Master Thesis project proposals

Q1 2025-2026

Power **&** Flow

Department of Mechanical Engineering

Eindhoven University of Technology

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Preface

This is an overview of all the Master Graduation project proposals available in the Power and Flow section.

Please select 3 choices of different projects in order of preference and write a short motivation for your first choice to Giulia Finotello (G.Finotello@tue.nl). Something like:

- My first preference is project…because I am very motivated to work on…
- Second preference is…(no motivation needed)
- Third preference is.. (no motivation needed)

If you need more information on a proposal you can contact directly one of the supervisors (the emails are in each project proposal).

General Information	
Supervisor	Noud Maes
Mentor	Stan Latten
Internal/External	Internal
Exp./Num./Design	Experimental

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Experimental study on diesel / H_2 dual-fuel operation in a heavy-duty engine

Stan Latten*, Noud Maes
*E-mail: s.latten@tue.nl

INTRODUCTION

The usage of hydrogen as a fuel for Internal Combustion Engines (ICEs) can reduce or even eliminate carbon-related emissions such as CO_2 or soot. However, this fuel also presents several challenges such as NO_x emissions, pre-ignition, high auto-ignition temperature, and low fuel density.

One way of implementing H_2 in existing heavy-duty diesel engines consists of installing H_2 Port-Fuel Injectors (PFI) into the intake manifold, allowing the engine to run purely on diesel if necessary or to substitute a certain amount of diesel for H_2, in which case the diesel acts as a pilot to ignite the H_2-air mixture.

EXPERIMENTAL SETUP

The goal of this project is to run experiments on the single-cylinder MX13 engine setup, which will be modified to include a H_2-PFI system. This setup contains various pressure, temperature, and emission sensors to study the combustion and emissions, and to achieve certain engine operating conditions. These experiments will also serve as a benchmark for a dedicated optical H_2 engine, which is currently being built.

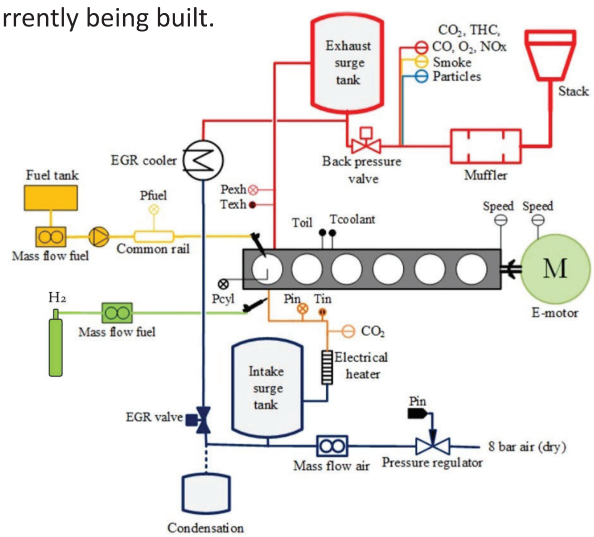


Figure 1: Schematic overview of the modified MX13 engine setup

OBJECTIVE

Exploring the possibilities and limitations of H_2-PFI systems in heavy-duty diesel engines in terms of engine-out emissions and performance.

APPROACH

- Literature study on (H_2) engines, getting familiar with the setup and analytical methods
- Involvement with preparing the engine setup
- Running benchmark tests on diesel fuel
- Progressively replacing more diesel with H_2
- Analyzing the data, writing a report and presenting the results

STUDENT PROFILE

- Affinity with combustion engines
- Knowledge on the working principles of ICEs
- MATLAB programming and post-processing skills
- Followed the course Clean engines and future fuels

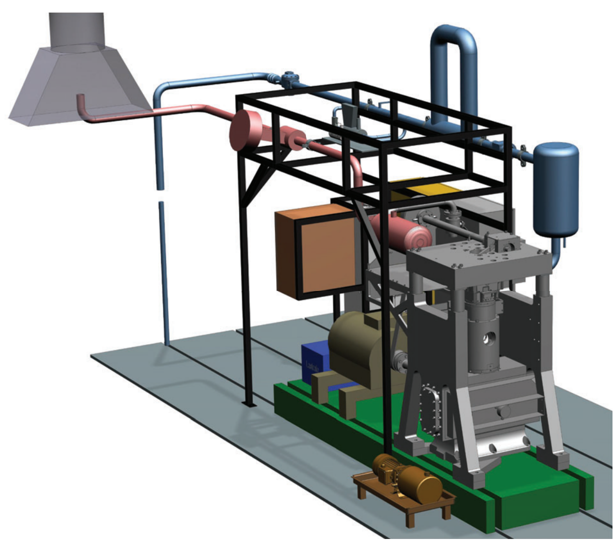


Figure 2: CAD render of the optical H_2 engine setup, currently being built in ZEL

Supervisor	Giulia Finotello
2nd supervisor	
Daily supervisor	Anke Smeets
Company	
Starting date	
Exp./Num./Design	Experimental

Power &Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Hydrogen based reduction of iron oxides in a fluidized bed reactor

Anke Smeets* and Giulia Finotello

*Email: a.h.j.smeets@tue.nl

INTRODUCTION

In the past years, iron as an energy carrier has become one of the topics which has been thoroughly investigated at the TU/e. Several projects have been on the combustion and reduction of iron powder. Team SOLID has been working with it, and several startups have emerged. The main focus has been on the combustion of the powder. However, to make the Iron power cycle a full cycle, the reduction side has to be investigated more.

Several experimental setups are available at the Power and Flow research group for the reduction of iron oxides. The performed experiments on iron oxide reduction show limitations in reduction efficiency due to the sticking behaviour of particles. Therefore, a turbulent fluidized bed set-up is built to prevent this by allowing better mixing and heat transfer of the particles. One problem observed is still partial defluidization during reduction in the bed.

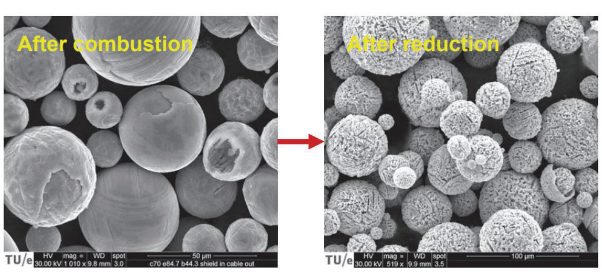


Figure 1: SEM images of combusted iron (left) and direct reduced iron (right) [1]

PROJECT

You will contribute to this field by investigating the underlying causes of the partial defluidization during reduction. Different experiments will be executed, before and after defluidization, to analyze the change in material properties. Therefore, the iron(-oxides) powder should be sampled after each experiment and analyzed on particle size, composition and morphology. Moreover, with the new gas analyzer, it is possible to investigate the reduction kinetics over time. Different flow rates, temperatures and reduction times will be investigated.

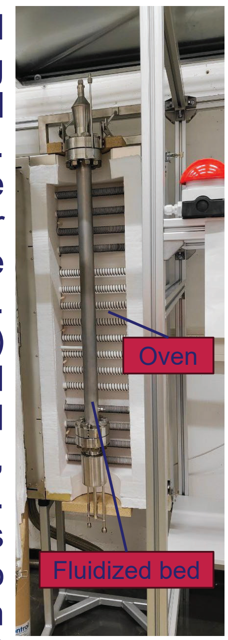


Figure 2: fluidized bed

GOALS

This project consists of two main goals:

1. Investigate the evolution of the material properties
2. Find the cause for de-fluidization during reduction.

TASKS

- Study the theory on iron oxide reduction and sticking behaviour.
- Investigate sticking behaviour for different reduction times and temperatures in the fluidized bed.
- Assess the powder in terms of size, morphology, porosity, composition and reduction rate (gas analyzer).

[1] Hessels C.J.M. et. al. 'Reduction kinetics of iron oxides fines, produced from iron combustion, using hydrogen' Powder Technology. 2021. (in process)

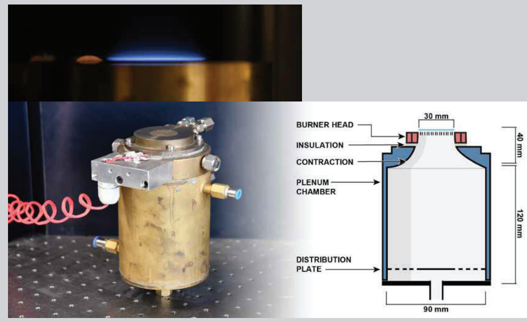
What is it about?

Starting date: 01 Sep 2025
Project type: Experiment & Simulation
Internal project

AmmoniaDrive is a project developing innovative power system, integrating a Solid Oxide Fuel Cell with an Internal Combustion Engine. This hybrid system runs on NH_3/H_2 to help combat global warming. To design and optimize the internal combustion engine, fundamental fuel characteristics are required.

Laminar Burning Velocity is a key combustion property. It is essential for calculating turbulent flame behavior and serves as a critical reference for validating and optimizing chemical mechanisms, which are vital for combustion simulation.

This Master's project aims to measure the laminar burning velocity of NH_3/H_2 using a Heat Flux Burner. The results will be used both for chemical mechanism optimization and by our colleagues in the AmmoniaDrive project for modeling an Internal Combustion Engine.



Experimental Setup: Heat Flux Burner

Laminar Burning Velocity Measurement of NH_3/H_2 using Heat Flux Burner

Why this Project?

i. You are a part of AmmoniaDrive project, a collaboration between multiple universities and industry

ii. Work in the Zero-Emission Lab and gain hands-on experimental experience

iii. Recieve close guidance and supervision

iv. Build solid fundamental knowledege of combustion, useful for academia and industry

What will you do?

i. Literature review of NH_3/H_2 Combustion

ii. Laminar Burning Velocity Measurement using Heat Flux Burner

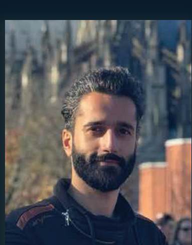
iii. Perform simulation and analysis of results using Cantera/Chem1d

Who are we looking for?

i. Science passionate (pereferably Combustion ;))

ii. Experience with experimental measurement is a plus

iii. Basic knowlege of coding (if not, we'll learn together!)



Who are the supervisors?

MohammadReza Kohansal
m.kohansal@tue.nl
Rob Bastiaans

EINDHOVEN UNIVERSITY OF TECHNOLOGY

General information	
Supervisor	Nico Dam
Mentor	Youri van den Brink
Internal/External	Internal
Exp./Num./Design	Experimental / Design

EINDHOVEN UNIVERSITY OF TECHNOLOGY

3D Particle Tracking in an Acoustic Device using Defocusing Particle Tracking

Youri van den Brink*, Nico Dam
*E-mail: y.v.d.brink@tue.nl

INTRODUCTION

Preventing the release of microplastics (< 100 µm) into the environment is a significant challenge, particularly in wastewater treatment [1]. High-frequency acoustics offers a promising avenue for capturing these particles by establishing standing waves that guide them towards collection points (acoustophoresis). Optimizing the design and performance of these acoustic devices is greatly aided by accurately understanding the complex, three-dimensional particle motion induced by the acoustic fields. While conventional microscopy provides valuable 2D information, it cannot capture the full out-of-plane movement critical for analysis.

General Defocusing Particle Tracking offers an elegant solution. Its key advantage lies in reconstructing 3D particle positions using only a single camera and standard optics, analyzing the characteristic shape changes in defocused particle images (See Figure 1) [2]. This simpler setup eases the integration compared to multi-camera setups. This project aims to develop and implement a defocused tracking system (imaging setup and post processing) to investigate particle dynamics within acoustic devices, enabling device optimization and validation of COMSOL simulation models.

TASKS & GOALS

Design, build (imaging setup & analysis scripts), and validate a defocused tracking system to measure 3D particle motion under acoustic actuation.

Design & Setup: Design and assemble the optical measurement setup compatible with existing equipment.
Processing (MATLAB): Implement core defocused tracking and calibration functions in MATLAB.
Experimentation & Analysis: Record particle motion in acoustic devices; analyze 3D trajectories and potentially compare with COMSOL model results.

STUDENT PROFILE

- Interest in creating measurement systems, involving both hardware setup and MATLAB programming.
- Desire to learn advanced image processing and data analysis techniques.
- Interest in the process of measurement and interpreting experimental findings. Prior optics/tracking experience is beneficial but not essential.

BENEFITS

- Gain hands-on experience building and calibrating an optical imaging system.
- Improve your skills in MATLAB programming, particularly for image processing and data analysis.
- Learn a novel 3D particle tracking technique and apply it to acoustic manipulation.

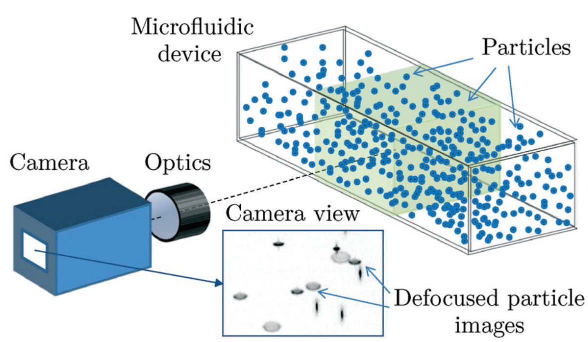


Figure 1: Schematic of the defocused tracking setup showing the camera view capturing defocused particle images within the microfluidic device. [3]

REFERENCE

[1] Talvitieetal.,(2017).WaterResearch,123. DOI:10.1016/j.watres.2017.07.005.
[2] Rossi, M., & Barnkob, R. (2020). A fast and robust algorithm for general defocusing particle tracking. Measurement Science and Technology, 32(1), 014001. DOI: 10.1088/1361-6501/abad71
[3] Barnkob, R., Kähler, C. J., & Rossi, M. (2015). General defocusing particle tracking. Lab on a Chip, 15(17), 3556-3560. DOI: 10.1039/c5lc00562k

General information	
Supervisor	Nico Dam
Mentor	Youri van den Brink
Internal/External	Internal
Exp./Num./Design	Experimental / Design

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Mini-Hydrocyclone for Microplastic Filtration

Youri van den Brink*, Nico Dam
*E-mail: y.v.d.brink@tue.nl

INTRODUCTION

The release of microplastics, particularly synthetic fibers from washing machines, is a major contributor to aquatic pollution. Conventional filtration methods for capturing these sub-millimeter particles often rely on mesh filters that can clog, require frequent replacement, and may not be effective for the smallest particles. Hydrocyclones offer a promising alternative, using centrifugal force to separate particles from a fluid stream without any filter media or consumables [1, 2]. This allows for continuous operation and a robust, low-maintenance design.

While the principle of hydrocyclone separation is well-established, a compact system optimized for this specific application is not yet available. Understanding how performance is affected by the irregular, non-spherical shapes of microplastics (e.g., fibers) is critical for designing an effective filtration system.

This project will characterize the performance of a prototype hydrocyclone system designed for microplastic capture. The system will be built, tested, and potentially optimized using CFD to provide a benchmark for comparison against other mesh-free technologies, such as acoustofluidic filters.

TASKS & GOALS

Design, build, and characterize a hydrocyclone filtration system to benchmark its performance against other separation methods.

Research & Conceptualize: Investigate and propose a suitable hydrocyclone design for microplastic filtration.

Design & Build: Fabricate the hydrocyclone (e.g., (metal) 3D printing/CNC) and assemble the experimental setup.

Integrate & Test: Integrate the hydrocyclone into a test setup and validate its functionality.

Measure & Analyze: Evaluate separation efficiency under various conditions and analyze data (e.g., MATLAB/ Python).

STUDENT PROFILE

- Enjoys hands-on design and the process of building experimental setups.
- Interested in fluid mechanics and particle dynamics.
- An interest in analyzing experimental data to understand and improve system performance.

BENEFITS

- Gain practical experience in designing and fabricating a complete particle separation system.
- Develop skills in experimental planning, data acquisition, and performance characterization.
- Learn about hydrocyclone principles and their practical application in environmental technology.

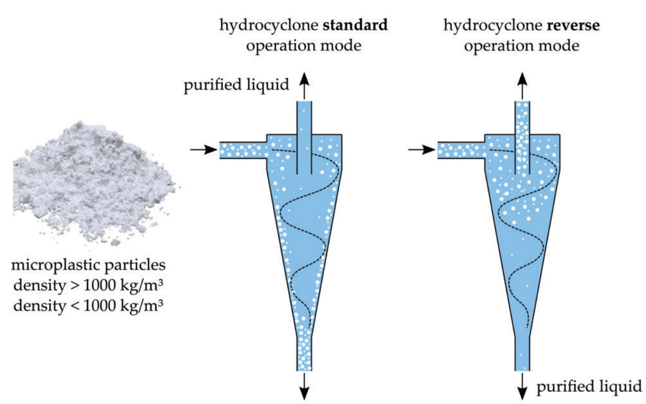


Figure 1: Schematic of hydrocyclone operation modes. In standard mode (left), particles denser than the fluid are separated via the underflow. In reverse mode (right), particles less dense than the fluid are separated via the overflow. [3]

REFERENCE

[1] Liu, L. et al., (2022). Science of the Total Environment, 840. DOI: 10.1016/j.scitotenv.2022.156697
[2] He, L. et al., (2022). Particuology, 71. DOI: 10.1016/j.partic.2022.01.011
[3] Senfter, T. et al., (2024). Microplastics, 3. DOI: 10.3390/microplastics3010002

Supervisor	Conrad Hessels
2nd/3rd supervisor	Noud Maes, Nico Dam
Company	N.A.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Experimental

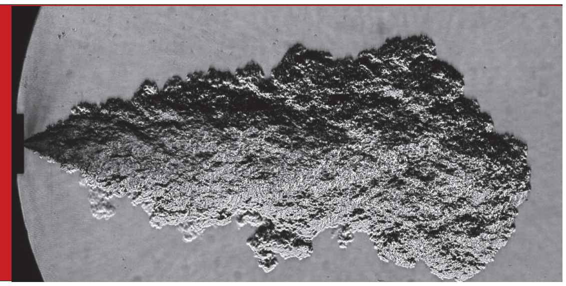
Available for ME, AT, SET

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Measuring the temperature field of a high-pressure H_2 injector using spontaneous Raman scattering.

C.J.M. Hessels*, N.C.J. Maes, & N.J. Dam

*E-mail: c.j.m.hessels@tue.nl



Background

During the last decades, the need for efficient and clean combustion has been growing steadily. With increasing emission legislation and sustainability in mind, hydrogen, along with other renewable fuels, seems viable for the internal combustion engines of the future.

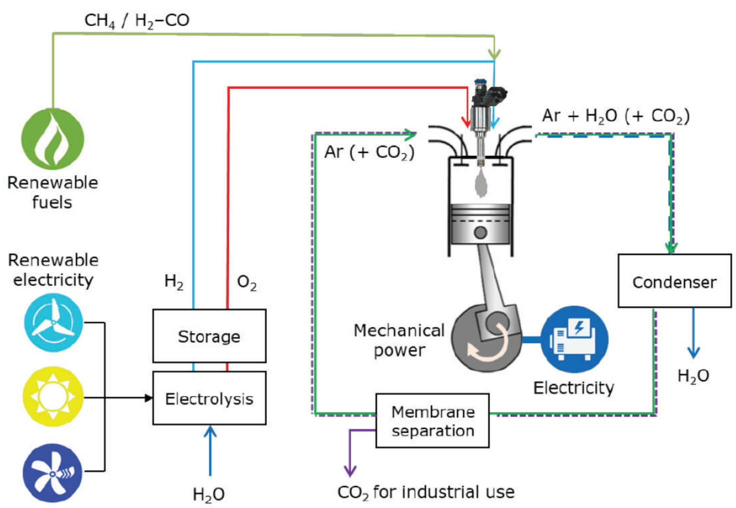


Figure 1: Schematic layout of the Argon Power Cycle.

To increase the efficiency for all these renewable fuels, the revolutionary Argon Power Cycle is investigated. Using Argon instead of air as the working fluid, the cycle efficiency could increase by about 25%, reaching values above 80%!

Working together with Noble Thermodynamic Systems (Berkeley, CA, USA) and IFP Energie nouvelles (Paris, France), non-intrusive temperature measurements of a modified Gasoline Direct Injection (GDI) injector need to be investigated. Earlier research (inverse-LIF & Rayleigh scattering) indicated the importance of compressibility effects inside the jet.

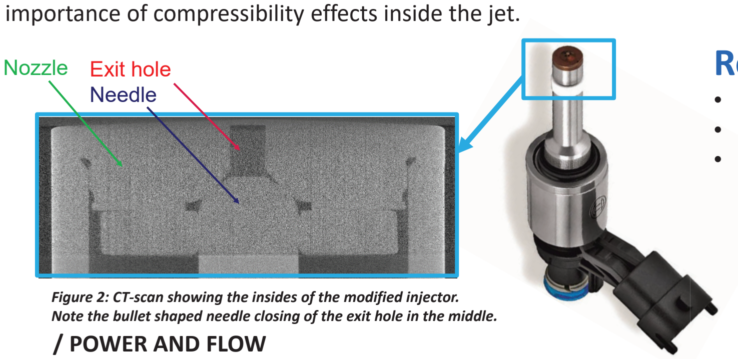


Figure 2: CT-scan showing the insides of the modified injector. Note the bullet shaped needle closing of the exit hole in the middle.

/ POWER AND FLOW

Objective

The decompression of high-pressure hydrogen (during injection) into a lower pressure environment will lead to a drop in temperature and pressure in the barrel shock until the Mach disk is reached. These temperature/density effects influence the mass flow through the nozzle and characterizing them can lead to valuable insights in the needed assumptions/corrections for various optical diagnostics, i.e., Rayleigh scattering (TU/e) and inverse LIF (IFP+TU/e). Raman scattering is selected to perform non-intrusive temperature measurements in the jet.

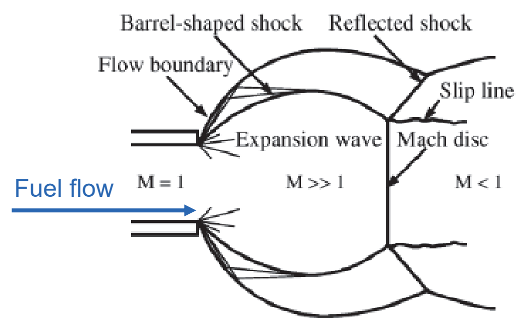


Figure 3: Classical model by Ewan and Moodie[1] on the injection of a compressible fluid.

Approach

You will:

- Study literature and get familiar with Raman scattering on underexpanded jets and earlier performed research.
- Start with Nitrogen through an always-open fuel injector, using an existing & documented setup. (HPVB[2])
- Perform experiments on hydrogen jets and process and analyze the measured data using (existing) MATLAB routines.
- Compare the results with literature and existing Rayleigh & inverse-LIF measurements.

Recommended courses:

- *Optical diagnostics for combustion and fluid flow (4BM40)*
- *Clean engines and future fuels (4AT020)*
- *Experimentation for Mechanical Engineering (4BM20)*

References

1) Structure and velocity measurements in Underexpanded Jets, Ewan & Moodie [1986]
2) Spectroscopy on the verge of soot formation, Robin Doddema [2023]

Supervisors	Xiaoxing Li, Prof. Hans Kuerten
Daily supervisor	Xiaoxing Li
Company	Canon Printing Production
Starting date	Any time
Exp./Num./Design	Numerical

EINDHOVEN UNIVERSITY OF TECHNOLOGY

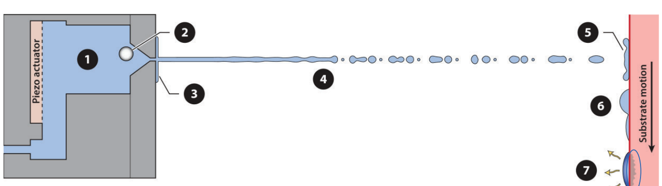
Absorption and Evaporation of Ink in Paper Sheets

Xiaoxing Li, Hans Kuerten
x.li5@tue.nl

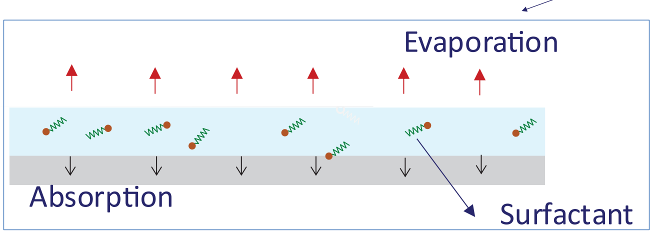
INTRODUCTION

Inkjet printing technology is a common deposition technique with many applications, such as print-based advertising and books. A significant fluid dynamics challenge in inkjet printing is the absorption and evaporation of ink in paper [1-2]. Fast penetration of the ink is desirable to minimize the time droplets remain on the paper [3]. Additionally, fast evaporation can save time and improve efficiency. However, much uncertainty still exists about how to control the absorption and evaporation process of surfactant-laden ink liquid.

Our project has collaborations with Canon printing company.



Lohse D. 2022, Annu. Rev. Fluid Mech.

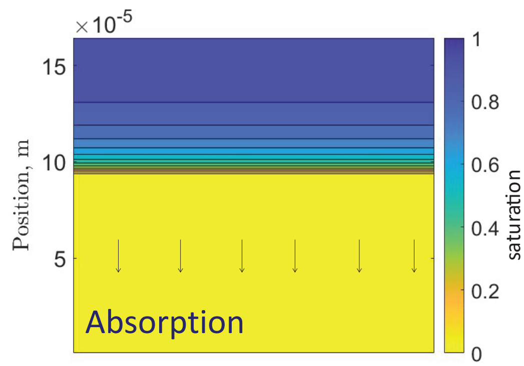


GOALS

The objective of this research is to numerically simulate and study absorption and evaporation of surfactants-laden ink in thin paper sheets.

TASKS

- Summarize the evaporation model in porous media in the literature.
- Integrate the evaporation model into existing models, which govern the absorption process and surfactant transport in unsaturated porous media.
- Write a computer program to solve the mathematical equations developed using the finite volume method in space and an explicit method in time.



STUDENT PROFILE

- Knowledge of MATLAB or Python, or similar programming languages.
- Knowledge of mass transport equations

REFERENCES

[1] Lohse, D. (2022). Fundamental fluid dynamics challenges in inkjet printing. Annual review of fluid mechanics, 54(1), 349-382.

[2] Stenström, S. (2020). Drying of paper: A review 2000–2018. Drying technology.

[3] Daniel, R. C., & Berg, J. C. (2006). Spreading on and penetration into thin, permeable print media: Application to ink-jet printing. Advances in colloid and interface science, 123, 439-469.

General Information	
Supervisor	Noud Maes
Mentor	Ralph Maas
Internal/External	Internal
Exp./Num./Design	Numerical / Design

Numerical investigation of a side-mounted prechamber for hydrogen engines

Ralph Maas*, Noud Maes
***E-mail: j.f.p.w.maas@tue.nl**

INTRODUCTION

Hydrogen internal combustion engines (H_2-ICEs) offer a promising path to decarbonizing transportation but face key challenges such as NO_x emissions, combustion stability, and knock. One approach to address these issues is the use of prechamber ignition systems.

The concept involves the use of a small pre-chamber, separate from the main combustion chamber, where a rich air-fuel mixture is ignited. This ignition generates high-energy turbulent jets of hot combustion products that are forcefully expelled into the main chamber through small orifices.

A previous CFD study using CONVERGE, has investigated a centrally mounted passive prechamber. The results were promising, and the research is continued experimentally. Planned research will be conducted using an optically accessible engine at the Zero Emissions Lab. Due to design constraints of this engine, the prechamber must be located on the side of the cylinder, introducing new complexities.

In this master thesis project, CFD software will be used to gain insights into this novel configuration. The outcomes aim to advance the understanding and feasibility of prechamber-based hydrogen combustion systems.

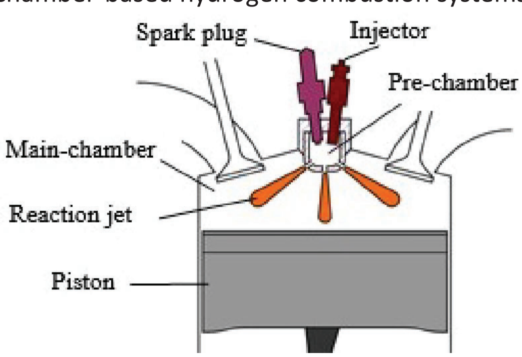


Figure 1: Schematic image of a centrally mounted active prechamber

OBJECTIVE

Improve the understanding of a side-mounted active prechamber on the combustion process inside the engine. This understanding is to be used to optimize the design of the prechamber.

APPROACH

- Short literature study on hydrogen combustion engines and turbulent jet ignitions systems
- Review of existing model [1], and adaptation to include the side-mounted prechamber
- Study the combustion process with the side mounted prechamber and optimize prechamber configuration
- Writing report, presenting result and give advice on the design of the prechamber

STUDENT PROFILE

- Affinity with internal combustion engines
- Good understanding of thermodynamics

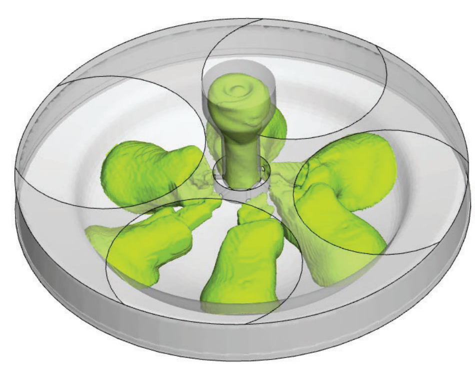


Figure 2: Result from the simulations performed with a centrally mounted passive prechamber [1]

REFERENCE

[1] *Maas, R., Bekdemir, C., and Somers, B., "Numerical Study on the Design of a Passive Pre-Chamber for a Heavy-Duty Hydrogen Combustion Engine," SAE Technical Paper 2024-01-2112, 2024, https://doi.org/10.4271/2024-01-2112.*

Supervisor	Jeroen van Oijen
2nd supervisor	Boyan Xu, R.J.M.Bastiaans, Hesheng Bao
Company	N.A.
Starting date	Anytime
Exp./Num./Design	Numerical

Power &Flow

Modelling bluff-body stabilized flame with FGM model

Jeroen van Oijen, Boyan Xu, R.J.M.Bastiaans, Hesheng Bao
J.A.v.Oijen@tue.nl
b.xu1@tue.nl

Keywords: Turbulent combustion, FGM

INTRODUCTION

- Current simulation [1][2] for bluff body stabilized premixed $NH_3/H_2/N_2$ flame shows good agreement with experiment but needs long simulation time.
- Flamelet-Generated Manifolds (FGM) model can reduce computational time (order-of-magnitude speedups).
- Turbulent-FGM modelling framework has been well established[3] .

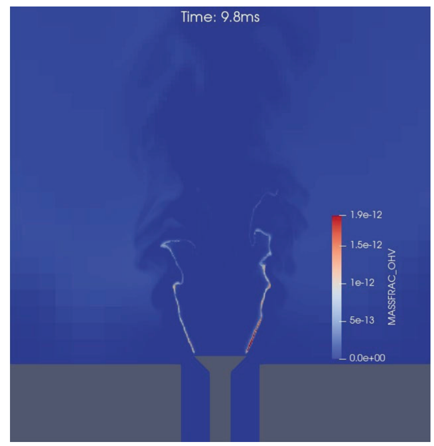


Figure 1. Bluff-body stabilized flame.

TASKS

- Follow the assignment in course "Modelling combustion", build FGM table for premixed $NH_3/H_2/N_2/O_2$ flame using CANTERA /CHEM1D.
- Build the geometry and mesh in ANSYS Workbench (same step as the course assignment) and implement FGM for simulation in OpenFOAM (well established in-house modelling framework).
- Compare the result (flow field & flame position) with previous simulation/experiment.
- Predict the blow-off behavior (analyze the NOx emission).

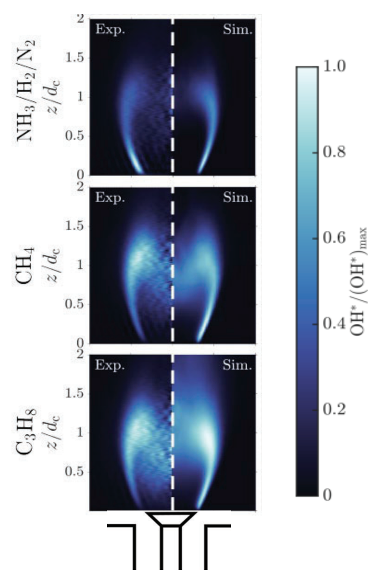


Figure 2. Time-averaged comparison: LES/experiment [1].

REFERENCES

[1] Su, T., Xu, B., Bastiaans, R. J. M., and Worth, N. A. "Lean Blow-Off Behaviour of Premixed Bluff-Body Stabilized Hydrocarbon-Air Flames and Ammonia/Hydrogen/Nitrogen-Air Flames." ASME. J. Eng. Gas Turbines Power. November 2024; 146(11): 111011.

[2] Su, T., Xu, B., Bastiaans, R. J., & Worth, N. A. (2024). The behaviour of NH3/H2/N2, CH4 and C3H8 turbulent premixed bluff-body stabilized flames near lean blow-off. Proceedings of the Combustion Institute, 40(1-4), 105739.

[3]Hesheng B, Hayri Y, Dirk R, Bart S, The inclusion of scalar dissipation rate in modeling of an n-dodecane spray flame using flamelet generated manifold, Combustion and Flame,Volume 249, 2023

Supervisor	Dr. Stein Stoter
2nd supervisor	N.A.
Mentor	Dr. Stein Stoter
Company	N.A.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Numerical

Available for ME

EINDHOVEN UNIVERSITY OF TECHNOLOGY

High-performance finite element-based Navier-Stokes solver

Stein Stoter
E-mail: k.f.s.stoter@tue.nl

Keywords: *CFD, Finite element method, high-performance*

INTRODUCTION

Various advances within the field of computational mechanics have opened the door to the development of highly efficient and scalable Navier-Stokes flow solvers. This project will give you the opportunity to build such a solver (nearly) from scratch. You will use the python finite element library FEniCS (written in C++ and fully supportive of parallel processing), enabling rapid development (python) while being able to use highly optimized finite element routines and linear solvers.

FEM for flow

This project revolves around the use of the finite element method for the flow solver. Most commercial CFD solvers are finite volume based. When tailored correctly, the finite element method can yield pointwise mass conserved flow approximations, a very powerful concept. Additionally, FEM can achieve arbitrary order of spatial convergence on unstructured grids. Both of these concepts will play a role in your project.

Advanced computational methods

Achieving this ambitious goal requieres combining various advanced numerical solution techniques:

- Higher-order mixed implicit/explicity time stepping, to separate non-linear from linear effects.
- Higher-order specialized finite element pairs to achieve exactly divergence-free approximations.
- Hybridization and static condensation of degrees of freedom to reduce the size of the matrix solve.
- Dedicated preconditioning for using efficient iterative solving algorithms.

Your project will either zoom into one of the above points or combine multiple, depending on your interests, goals and progress.

STUDENT PROFILE

- Affinity towards (advanced) numerical solution methods,
- Strong interest in programming and eager to improve upon their existing programming skills (e.g., Python).
- Having taken the course Advanced Discretization will enable you to make a flying start.

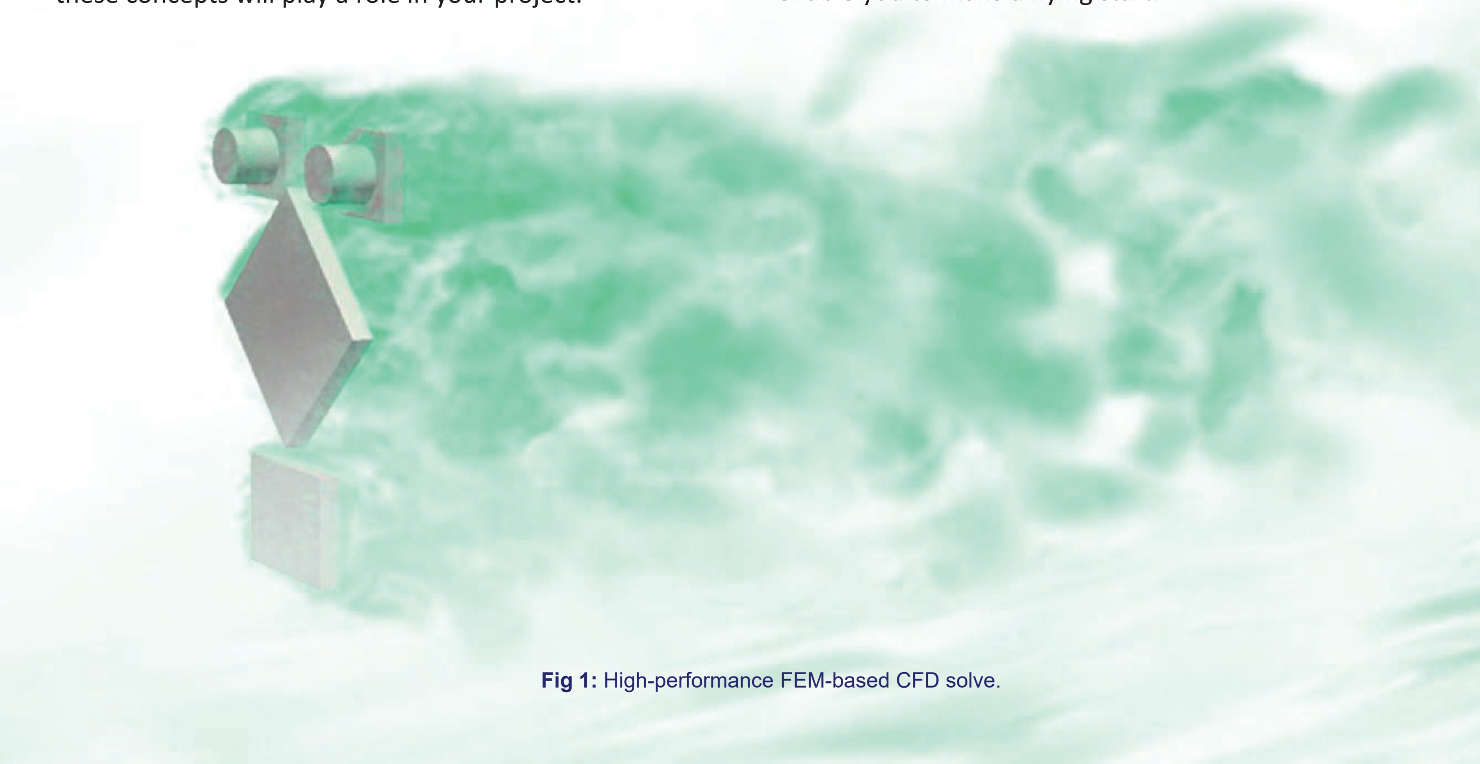


Fig 1: High-performance FEM-based CFD solve.

Supervisor	MSc. Aramesh Mirsaeidi
2nd supervisor	Dr. Stein Stoter
Mentor	
Company	N.A.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Numerical

Available for ME

EINDHOVEN UNIVERSITY OF TECHNOLOGY

The sound of turbulence

Aramesh Mirsaeidi, Stein Stoter
E-mail: k.f.s.stoter@tue.nl

Keywords: *Turbulence modelling, Fourier representation*

INTRODUCTION

Turbulent flow is characterized by the interaction between large-scale and small-scale flow structures, which together govern the transfer of energy and momentum. These structures often exhibit repeating, periodic patterns, providing an opportunity to better understand the flow by "listening" to its dominant frequencies.

RESEARCH TOPICS

- Search for available turbulent flow datasets.
- Analyze a suitable turbulent flow datasets, perform a discrete Fourier Transform, and search for the prevalent modes.
- Investigate how well a hand-full of modes can reproduce the important characteristics.

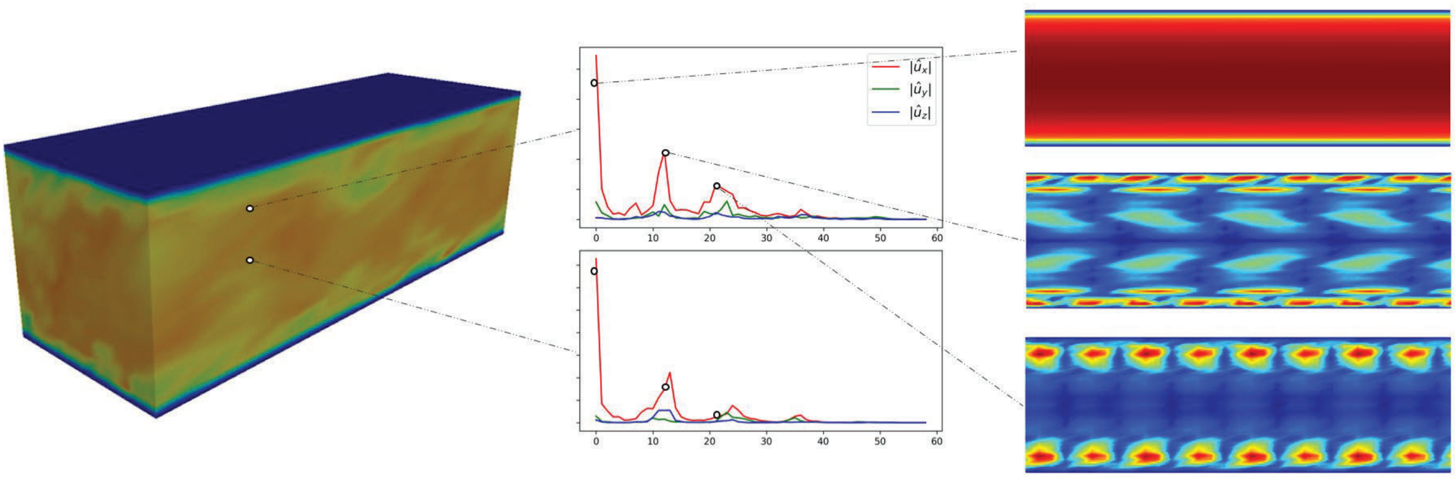


Fig 1: Turbulent flow decomposed into its dominant mode shapes.

Project description

In this project, you will dig into a turbulent flow solution to obtain the fundamental modes that make up the flow field. By you will transfer this information to a sound profile. Performing different band-pass filters, enables you to intuitively investigate the intensity and characteristics of specific frequency components within the flow-induced sound. This approach allows for a clearer understanding of the relationship between flow structures and their acoustic signatures.
Next, you investigate how only a few acoustic modes can preproduce most of the sound of the turbulent flow. This opens the door to model reduction in turbulence models.

STUDENT PROFILE

We are looking for a MSc student who has:

- Affinity towards Fluid Mechanics.
- Interest in programming.

REFERENCES

[1] Pope, S.B. (2000). *Turbulent flows*

Master Thesis Project Proposals
Q1 2025-2026

Energy Technology & Fluid Dynamics
Department of Mechanical Engineering
Eindhoven University of Technology

Preface

This is an overview of all the Master Graduation project proposals available in Energy Technology & Flow Dynamics.

Please select 3 choices of different projects in order of preference and write a **short motivation** for your first choice to Azahara Luna-Triguero (a.luna.triguero@tue.nl).

Example:

- My first preference is project... because I am very motivated to work on...
- Second preference is... (no motivation needed)
- Third preference is.. (no motivation needed)

If you need more information on a proposal you can contact directly one of the supervisors (the emails are in each project proposal).

Supervisor	Dr. Clemens Verhoosel
2nd supervisor	
Mentor	
Company	TouchWind BV.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Experimental

Available for ME

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

Validation study on the wake behavior of wind farms with tilted turbines

Clemens Verhoosel
*E-mail:

Keywords: *Validation, CFD, Rotor tilt, Wake steering*

INTRODUCTION

Climate change is a big problem facing us today. There is a strong need for renewable energy sources and it is increasing each year. TouchWind is developing a floating offshore wind turbine. As a startup, it is hard to compete with the already existing turbine manufacturers. However, with the design of a tilting one-piece rotor, TouchWind believes that the cost per kWh can decrease. Furthermore, TouchWind estimates that tilted turbines can be placed closer to each other and thus occupying less space.

Figure 1: TouchWind rotor during normal operating conditions.

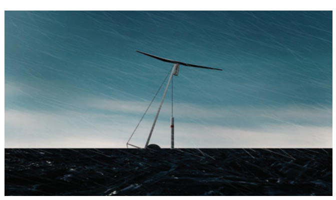


Figure 2: TouchWind rotor during storm conditions.

Both beliefs arise from the so-called 'Park effect' of the TouchWind turbines. The TouchWind rotors can tilt. This means that the rotor will be, as visible in Figure 1 and 2 , positioned at an angle relative to the wind. This has a lot of benefits for one turbine itself, but with multiple turbines placed in a row, more benefits can arise. The benefits that arise by placing multiple turbines is called the TouchWind park effects.

Wind turbines form wakes downwind of the turbines. Wakes are the more turbulent and less energy rich flow that form due to the energy extraction of wind turbines. By applying tilt to a rotor the wake gets redirected downwards creating a new fresh stream of energy for the downwind placed turbines. This phenomena is visualized in Figure 3. Research has shown that with a wind gradient present, even higher energy outputs can be reached for a wind farm with tilted turbines then for a conventional wind farm.

However, tilted rotors are not commonly used and so only a little information is known about the airflow/wake behavior of these turbines. With the use of CFD simulation, TouchWind has gained better insight in the wake behavior. To ensure that LES simulations accurately reflect real-world behavior, validation with experimental data is essential. The goal of this assignment is to contribute to the development of a validated CFD model for a tilted wind turbine.

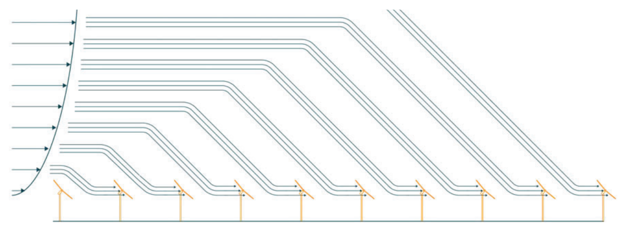


Figure 3: TouchWind rotor during storm conditions.

TASKS

- Conduct research into how LES simulations can best be validated using experimental data.
- Develop an experimental plan focused on tilted turbines, aligned with the required validation insights.
- Support or conduct experiments and compare the measured data with LES simulation results.

STUDENT PROFILE

We are looking for a MSc student who:

- Is eager to contribute to the validation of aerodynamic models for wind energy applications.
- Has interest in working in a start-up. This involves working in a small team, but also work independent.
- Has interest in working in the sustainable energy market contributing to creative energy solutions.

TouchWind
Pulling Forward

Supervisor	Arjan Frijns
Daily supervisor	Bas Gieling
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Numerical

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Extending the DSMC method for Polyatomic Gas Mixtures in Rarefied Flow Regimes

Arjan Frijns*, Bas Gieling, Michael Abdelmalik

*E-mail: a.j.h.frijns@tue.nl

Introduction

When Size Shrinks or Pressure Drops

As pressures decrease or system sizes shrink, traditional fluid behavior changes dramatically. Molecular interactions become important, and rarefaction effects dominate the flow.

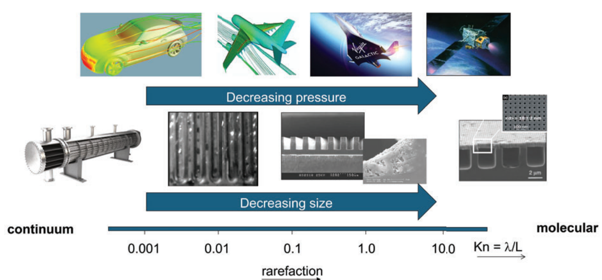


Figure 1: Rarefaction effects occur when a system's pressure or size decreases.

Where This Matters in Industry

Some examples:

- Semiconductor Lithography
  Micro-scale vacuum chambers
- Spacecraft Re-entry
  Low-pressure atmospheric conditions
- Carbon Capture Technologies
  Gas transport in micro-porous materials

Why Traditional Models Fail

In these transition and free-molecular flow regimes, the Knudsen number is high, meaning the assumptions behind continuum models like the Navier-Stokes or Euler equations no longer hold.

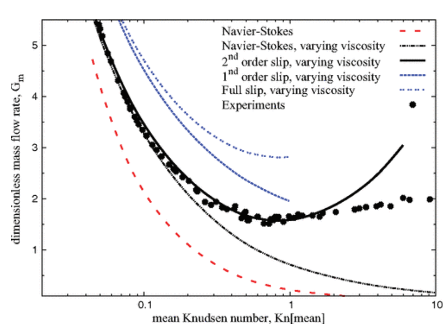


Figure 2: The mass-flow rate through a micro-channel attains a minimum in the transition regime, which continuum models cannot predict. *Figure from Kokou Dadzie, S. and Brenner, H. (2012). DOI: 10.1103/PhysRevE.86.03631*

Project

Aim

To design and optimize systems in rarefied flow regimes, industry requires accurate physical models that go beyond classical fluid dynamics.

This project focuses on:

- Extending the Direct Simulation Monte Carlo (DSMC) method from monatomic to polyatomic molecules.
- Developing a structure-preserving Machine Learning model to infer the collision operator from particle-based simulations.
- Implementing this model within DSMC and comparing results with a Boltzmann solver.

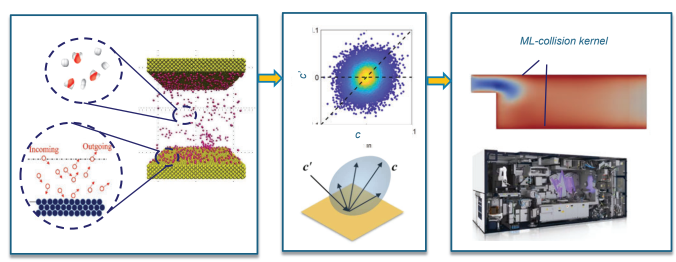


Figure 3: The multiscale approach: Molecular dynamics → ML-based stochastic kernels → DSMC

Project Tasks

As a student researcher, you will:

- Conduct a literature review on rarefied flows, collision operators for polyatomic mixtures, and particle-based methods.
- Implement the new numerical model within our existing framework.
- Assess results both qualitatively and quantitatively.
- Conclude and provide recommendations on modeling and numerical implementation.

Student Profile

We are looking for a Master's student who:

- Has a strong interest in advanced numerical methods.
- Enjoys programming and is eager to enhance his/her skills (e.g., in Python).
- Is curious, analytical, and motivated to contribute to cutting-edge research.

Supervisor	Camilo Rindt
2nd supervisor	Michel Speetjens
Daily supervisor	Nikolaos Georgousis
Company	VITO
Starting date	As soon as possible
Exp./Num./Design	Experimental

ETFD

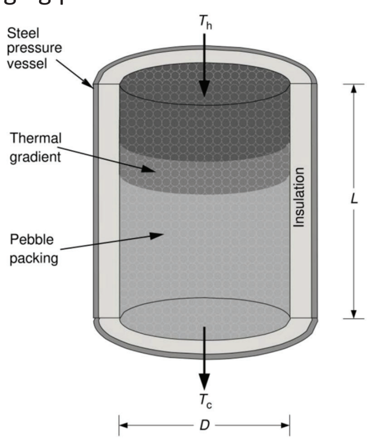
EINDHOVEN UNIVERSITY OF TECHNOLOGY

EXPERIMENTAL INVESTIGATION OF FULL CYCLE PACKED-BED THERMAL STORAGE OPERATION

Camilo Rindt: c.c.m.rindt@tue.nl

INTRODUCTION

Packed-bed thermal storage (PBTS) systems are thermal energy storage (TES) devices and they typically constitute of a storage tank and a solid phase material, in the form of packing elements (PEs) for the storage of heat. Also, a heat transfer fluid (HTF), either liquid or gas, is used to exchange heat with the PEs[1]. Precise computation of the HTF and PEs temperature fields inside the storage tank are of great importance for the analysis and design of PBTS systems. The numerical analysis of the PBTS systems is commonly done through a volume-averaging technique for the porous media structure solid and the HTF phases (2 phases). In literature a common numerical model used is the 2 phase – 1 dimensional (2P-1D)[2] model which can calculate the HTF and PEs temperature distribution in the axial direction of the PBTS. However, in literature there is a lack of experimental studies validating the 2P-1D numerical model under full cycle operating conditions. A full cycle consists of a charging, stand-by and discharging phase.

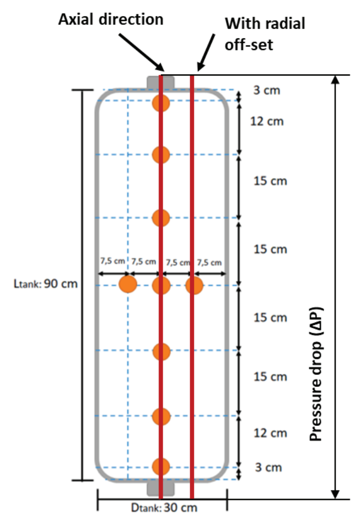


Keywords: *PBTS systems, experimental validation, full cycle operation, characteristic dimensionless numbers*

TASKS

The main expectations of this MSc project are:

- Construction of a vertically oriented PBTS system with a cylindrical storage tank (HTF: preferably air, PEs: spheres or rocks).
- Conducting full cycle experiments and measuring at least the HTF (or PEs) temperature at various axial and radial positions and the pressure drop across the PBTS height.
- Conducting experiments under conditions similar to those in a real PBTS system by using a set of characteristic dimensionless numbers, defined for PBTS systems.



GOALS

- Designing a PBTS setup that allows accurate measurements under various experimental conditions.
- Conduct a thorough experimental validation of the commonly used 2P-1D numerical model.

STUDENT PROFILE

We are looking for a highly motivated MSc student with interests in energy technology and experimental investigations for this quite unexplored scientific field. Any experience with the design of an experimental set up is appreciated. (A 2P-1D numerical model, coded in Python, can be provided for the numerical validation of the experimental temperature and/or pressure drop measurements.)

REFERENCES

[1] McTigue JD *et al.* (2018), Performance of packed-bed thermal storage to cycle duration perturbations, Journal of Energy Storage, (19) 379-392.

[2] Kocak B & Paksoy H (2019), Performance of laboratory scale packed-bed thermal energy storage using new demolition waste based sensible heat materials for industrial solar applications, Solar Energy, (211) 1335-1346.

Supervisor	Paul Grassia
2nd supervisor	Paul Grassia
Starting date	Flexible
Exp./Num./Design	Modelling/Numerical

Bubble Trains Flowing in a Channel

Paul Grassia*
p.s.grassia@tue.nl,

INTRODUCTION

There are many scenarios in which trains of bubbles flow along narrow channels (e.g. foam-based gas storage, foam-based soil remediation). As throughput is increased in such processes, there is a risk that viscous drag forces will break the train of bubbles apart. However it is also possible that the structure can stay together provided foam films between bubbles flatten out [1]. This project will explore the geometry of such flat film states.

GOALS

The goal is to develop models establishing limits on bubble sizes that can stack into a flat film state as a function of the number of bubbles within a train. This will in turn identify the domain of bubble sizes that admit rapid throughput within a channel or porous medium.

BENEFITS

You will be studying a system which admits a rich physical behaviour, but which simultaneously can be used in engineering practice. You will also be studying an unconventional class of models in which dynamics is largely replaced by geometry.

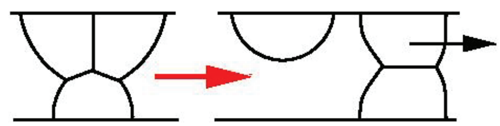


Figure 1: A bubble train that breaks up

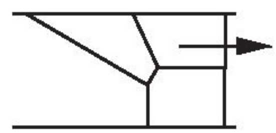


Figure 2. A flat film state that does not break

PROFILE

The project requires a student with an interest in foams and an understanding of and liking for geometry. Programming experience is also an advantage.

REFERENCES

[1] C. Torres-Ulloa and P. Grassia. Viscous froth model applied to the motion and topological transformations of two-dimensional bubbles in a channel: Three-bubble case. Proc. Roy. Soc. London Ser. A, 478:20210642, 2022 doi: 10.1098/rspa.2021.0642.

Supervisor	Paul Grassia
2nd/3rd supervisor	Maja Rucker/David Rieder
Starting date	Flexible
Exp./Num./Design	Modelling/Numerical

ETFD

Foamed Gas Flow in Porous Media for Energy Storage

Paul Grassia, Maja Rucker, David Rieder
p.s.grassia@tue.nl, m.rucker@tue.nl, d.r.rieder@tue.nl

INTRODUCTION

There are many scenarios which involve gas displacing other fluids in porous media (e.g. soil/aquifer remediation, CO2 capture/storage, seasonal storage of green hydrogen). However the high mobility of gas in porous media leads to flow control issues (gas override, fingering and channelling phenomena). These can be overcome by foaming the gas, which reduces its mobility by orders of magnitude. Looking inside porous media to see how foamed gas distributes remains however challenging. Hence it is useful to have models describing both foamed gas mobility and how foamed gas flows. One additional challenge in the case of seasonal hydrogen storage is that stored gas eventually needs to recovered by flowing gas out of the medium again. Models suggest however that foamed gas flowing back out of a medium has an even lower mobility than foamed gas flowing in [1]. This impact of this upon how readily hydrogen can be recovered from porous media is not yet understood.

GOALS

The goal is to develop a computer simulation for foamed gas moving in porous media. The model will be used to design geological gas storage processes with a balance being sought between the amount of gas stored and the subsequent ease/rapidity of extracting it.

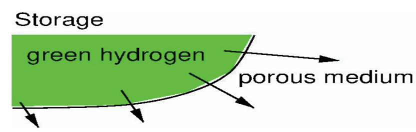


Figure 1: Seasonal storage of green hydrogen

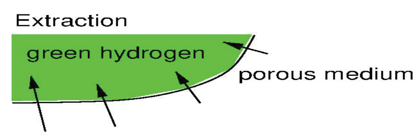


Figure 2. Seasonal extraction of green hydrogen

BENEFITS

The project will contribute to the goals of energy transition. Technology already exists to generate green hydrogen from renewable sources (solar, wind), but challenges remain in storing it safely and in large quantities.

PROFILE

The project requires a student interested in porous media and multiphase flows. Experience with numerical methods for partial differential equations and with computer programming will also be an advantage.

REFERENCES

[1] M. Eneotu and P. Grassia. Foam improved oil recovery: Towards a formulation for pressure-driven growth with flow reversal. Proc. Roy. Soc. London Ser. A, 476:20200573, 2020 doi: 10.1098/rspa.2020.0573.

Supervisor	**See below**
2nd supervisor	**N.A.**
Mentor	**N.A.**
Company	**N.A.**
Internal / External	**Internal**
Starting date	**Any time**
Exp./Num./Design	**Numerical**

Available for ME

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Computational methods in Thermal Fluids Engineering

Clemens Verhoosel, Harald van Brummelen, Michael Abdelmalik or Stein Stoter

Keywords: *Computational Mechanics*

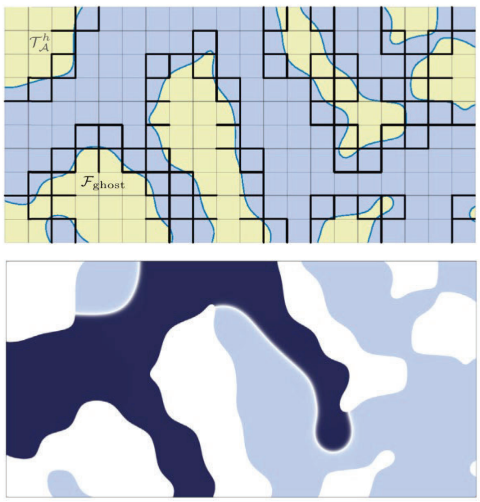


Fig 1: Multi-phase flow through porous material

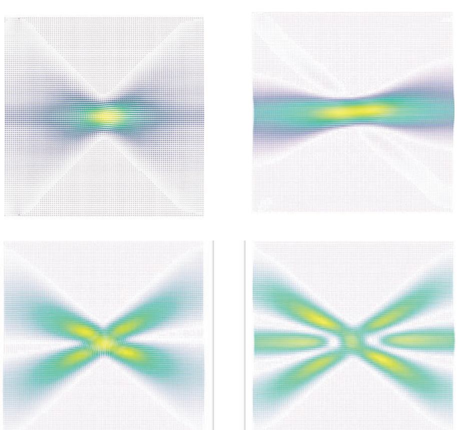


Fig 2: Model order reduction

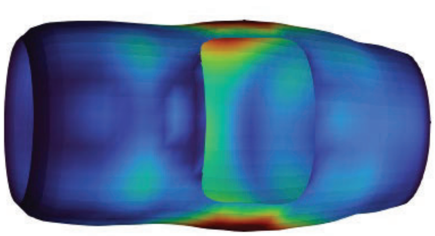


Fig 3: Explicit analysis of a shell-structure

INTRODUCTION

Computational method development plays a crucial role in advancing scientific research and engineering applications. As the complexity of real-world problems continues to grow, there is an increasing demand for efficient, accurate, and robust numerical methods to simulate, analyze, and predict complex systems.
Within this Computational Methods in Thermal Fluids Engineering consortium, we offer a broad scope of master thesis projects on solving complex problems across various scientific and engineering domains.

Project focus

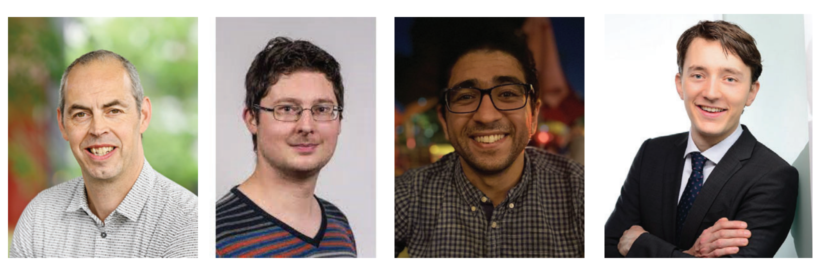
The specific direction of the project can be tailored to your interests, allowing you to work on topics ranging from fluid dynamics and structural mechanics to machine learning and high-performance computing. If you're motivated to tackle challenging problems and eager to make a meaningful impact through computational innovation, we encourage you to reach out to one of the supervisors below to discuss ideas and potential project paths.

STUDENT PROFILE

We are looking for a MSc student who has:

- Affinity towards (advanced) numerical solution methods,
- Interest in programming and eager to improve upon their existing programming skills (e.g., Python).

Contact



From left to right:

- Harald van Brummelen — E.H.v.Brummelen@tue.nl
- Clemens Verhoosel — CVerhoose@tue.nl
- Michael Abdelmalik — M.Abdel.malik@tue.nl
- Stein Stoter — K.F.S.Stoter@tue.nl

Supervisor	Rick de Lange
2nd supervisor	-
Company supervisor	David van Venrooij
Company	Weheat
Starting date	-
Exp./Num./Design	Exp/Design

EINDHOVEN UNIVERSITY OF TECHNOLOGY

weheat

Smart EXV superheat control in residential heat pumps

Rick de Lange
H.C.d.Lange@tue.nl

Introduction

Weheat develops state-of-the-art heat pumps for central heating systems. Reliability and performance are large contributors to Weheat's success. Superheat is a critical component in the efficiency and safety of our heat pump operations. It ensures the refrigerant fully transitions into vapor before compression, crucial for both the system's efficiency and safety. Your graduation assignment will involve developing and optimizing control strategies for this intricate and captivating process.

Objective

The goal of this thesis is to develop and implement a robust control method for the electronic expansion valve (EXV) in our heat pumps, ensuring precise superheat control while adapting to dynamic operating conditions. The control algorithm, implemented in C or Python, must effectively handle a wide range of air and water temperatures, adapt to varying load conditions, compensate for sensor inaccuracies and phase shifts, and be transferable across our entire heat pump catalog.

Goals

- **Develop a control method** for the expansion valve which can handle a varyity of load conditions
- **Validate its performance** and robustness, extensive testing will be conducted in the in-house climate chamber, which allows for simulations across the full operating range. The control method will be evaluated under extreme conditions, such as sudden changes in temperature and flow, rainfall on the evaporator, and evaporator freezing.

This thesis presents a challenging theoretical basis of control theory and thermodynamic behavior while providing extensive hands-on opportunities to test and refine hypotheses in a controlled test environment.

Supervisor	Ir. Ronald Lamers
2nd supervisor	Dr. Michel Speetjens
Mentor	
Company	Thermo Fisher Scientific
Internal / External	External
Starting date	Any time
Exp./Num./Design	Num./Exp.

Available for ME-SET

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

MODEL-BASED THERMAL DRIFT COMPENSATION FOR ELECTRON MICROSCOPES

ThermoFisher SCIENTIFIC
The world leader in serving science

Ronald Lamers, Michel Speetjens*
*E-mail: m.f.m.speetjens@tue.nl

INTRODUCTION

Thermo Fisher Scientific develops electron microscopes capable of visualizing individual atoms. A common problem for these devices is so-called “thermal drift”, i.e. positioning inaccuracies of O(nm/min) due to thermal expansion, which can already be caused by heat loads of O(mW). Thermal drift can often be mitigated by a microscope design that is less sensitive to thermal deformation and/or by reducing the drift (heat) sources. However, if insufficient, then a further option is active adjustment of the position of the stage holding the sample to counteract the thermal drift. Essential for such methods for “thermal drift compensation” is to understand the thermo-mechanics behind the drift and translate this in predictive models for practical use.

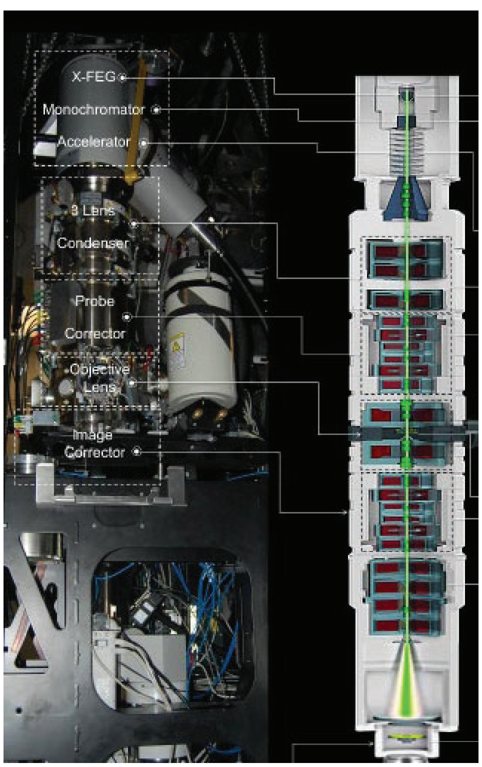


Fig. 1. Typical electron-microscope system.

MODELLING ISSUES

Modelling techniques for predicting thermal drift exist yet their application to realistic systems has proven challenging in previous studies done at Thermo Fisher Scientific. This stems primarily from fundamental assumptions in the early modelling stages. However, the impact of such assumptions on the prediction accuracy and how to improve current models while maintaining practical usability remains an open question.

PROJECT

The main project goal is the ***development of new models for thermal drift compensation*** in electron microscopes. This will be done via a modular test set-up at Thermo Fisher based on an actual microscope and involves the following steps and subgoals:

- Literature study on the impact of thermo-mechanical effects on positioning accuracies in complex industrial precision systems and ways to model and tackle this.
- Identify assumptions and factors causing mismatches between current models and test set-up such as e.g. convective/radiative boundary conditions, simplified geometries, contact resistances, uncertain material properties, impact of vacuum, measurement errors.
- Use insights from the above literature study and analysis of the test set-up versus current models to develop new models for thermal drift compensation.
- Develop an experimental strategy that enables systematic validation and testing of the new models using the test set-up and its instrumentation.
- Further develop and fine-tune the new models using insights gained from the experimental study.

Supervisor	Dr. Michel Speetjens
2nd supervisor	Ir. Benno van der Werff (IF Technology)
Mentor	
Company	IF Technology
Internal / External	External
Starting date	Any time
Exp./Num./Design	Num./Exp./Design

Available for ME-SET

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

SUSTAINABLE HEATING FOR THE BUILT ENVIRONMENT

Michel Speetjens*, Benno van der Werff
*E-mail: m.f.m.speetjens@tue.nl

INTRODUCTION

The engineering & consultancy company **IF Technology** (https://iftechnology.nl) specializes in development of sustainable energy systems. One area of application concerns *sustainable heating & cooling of the built environment* using technologies such as e.g. geothermal or aqua-thermal systems and heat pumps.

RESEARCH TOPICS

1. Performance of thermal-energy storage systems

Two promising forms of thermal energy for heating & cooling of the built environment are:

- Aqua-thermal energy (Fig. 1);
- Acquifier Thermal Energy Storage (ATES).

Both systems rely on the seasonal availability of energy, which results in "charging" and "discharging" of a reservoir (i.e. a body of water or a subsurface porous rock layer) either directly by exchange of water or indirectly via a heat pump. Efficient design and operation is essential for an optimal and sustainable performance of such systems.

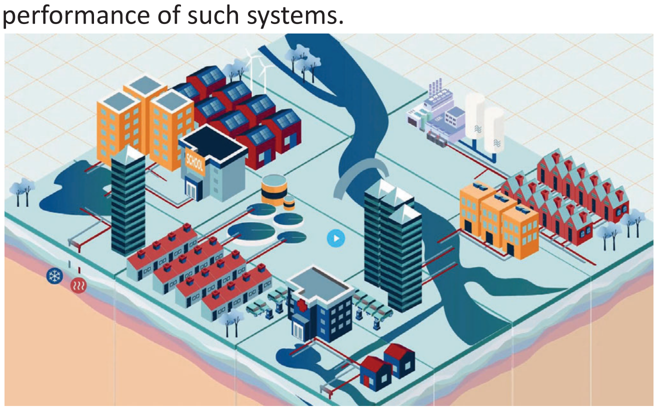


Fig. 1. Heating & cooling of the built environment by aqua-thermal energy (from IF Technology).

Challenges within this scope include:

- Thermal and hydraulic modelling of the seasonal heat exchange of the thermal-energy reservoirs.
- Optimization of the seasonal (dis)charging for both technical and economic performance.
- The impact of variable operating temperatures on the COP of the heat pump and the overall efficiency.

2. Integration of power-to-heat (P2H) systems

Intermittency and fluctation due to weather is a major challenge for sustainable electricity generation by e.g. solar panels and wind turbines. Electricity storage in e.g. Li-ion batteries can mitigate this yet is a costly option. The fact that a substantial part of the generated electricity is eventually used for heating naturally advances storage of electricity in the form of thermal energy via heat pumps (so-called "power-to-heat" or P2H) as a promising alternative. Challenges include:

- Integration of the P2H concept in existing thermal-energy systems & energy networks.
- Techno-economic optimization of such integrated energy systems for certain ranges of energy prices.
- Techno-economic feasibility of heat pumps for P2H applications: dynamic response to electricity variations; economic viability of P2H units; optimal P2H operation on basis of predicted heat demands.

MSc PROJECTS

MSc projects can be defined for both research topics (or closely related topics) and tailored to personal interests and preferences. We invite students with an interest in these topics to contact us (via m.f.m.speetjens@tue.nl) to discuss opportunities for MSc projects.

Supervisor	Ir. Wouter Kuyper, Dr. Theo Ruijl
2nd supervisor	Dr. Michel Speetjens
Mentor	
Company	VDL ETG and MI Partners
Internal / External	External
Starting date	Any time
Exp./Num./Design	Num./Exp.

Available for ME-SET

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

MODELING THERMAL CONTACT CONDUCTANCE IN MECHATRONIC SYSTEMS

VDL ETG

MI PARTNERS PARTNERS IN MECHATRONIC INNOVATION

Wouter Kuyper (VDL), Theo Ruijl (MI), Michel Speetjens*
*E-mail: m.f.m.speetjens@tue.nl

INTRODUCTION

Thermal effects are often important error sources in high-end mechatronic systems (e.g. high-precision production equipment, instruments) in that sub-Kelvin temperature changes can easily cause micro-meter deformations. Many applications operate under near-vacuum conditions and thermal contact conductance at mechanical interfaces then often dominates thermal system behaviour. However, models for predicting such behaviour remain very inaccurate due to the still limited understanding of thermal contact conductance. Existing methods mostly employ a statistical approach to model the interaction between roughness asperities. Their distribution determines a critical model parameter: the interfacial contact area. The thermal contact resistance namely ensues from the contraction of the heat-flux lines from the bulk to the contact areas (see figure). Determining the interfacial contact area is therefore extremely important to accurately model the thermal contact conductance and predict the system behaviour

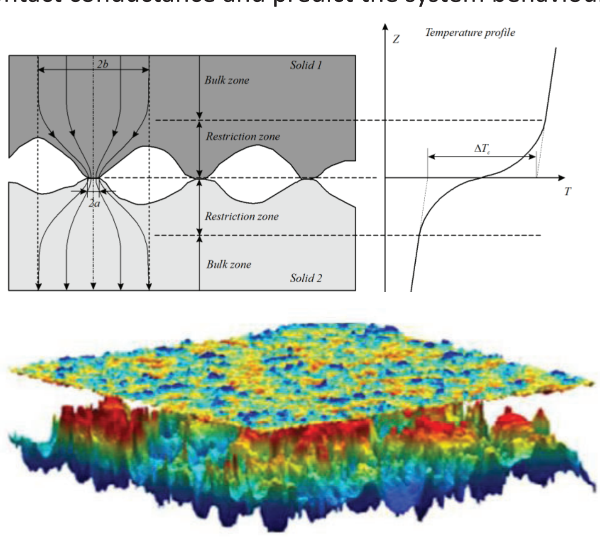


Fig. 1. Top: thermal contact resistance (in terms of temperature drop ΔT_c) due to contraction of heat-flux lines from bulk to contact areas (width 2a). Bottom: contact pressure (blue/red=low/high) between 2 rough surfaces

PROJECT

The main project goal is the ***development of a model for the accurate prediction of thermal contact conductance*** between real engineering surfaces. Key to this end is the ability to predict the real contact area as a function of applied compression pressure and relevant roughness parameters. This involves the following steps:

- Literature study on (i) classical contact mechanics, (ii) contact mechanics at real roughness asperities and ways to model this (iii) models for thermal contact conductance as a function of the contact mechanics.
- Specification of rough surfaces with well-defined characteristics (e.g. non-flatness and waviness) that are representative of real engineering surfaces.
- Development of analytical/numerical models for contact mechanics for these representative surfaces.
- Development of numerical models for simulation of temperature distributions and heat transfer within/between these representative surfaces.
- Development of predictive (analytical/numerical) models for thermal contact conductance for these representative rough surfaces from the simulated temperature distributions and heat transfer.
- Experimental validation of thermal conductance models on real problems (time permitting).

ABOUT VDL AND MI-PARTNERS

MI-Partners is specialized in the development of high-end mechatronic systems for e.g. production processes and measurement devices. **VDL Enabling Technology Group** (ETG) works on innovating state-of-the-art industrial systems with a (growing) focus on solutions for thermal effects in precision engineering.

Supervisor	Dr. Michel Speetjens
2nd supervisor	Dr. Erik Steur (Dynamic & Control)
Mentor	
Company	N.A.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Num./Exp.

Available for ME-SET

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

OPTIMAL HEAT TRANSFER BY SMART FLOW CONTROL

Michel Speetjens*, Erik Steur

*E-mail: m.f.m.speetjens@tue.nl

INTRODUCTION

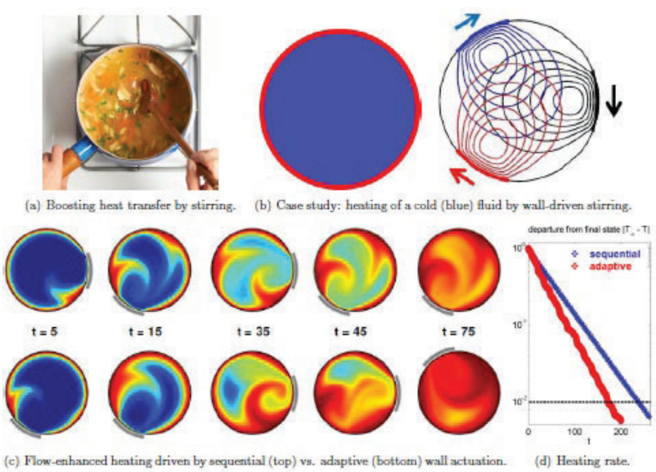
Thermal transport phenomena in flows is relevant to many (industrial) applications:

- heating and cooling in the processing industry;
- thermal management of power electronics;
- thermal conditioning of high-precision systems.

The common goal in such applications essentially is accomplishment of maximum heat transfer between an object and a flow via active manipulation of the flow.

CASE STUDY

Boosting of heat transfer by stirring of the fluid (Fig. 1a) is adopted as a case study. An industrial implementation of this principle exists in the heating of an initially cold fluid inside a circular container via the hot boundary (Fig. 1b, left) [1]. Stirring occurs via switching between the three flows that each are driven by a sliding wall (arrows) along apertures (heavy arcs) in the boundary (Fig. 1b, right). **The key question is: *"<u>How</u> to stir?"***



(a) Boosting heat transfer by stirring. (b) Case study: heating of a cold (blue) fluid by wall-driven stirring.

(c) Flow-enhanced heating driven by sequential (top) vs. adaptive (bottom) wall actuation. (d) Heating rate.

Figure 1: Accomplishing optimal heat transfer in fluid flows by "smart" flow control.

Stirring by <u>sequential</u> actuation of the sliding walls is commonly regarded the best way to boost heat transfer (Fig. 1c, top). However, this approach is sub-optimal [1].

<u>Adaptive</u> stirring by a controller that step-wise actuates the sliding walls on the basis of the most effective temperature evolution (Fig. 1c, bottom) can namely significantly accelerate the heating rate (Fig. 1d) [1].

RESEARCH TOPICS

The above findings demonstrate the potential of dedicated flow control for heat-transfer enhancement and its further exploration (by the case study of Fig. 1) is the principal goal of this research line. Specific research topics within this scope include (but are not limited to):

- Further development of the control strategy of [1] for achieving maximum heat transfer.
- Development of control strategies for other control targets (e.g. establishment of thermal fronts).
- Data-based modelling of the temperature evolution using (nonlinear) system identification.
- Development of an observer that enables state estimation from discrete sensor data.
- Experimental investigation & validation of adaptive flow control using the laboratory set-up of [2].

A multi-disciplinary approach involving expertise on heat & flow versus dynamics & control is essential to address these topics and therefore research is performed as a collaboration between research groups *Energy Technology & Fluid Dynamics* and *Dynamics & Control*.

MSc projects can be defined around any of the above topics and tailored to personal interests and preferences. We therefore invite students with an interest in these (and related) multi-disciplinary topics to contact us to discuss opportunities for MSc projects.

REFERENCES

[1] R. Lensvelt, M.F.M. Speetjens, H.Nijmeijer, Fast fluid heating by adaptive flow reorientation, Int. J. Therm. Sci. **180**, 107720 (2022).

[2] O. Baskan, M. F. M. Speetjens, G. Metcalfe, H. J. H. Clercx, Experimental and computational study of scalar modes in a periodic laminar flow, Int. J. Therm. Sci. **96**, 120 (2015).

Supervisor	**Frank Peters**
Company	**Suncom Energy**
Location	**Houten (Utrecht), the Netherlands**
Starting date	**Q4 2025 / Q1 2026**
Exp./Num./Design	**Experimental, numerical and design**

Mitigation and Validation of Thermal Ratcheting in Packed-Bed Thermal Energy Storage

Frank Peters

*E-mail: .F.Peters@suncom-energy.com

INTRODUCTION

Suncom is a Dutch start-up developing concentrated solar thermal (CST) systems that deliver clean industrial heat up to 425 °C. To be able to provide this energy at all times, Suncom is developing a thermal energy storage (TES) system using a steel tank filled with solid material such as basalt, magnetite or steel slag. One of the main challenges in such a system is thermal ratcheting. With each thermal cycle, the steel tank expands more than the filler. Over time, this mismatch causes the rocks to settle and shift, leading to internal stress build-up and permanent deformation of the tank wall. This must be resolved to ensure safe and long-lasting operation. Through the European TESSOTOF project, Suncom collaborates with Fraunhofer and Dehoust to solve this issue using simulations, lab tests, and a full-scale validation. The goal is to develop a robust and scalable TES solution that lowers the Levelized Cost of Heat (LCOH) and makes solar heat a reliable, cost-effective alternative to fossil fuels.

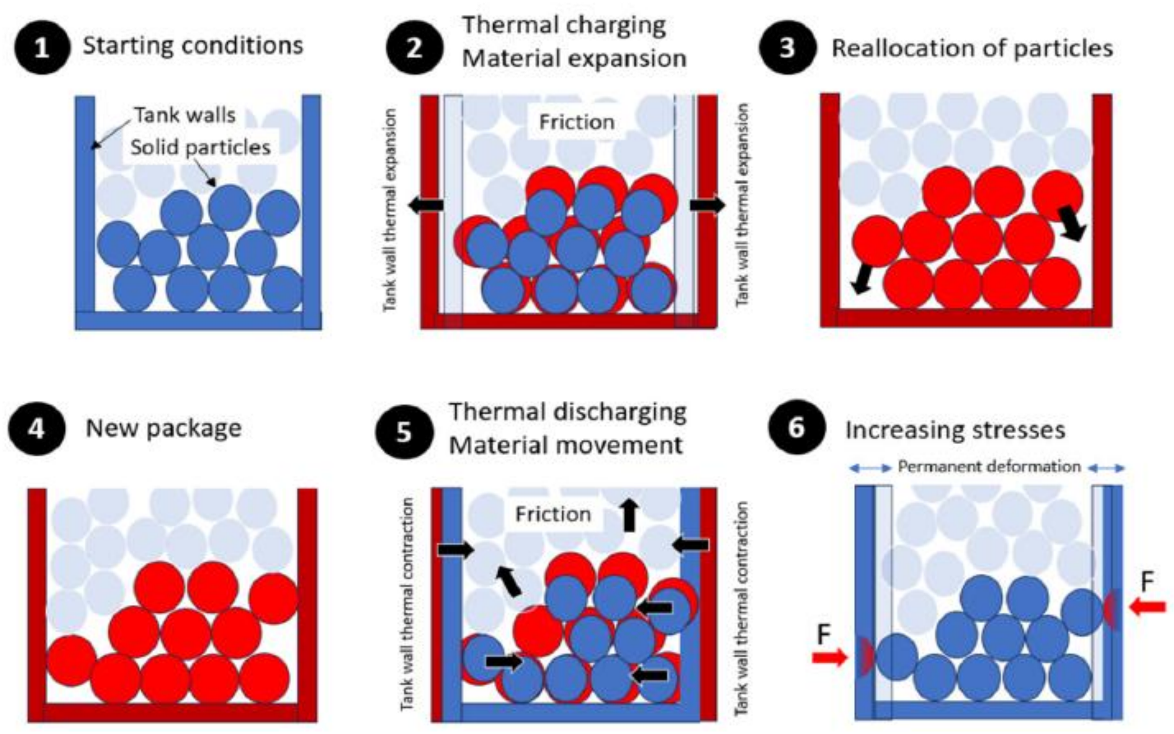


Fig. 1. Description of process that needs to be solved [Garitaonandia et al.]

TASK

We are looking for an entrepreneurial and enthusiastic MSc student with a strong interest in (thermal) modelling, energy systems, and experimental validation. You will continue the work of the current intern, who developed a modelling framework to quantify thermal ratcheting in a packed-bed TES system. In this project, you are expected to:

- Simulate mitigation strategies using existing models.
- Translate promising strategies to the TESSOTOF test setup and support experimental validation.
- Analyze measured data (filler height and diameter changes per cycle) to improve model accuracy.

GOALS

- Develop numerical model and simulations of mitigation strategies (e.g. vertical tubes, layering, different fillers, wall modifications).
- Validating these strategies in the TESSOTOF test setup with height and diameter measurements per cycle.
- Assisting with the full-scale implementation phase (designing and testing the selected solution in a real-life environment).

Fig. 2. Latest system opened in Jerez de la Frontera, Spain

BENEFITS

- Work in a mission-driven start-up which is decarbonizing the world's energy supply.
- Gain hands-on experience with multi-physics modelling and energy storage.
- Collaborate and learn from experienced engineers from Suncom, Fraunhofer, and Dehoust as part of a funded international project.
- Internship compensation of €500 per month.
- Opportunity to join the team long-term after your internship, with the possibility to become co-owner

REFERENCES

Qin, S., Yoo, J., & Morton, T. (2022). Thermal Ratcheting Analysis of TEDS Packed-bed Thermocline Energy Storage Tank – Modeling Methodology and Data Validation. Idaho National Laboratory. https://doi.org/10.2172/1868532

Garitaonandia, E., Arribalzaga, P., Miguel, I., & Bielsa, D. (2024). Characterization of the Ratcheting Effect on the Filler Material of a Steel Slag-Based Thermal Energy Storage. Energies, 17(7), 1515. https://doi.org/10.3390/en17071515