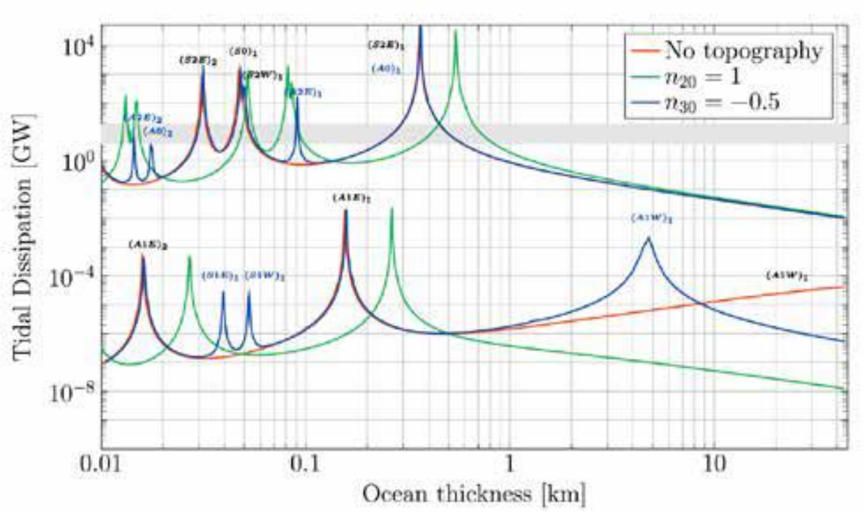
Information
L Maas
06 23540843
L.R.M.Maas@uu.nl

Project title
Internal wave attractors

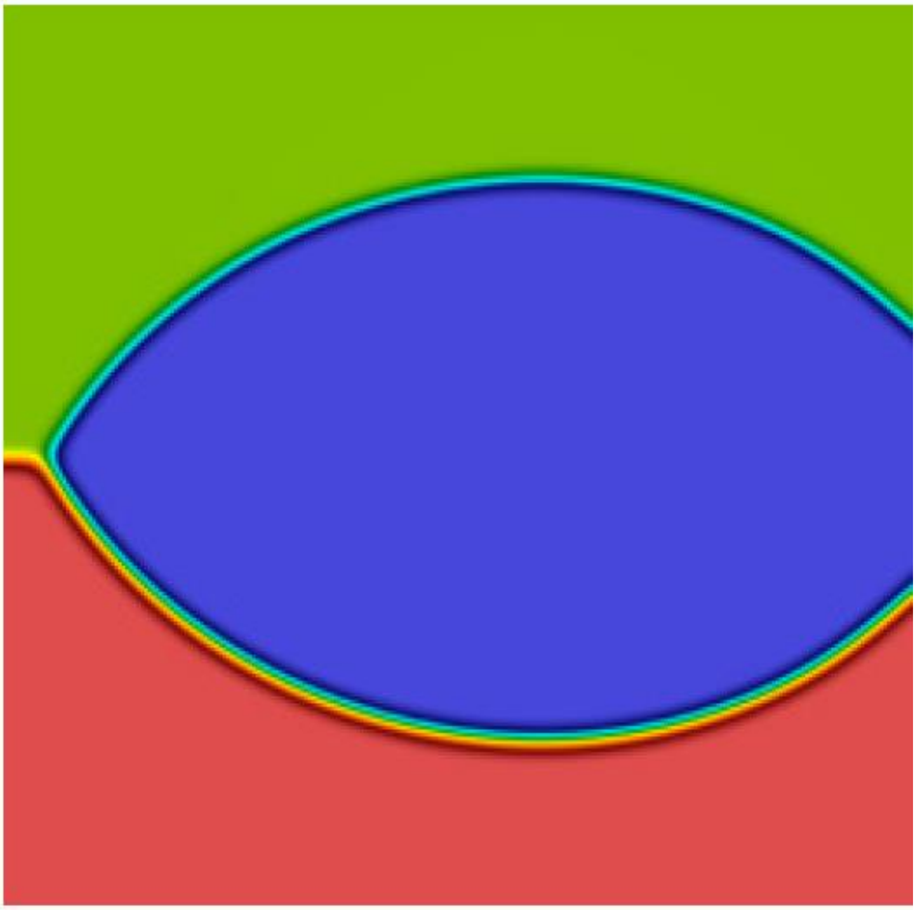
Project aim
Anisotropic media, such as stratified and/or rotating oceans, develop preferred locations of wave activity and mixing in the vicinity of so-called wave attractors that may be relevant for ocean mixing and particle transport.

Progress
We study tides in the subsurface ocean of Saturn's moon Enceladus solving the classical Laplace Tidal Equations for a homogeneous density ocean of variable thickness. The ocean response to the eccentricity and obliquity tide in the form of gravity-wave and a westward propagating Rossby-Haurwitz modes respectively is characterised by a series of resonant peaks. Here tidal dissipation surpasses Enceladus's observed heat flux but does not agree with the observed distribution of heat. Further study therefore intends to incorporate ocean stratification and the relevance of wave attractors to investigate additional heating and its spatial distribution. Tidal dissipation for different ocean thicknesses for an ocean of constant thickness and oceans with topography (n20 and n30). Each panel shows tidal dissipation due to the obliquity tide and eccentricity tide with the curves for the obliquity tide corresponding to lower values of tidal dissipation. The different panels show the solution for different ice shell thicknesses (0,1 and 10 km) and damping Rayleigh coefficients. Different resonances are labelled, those excited due to antisymmetric topography are indicated in blue. The horizontal grey band corresponds to the observed thermal output of Enceladus.

2020 Publications
Rovira-Navarro, M., Gerkema, T., Maas, L. R. M., van der Wal, W., van Ostayen, R., & Vermeersen, B. (2020). Tides in subsurface oceans with meridional varying thickness. Icarus, 343, 113711.



WHO & WHERE



Delft University of Technology (TUD)

<u>**Faculty Mechanical Maritime and Material Engineering (3mE)**</u>

Department Process & Energy

Energy Technology
Prof.dr.ir. BJ Boersma - b.j.boersma@tudelft.nl

Fluid Mechanics
Prof.dr.ir. J Westerweel - j.westerweel@tudelft.nl
Prof.dr. DJEM Roekaerts - d.j.e.m.roekaerts@tudelft.nl
Prof.dr.ir. RAWM Henkes *(part-time)* - r.a.w.m.henkes@tudelft.nl
Prof.dr.ir. G Ooms *(em.)* - prof.ooms@gmail.com

Multiphase Systems
Prof.dr.ir. C Poelma - c.poelma@tudelft.nl

Complex Fluid Processing
Prof.dr.ir. JT Padding - j.t.padding@tudelft.nl

Department Maritime and Transport Technology

Prof.dr.ir. C van Rhee - c.vanrhee@tudelft.nl
Prof.dr.ir. TJC van Terwisga *(part-time)* - t.v.terwisga@tudelft.nl

<u>**Faculty of Applied Sciences**</u>

Department Chemical Engineering

Transport Phenomena
Prof.dr.ir. CR Kleijn - c.r.kleijn@tudelft.nl
Prof. dr. S. Kenjeres, Dipl.-Ing - S.Kenjeres@tudelft.nl
Prof.dr.ir. AB de Haan - a.b.dehaan-1@tudelft.nl
Prof.dr.ir. HEA van den Akker *(part-time)* - h.e.a.vandenakker@tudelft.nl
Prof.dr.ir. K Hanjalic (*em.*) - k.hanjalic@tudelft.nl

Product and Process Engineering
Prof.dr.ir. JR van Ommen - j.r.vanommen@tudelft.nl

<u>**Faculty of Electrical Engineering, Mathematics and Computer Science (EEMCS)**</u>

Delft Institute of Applied Mathematics (DIAM)

Numerical Analysis
Prof.dr.ir. C Vuik - c.vuik@tudelft.nl
Prof.dr.ir. CW Oosterlee *(part-time)* - c.w.oosterlee@tudelft.nl

Mathematical Physics
Prof.dr.ir. AW Heemink - a.w.heemink@tudelft.nl
Prof.dr.ir. M Verlaan *(part-time)* - m.verlaan@tudelft.nl

<u>**Faculty of Aerospace Engineering**</u>

Department Aerodynamics, Wind Energy, Flight Performance and Propulsion

Aerodynamics
Prof.dr. F Scarano - f.scarano@tudelft.nl
Prof.dr. S Hickel - s.hickel@tudelft.nl

Wind Energy (Aeroacoustics)
Prof.dr. D Casalino - d.casalino@tudelft.nl

Faculty of Civil Engineering and Geosciences

Department Hydraulic Engineering

Environmental Fluid Mechanics
Prof.dr.ir. WSJ Uijttewaal - w.s.j.uijttewaal@tudelft.nl
Prof.dr. J Pietrzak - j.d.pietrzak@tudelft.nl
Prof.dr.ir. AJHM Reniers - a.j.h.m.reniers@tudelft.nl

Eindhoven University of Technology (TU/e)

Department of Applied Physics

Fluids & Flows
Prof.dr. HJH Clercx - h.j.h.clercx@tue.nl
Prof.dr.ir. GJF van Heijst (*em.*) - g.j.f.v.heijst@tue.nl
Prof.dr.ir. F Toschi - f.toschi@tue.nl
Prof.dr.ir. AA Darhuber - a.a.darhuber@tue.nl

Transport in Permeable Media
Prof.dr.ir. OCG Adan *(part-time)* - o.c.g.adan@tue.nl

Elementary Processes in Gas Discharges
Prof.dr.ir. GMW Kroesen - g.m.w.kroesen@tue.nl
Prof.dr. UM Ebert *(part-time)* - ute.ebert@cwi.nl

Department of Mechanical Engineering

Energy Technology
Prof.dr.ir. EH van Brummelen - e.h.v.brummelen@tue.nl
Prof.dr.ir. DMJ Smeulders - d.m.j.smeulders@tue.nl
Prof.dr.ir. AA van Steenhoven *(em.)* - a.a.v.steenhoven@tue.nl
Prof.dr.ir. CJ van Duijn *(em.)* - c.j.v.duijn@tue.nl
Prof.dr.ir. HMA Wijshoff *(part-time)* - h.m.a.wijshoff@tue.nl
Prof.dr. HA Zondag *(part-time)*) - h.a.zondag@tue.nl

Power & Flow
Prof.dr.ir. NG Deen - n.g.deen@tue.nl
Prof.dr. LPH de Goey - l.p.h.d.goey@tue.nl
Prof.dr. JGM Kuerten - j.g.m.kuerten@tue.nl
Prof.dr.ir. AW Vreman *(part-time)* - a.w.vreman@tue.nl
Prof.dr. M Golombok *(part-time)* - m.golombok@tue.nl

Microsystems
Prof.dr.ir. JMJ den Toonder - j.m.j.d.toonder@tue.nl

Department Chemical Engineering and Chemistry

Multi-scale Modelling of Multi-phase Flows
Prof.dr.ir. JAM Kuipers - j.a.m.kuipers@tue.nl

Chemical Process Intensification
Prof.dr.ir. M van Sint Annaland - m.v.sintannaland@tue.nl

Interfaces with Mass Transfer
Prof.dr. CWM van der Geld - c.w.m.v.d.geld@tue.nl (*until April 2020*)

Department Mathematics and Computer Science

Centre for Analysis, Scientific Computing and Applications (CASA)
Prof.dr.ir. B Koren - b.koren@tue.nl
Prof.dr. MA Peletier - m.a.peletier@tue.nl
Prof.dr. JJM Slot *(part-time)* - j.j.m.slot@tue.nl

Department of the Built Environment

Buiding Physics
Prof.dr.ir. B Blocken - b.j.e.blocken@tue.nl

University of Twente (UT)

Faculty of Science and Technology

Physics of Fluids (PoF)
Prof.dr. D Lohse - d.lohse@utwente.nl
Prof.dr. D van de Meer - d.vandermeer@utwente.nl
Prof.dr. J Snoeijer - j.h.snoeijer@utwente.nl
Prof.dr. M Versluis - m.versluis@utwente.nl
Prof.dr. X Zhang - x.zhang-3@utwente.nl
Prof.dr. A Prosperetti *(part-time)* - prosperetti@jhu.edu
Prof.dr. R Verzicco *(part-time)* - r.verzicco@utwente.nl
Prof.dr.ir. L van Wijngaarden (*em.*) - l.vanwijngaarden@tnw.utwente.nl

Physics of Complex Fluids
Prof.dr. F Mugele - f.mugele@utwente.nl

Soft Matter, Fluidics and Interfaces
Prof.dr.ir. RGH Lammertink - r.g.h.lammertink@utwente.nl

Faculty of Engineering Technology

Department of Thermal and Fluid Engineering

Engineering Fluid Dynamics
Prof.dr.ir. CH Venner - c.h.venner@utwente.nl
Prof.dr.ir. HWM Hoeijmakers *(em.)* - h.w.m.hoeijmakers@utwente.nl

Thermal Engineering
Dr.ir. JBE Kok - j.b.w.kok@utwente.nl
Prof.dr.ir. G Brem - g.brem@utwente.nl
Prof.dr.ir. ThH van der Meer (*em.*) - t.h.vandermeer@utwente.nl

Multiscale Mechanics
Prof.dr. S Luding - s.luding@utwente.nl

Department of Civil Engineering

Water Engineering and Management
Prof.dr. SJMH Hulscher - s.j.m.h.hulscher@utwente.nl
Prof.dr. KM Wijnberg - k.m.wijnberg@utwente.nl

Faculty of Electrical Engineering, Mathematics and Computer Science (EEMCS)

System Analysis and Computational Science (SACS)

Mathematics of Computational Science
Prof.dr.ir. JWW van der Vegt - j.j.w.vandervegt@utwente.nl

Multiscale Modelling and Simulation
Prof.dr.ir. BJ Geurts - b.j.geurts@utwente.nl

University of Groningen

Faculty of Science and Engineering

Computational and Numerical Mathematics
Prof.dr.ir. RWCP Verstappen - r.w.c.p.verstappen@rug.nl
Prof.dr. AEP Veldman *(em.)* – a.e.p.veldman@rug.nl

Wageningen University & Research (WUR)

Department of Animal Sciences

Experimental Zoology
Prof.dr.ir. JL van Leeuwen - johan.vanleeuwen@wur.nl

Department of Agrotechnology and Food Sciences

Food Process Engineering
Prof.dr.ir. CGPH Schroën - karin.schroen@wur.nl

Physical Chemistry and Soft Matter
Prof.dr.ir. J van der Gucht - jasper.vandergucht@wur.nl

Utrecht University (UU)

Institute for Meteorology and Oceanography (IMAU)
Prof.dr. HA Dijkstra - h.a.dijkstra@uu.nl
Prof.dr. LRM Maas - l.r.m.maas@uu.nl

Board of Directors

Prof.dr.ir. CJ van Duijn (TU/e, Chairman) - c.j.v.duijn@tue.nl
Prof.dr.ir. JWM Hilgenkamp (UT) - j.w.m.hilgenkamp@utwente.nl
Prof.dr.ir. JD Jansen (TUD) - J.D.Jansen@tudelft.nl
Prof.dr.ir. LPH de Goey (TUE) - l.p.h.d.goey@tue.nl
Ir. JJ Meerman (Teijin Aramid) - hans.meerman@teijinaramid.com

Management Team

Prof.dr. JGM Kuerten (TU/e) - j.g.m.kuerten@tue.nl
Prof.dr. D Lohse (UT) - d.lohse@utwente.nl
Prof.dr.ir. J Westerweel (TUD) - j.westerweel@tudelft.nl

Industrial Advisory Board

Dr. B Borkent (NWO) - b.borkent@nwo.n
Ir. G Hommersom (Dow Benelux) - ghommersom@dow.com
Dr.ir. J Janssen (Unilever) - jo.janssen@unilever.com
Ir. J de Jong (DSM) - jelle.jong-de@dsm.com
Dr. J Kok (NLR) - johan.kok@nlr.nl
Ir. JJ Meerman (Teijin Aramid) - hans.meerman@teijinaramid.com
Dr.ir. TWJ Peeters (Tatasteel) - tim.peeters@tatasteel.com
Ir. J Pennekamp (Deltares) - johan.pennekamp@deltares.nl
Dr.ir. HJ Prins (Marin) - h.j.prins@marin.nl
Ir. H Reinten (Oce) - hans.reinten@oce.com
Ir. M Riepen (ASML) - michel.riepen@asml.nl
Ir. M Roest (VORTECH) - mark.roest@vortech.nl
Ir. P Veenstra (Shell) (Chairman) - peter.veenstra@shell.com
Ir. Veraar (TNO Defence and Safety) - ronald.veraar@tno.nl
Dr.ir. FC Visser (Flowserve) - fvisser@flowserve.com
Prof.dr.ir. B Vreman (Nouryon & TU/e) - bert.vreman@nouryon.com

JM Burgerscentrum (the Netherlands)

Prof.dr.ir. GJF van Heijst, scientific director - g.j.f.v.heijst@tue.nl (until 1 Aug. 2021)
Prof.dr.ir. RAWM Henkes, scientific director (0652096201) - r.a.w.m.henkes@tudelft.nl (from 1 Aug. 2021)
I Hoekstein-Philips, secretariat - jmburgerscentrum@tudelft.nl
Mekelweg 2, 2628 CD Delft

Burgers Program Maryland (USA)

James Duncan, Professor, Dept. of Mechanical Engineering
https://eng.umd.edu/clark/faculty/535/James-Duncan
Director, Burgers Program for Fluid Dynamics
duncan@umd.edu

JMBC Annual Report & Research Programme
2020