Master Thesis project proposals

Q2 2023-2024

Power and Flow

Department of Mechanical Engineering

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).

Supervisor	Sina Tajfirooz
2nd supervisor	Hans Kuerten
Company supervisor	Sina Tajfirooz
Company	NRG
Starting date	Any time
Exp./Num./Design	Numerical

Power & Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Direct numerical simulation of turbulent stratified two-phase flow

S. Tajfirooz, J.G.M. Kuerten

tajfirooz@nrg.eu , j.g.m.kuerten@tue.nl

INTRODUCTION

*The **Nuclear Research and consultancy Group (NRG)** located in Petten, is one the major institutes responsible for nuclear research in the Netherlands. NRG is internationally recognized as the foremost provider of nuclear medicine, supporting over 30,000 patients annually. A significant focus of NRG's research is dedicated to enhancing nuclear reactor safety by utilizing Computational Fluid Dynamics (CFD) tools.*

*The modeling of **two-phase flows** holds significant importance in nuclear engineering applications. These applications include pressurized thermal shock, emergency core cooling, and boiling in a fuel rod bundle. In most instances, these applications involve turbulent multiphase flows. The characteristics of turbulent flow can vary considerably based on the type of two-phase flow regime present, such as dispersed, bubbly, slug, or stratified regimes.*

*This project focuses on the **stratified two-phase** regime. This regime, characterized by a large-scale interface between the gas and liquid phases, typically occurs in **reactor cooling systems**. The heat and mass transport across the interface is closely connected to the flow field and turbulence in both phases. Hence, accurate prediction of flow field and the coupled mass transfer is highly important for optimizing reactor cooling systems.*

*At NRG, in collaboration with national and international research partners (TU/e, ANL), we use **direct numerical simulation (DNS)** to capture the flow field in both phases in a **co-current turbulent stratified flow** (See Figure 3) .*

GOAL

*This proposal outlines a research direction focusing on the DNS of turbulent stratified two-phase flows using the **volume of fluid (VOF)** method. The open-source solver **Basilisk** will be used for this purpose.*

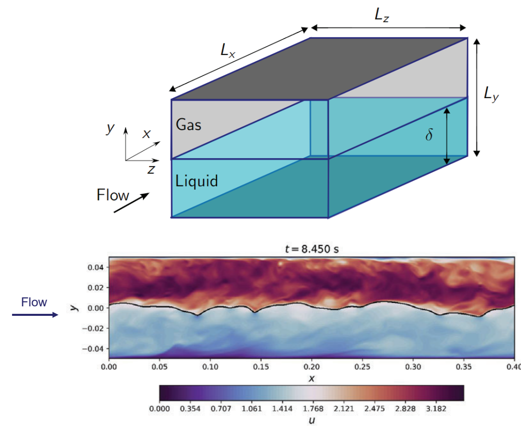


Figure 3: DNS of turbulent stratified flow. (a) schematic of the computational domain. (b) A snapshot of the interface and streamwise flow field at z=0.

*The goal is to set up a final **DNS case for generating benchmark data** that will be used for validation and improvement of lower-resolution models such as Reynolds- averaged Navier Stokes and large-eddy simulation.*

TASKS

- *Implement **averaging tools** to extract spatio-temporal statistics of the interface and flow field (e.g., time and space spectra, two-point correlation)*
- *Implement a **passive scalar transport** model in the current solver to investigate the combined effects of flow and heat/mass transfer.*
- *Perform **preliminary under-resolved simulations** to select appropriate computational settings for the final DNS simulation.*
- Post-process, analyze and report the results.

Prior knowledge of **C** and **python** is a plus.

BENEFITS

- Learn how to simulate two-phase flows using DNS
- Combination of programming and performing simulations for an industrially-relevant phenomenon
- Monthly allowance and compensation for housing and transportation for the period of your stay.

Supervisor	Wenjiang Tian
2nd supervisor	Conrad Hessels
Mentor	Xiaocheng Mi
Company	Internal
Starting date	Q2 2023
Exp./Num./Design	Experimental

P&F

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Study of solid-phase combustion mechanism of iron powder by thermogravimetric analyser (TGA)

Wenjiang Tian, Conrad Hessels, Xiaocheng Mi
E-mail : w.tian2@tue.nl, x.c.mi@tue.nl

INTRODUCTION

Iron powder is a promising clean energy carrier[1], but its current combustion mechanism is not fully understood. This project proposes to investigate the solid phase combustion mechanism by using TGA experiments to pre-oxidise the particles to obtain different degrees and then carrying out material characterization. This will help to improve the combustion performance of iron powders as well as design new burners.

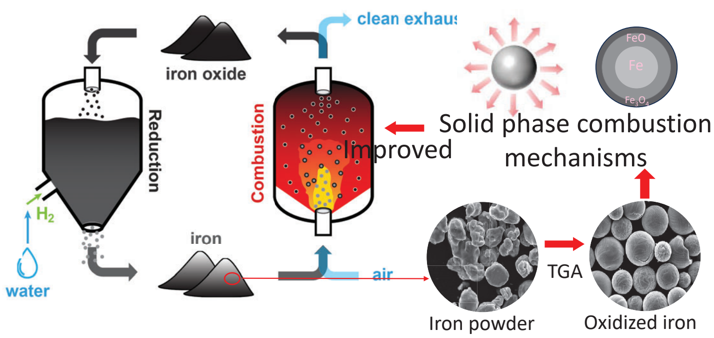


Figure 1. The implications of the TGA experiment

TASKS

The thermogravimetric analyser (TGA) is used to obtain iron powder particles with different oxidation levels by controlling the oxygen concentration, oxidation time, etc., and to determine the oxidation stage by calculating the oxygen supply.

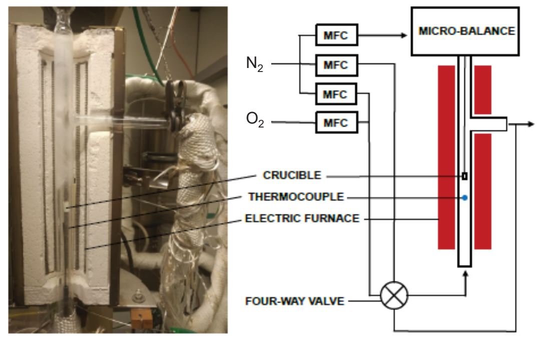


Figure 2. Diagram of the thermogravimetric experimental setup[2]

Material characterization experiments will be carried out on the collection of iron powders with different degrees of oxidation to study the evolution of their structure and composition.

GOALS

Elucidating the solid-phase combustion mechanism of iron powders.

STUDENT PROFILE

We are looking for a MSc student who has:

- Interested in experiments and metal fuel
- Hands-on attitude toward new challenges
- Analytical capacity

REFERENCES

[1] Bergthorson, J. M. Recyclable metal fuels for clean and compact zero-carbon power. Progress in Energy and Combustion, 68,(2018) 169-196.

[2] C.J.M. Hessels et al. Reduction kinetics of combusted iron using hydrogen. Powder Technology, 407 (2022) 117540.

Supervisor	Victor Habiyaremye
2nd supervisor	Hans Kuerten
Company supervisor	Victor Habiyaremye
Company	NRG
Starting date	Any time
Exp./Num./Design	Numerical

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Investigation and validation of immersed boundary methods for two-phase direct numerical simulations

V. Habiyaremye, J.G.M. Kuerten

habiyaremye@nrg.eu; J.G.M.Kuerten@tue.nl

INTRODUCTION

The Nuclear Research and Consultancy Group (NRG) is one of the main institutes involved in nuclear research in the Netherlands. In order to ensure nuclear reactor safety, NRG uses Computational Fluid Dynamics (CFD) for accurate predictions of the thermo-hydraulic behavior of the reactor.

Two-phase flow can occur in nuclear reactors, both in normal operation as well as in accident scenarios. It is therefore important to gain a better understanding of how such flows develop and evolve, and how they may affect nuclear reactor safety and performance. In order to achieve this goal, NRG is developing an in-house Direct Numerical Simulation (DNS) solver capable of efficiently simulating two-phase flows. This solver can make use of an Immersed Boundary Method (IBM) to simulate flows in complex geometries without complicating the numerical mesh.

GOALS

In this graduation project, the focus will be on further developing and validating the IBM in the in-house solver. First, this will be done in single-phase flow, and then, the interaction of two-phase flow with the IBM will be studied. The end goal is to have a robust and well-validated IBM implementation, as well as a first application of the IBM to a two-phase flow problem. The project’s location is at the NRG site in Petten.

NRG Nuclear. For life.

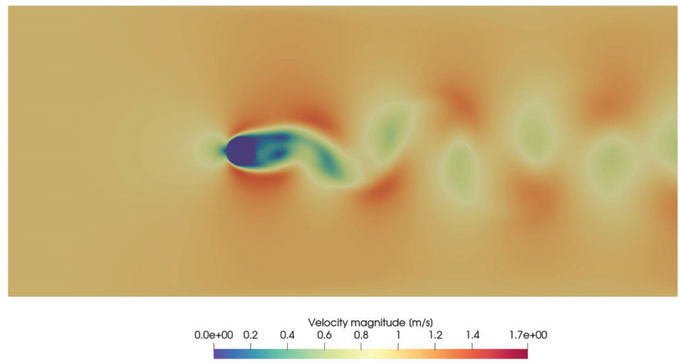


Figure 1. Vortex street behind a cylinder at Re=100, simulated using an IBM [1]

TASKS

- Review existing literature on IBMs to better understand the approach and identify validation cases
- Become familiar with the CFD code by running basic test cases
- Implement sampling and averaging tools for validation (for this, prior knowledge in C++ and/or OpenFOAM is a plus)
- Validate different IBM options in well-known flow configurations (e.g., see Figure 1).

BENEFITS

- Learn how to program in a CFD code
- Combination of writing code and running simulations
- Monthly allowance and housing/public transport compensation

REFERENCES

[1] Fadlun, E. A., Verzicco, R., Orlandi, P., & Mohd-Yusof, J. (2000). Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *Journal of computational physics*, *161*(1), 35-60.

Supervisor	Tess Homan
2nd supervisor	Nico Dam, Hanneke Gelderblom
Company	N.A.
Internal / External	Internal
Starting date	Now available
Exp./Num./Design	Experimental and data analysis

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Virus-laden droplets: characterization

TESS HOMAN, Nico Dam, Hanneke Gelderblom
EMAIL: T.A.M.HOMAN@TUE.NL

Introduction

During the COVID-19 pandemic we all had to adhere to the 1.5 meter rule. But where does this advice come from?
For any type of virus it is important to know how it spreads. The big question about the coronavirus was: Does it spread through the air, i.e. is it airborne? You would think this is easy to measure, but still the proof is very limited. And if it is airborne, what type of measures should we take: masks, ventilation, keeping distance? This depends on the size and behaviour of the droplets that carry the virus. And when these droplets reach a new person, will the virus still be infectious?

In a collaboration with Erasmus Medical Center we aim to answer these questions. At the TU/e we will build a test setup to make and collect droplets containing virus-like particles. The entire process needs to be gentle in order to not disintegrate the virus.

Build a spray device

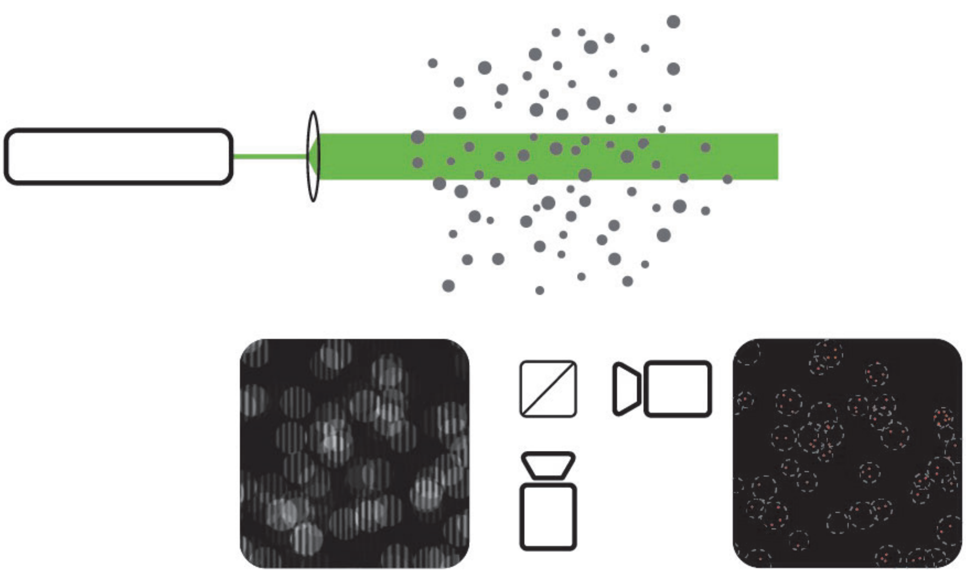
In this project you will build a device to create droplets in a gentle way. The resulting spray needs to be characterized: What is the droplet size distribution? How is the virus distributed over the droplets? Using optical techniques we will simultaneously measure the droplet sizes and the distribution of fluorescent virus-like particles.

Objectives

- Create a gentle spray of a visco-elastic (mucus-like) liquid
- Use IPI (Interferometric Particle Imaging) to measure the droplet sizes
- Use fluorescence to track the virus-like particles

You

- will be working in a lab
- will build a test setup
- will do experiments using optical techniques
- will use data analysis to interpret the data



To characterize the spray, we will use a thick laser sheet and a 2-camera setup that measures both the particle-size and the distribution of fluorescent nanoparticles. IPI image adapted from Evans *et al*. Rev. Sci. Instrum. **86**, 023709 (2015)

Supervisor	Jesse Hameete
2nd supervisor	XiaoCheng Mi
Company supervisor	
Company	TUE
Starting date	-
Exp./Num./Design	Exp/Design

Power &Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Deposition behaviour of iron particles in a lab scale combustion system

Jesse Hameete, XiaoCheng Mi
j.Hameete@tue.nl

INTRODUCTION

Iron can be used as an energy carrier to store and transport green hydrogen in a safe, cheap and recyclable way (Figure 1). In commercial combustion systems, particles that deposit on the wall and form large agglomerates may hinder efficient combustion and prevent recycling of the powder. Earlier research by a master student showed that welding, the effect where a liquid particle melts the reactor wall locally and fuses with the surface, is the most likely cause of this phenomenon. An example of his work, that is now being prepared for publication, is shown in Figure 2.

TASKS

Continuing the research by designing a surface of which we can accurately measure and control the temperature. Experiments should be performed to test the hypothesis of welding by actively controlling the plate temperature and measuring the deposition amount at different temperatures.

GOALS

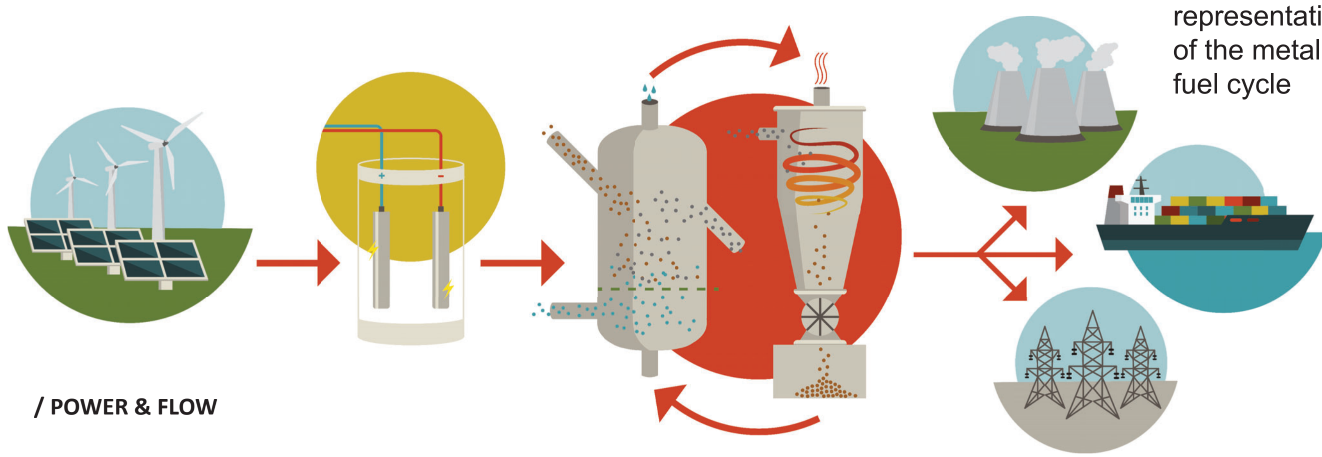
The main question that will be answered in this project will be: Can the deposition of metal particles on a reactor surface be controlled by cooling the surface?

Figure 2: An experiment performed by the previous master student. Iron particles are combusted before colliding with a metal plate to simulate deposition on a reactor surface.

BENEFITS

This question is incredibly valuable for the industry, and you get to work on a lab scale iron burner on a project with well-defined boundary conditions and a good chance on a publication!

Figure 1: A graphic representation of the metal fuel cycle



Supervisor	dr.ir. Noud Maes
2nd supervisor	dr.ir. Michel Cuijpers, Zhongcheng Sun (MSc), Benjamin Haefele (Shell)
Company	Shell
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Experimental

Power & Flow

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Ultra-clean engines using sustainable bio-fuel blends

Noud Maes, Michel Cuijpers, Zhongcheng Sun, Benjamin Haefele

Introduction

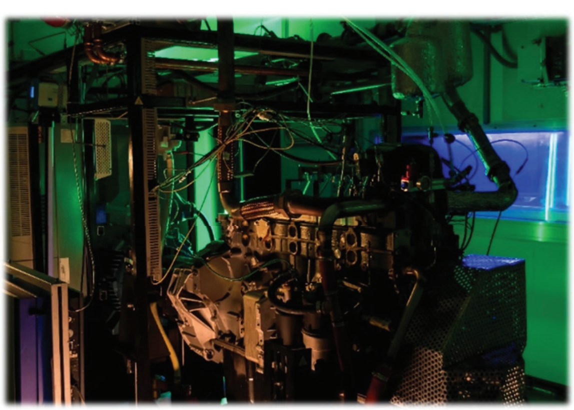
The energy transition from fossil- to bio-fuels (and eventually E-fuels produced by clean oversupply energy) will not be an overnight transition. While a future energy surplus from sustainable sources is expected, the availability of E-fuels is still limited to small batches. On the shorter term, the share of bio-fuels needs to be investigated to facilitate a faster decarbonization rate. In this work, particular focus will be given to different bio-fuel blends which have a lower CO_2 impact when compared to the standard EN590 B7 fuel that passenger cars, light-duty transport, but especially heavy-duty trucks currently use. In initial tests, however, some of these fuels showed opposite effects on other emissions such as soot and NO_x. Using a design-of-experiments (DOE) approach, the optimum between blend ratios for reduced global emissions (CO_2), and local emissions (soot and NO_x) should be determined.

Project description

1. Investigate properties of different bio-components
2. Determine combustion properties of selected fuels
3. Optimize fuel blends for local and global emissions

Contact

Noud Maes, n.c.j.maes@tue.nl



Photograph of the MX13 test engine

/ POWER AND FLOW

Experimental apparatus

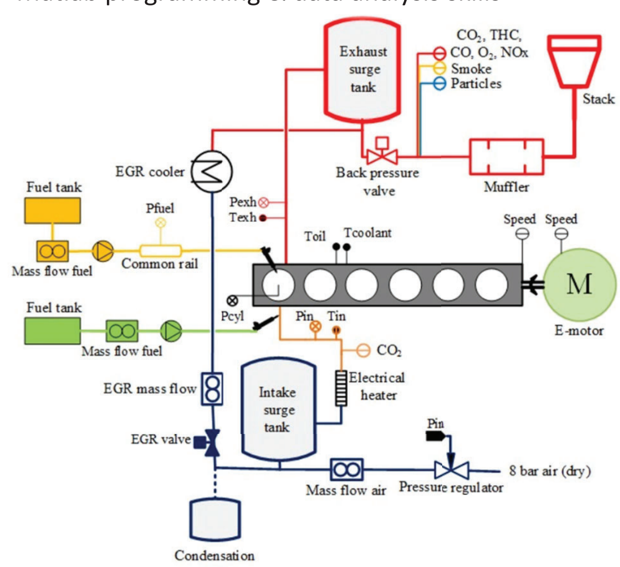
The goal of this project is to conduct experiments on a dedicated single-cylinder research engine. This engine is equipped with a state-of-the-art fuel-flexible injection system, pressure- & temperature sensors, and advanced emission analysis systems. The power generated by the engine is delivered back to the electricity grid via an electric motor. This motor is used both as a dynamometer and to motor the test cylinder when no fuel is injected.

Tasks

- Literature study, getting familiar with the fuels, engines, combustion properties, and analytic methods
- Involvement with preparing the engine setup
- Benchmark engine testing using conventional fuels
- Benchmark engine testing using different blends
- Analysis of the data to write a concise report

Requirements

- Affinity with experimental work & combustion engines
- Background knowledge on internal combustion engines
- Motivation and a hands-on mentality
- Matlab programming & data analysis skills



Schematic layout of the MX13 test engine

Supervisor	Noud Maes
2nd supervisor	Max Peters
Company	N.A.
Internal / External	Internal
Starting date	2023/2024
Exp./Num./Design	Experimental Research

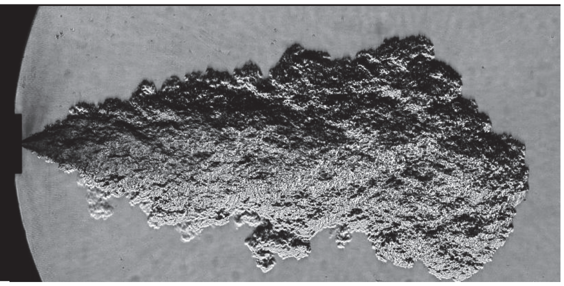
Power & Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Measuring needle lift and massflow of a DI hydrogen injector for ICE use

M.E.E. Peters* & N.C.J. Maes

*E-mail: m.e.e.peters@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, seem viable for the internal combustion engines of the future.

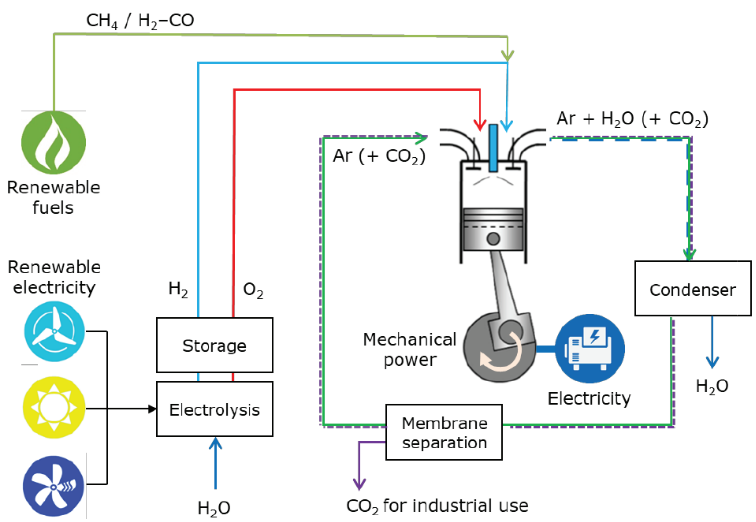


Figure 1: Schematic layout of the Argon Power Cycle.

In order 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 in Berkeley (CA, USA), the characterization of a modified Gasoline Direct Injection (GDI) injector needs to be performed in order to gain a better understanding of the mixing of hydrogen in a pressurized environment.

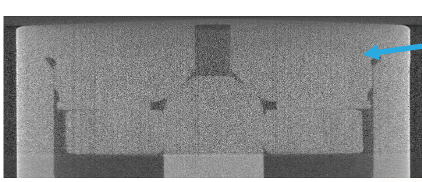


Figure 2: CT-scan showing the insides of the H2 injector. Note the bullet shaped needle closing of the exit hole in the middle.

/ POWER AND FLOW

Objective

To better understand the opening phase of the hydrogen injector, laser doppler vibrometer measurements on the needle lift need to be performed and compared to earlier high-speed optical measurements. The needle opens very fast over a short distance (<0,1 ms, 75 μm), which introduces various challenges.

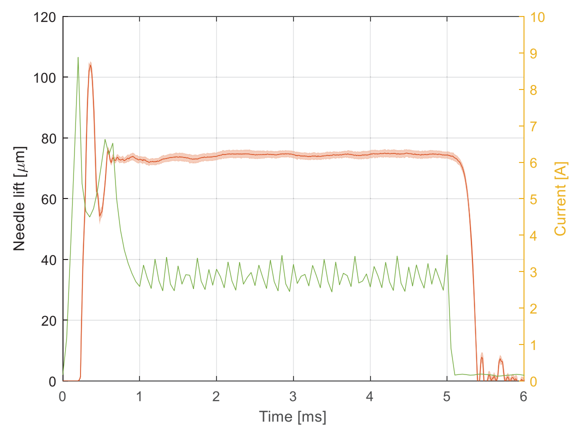


Figure 3: High-speed (100 kHz) needle lift results of the modified GDI injector versus electric signal.

Approach

You will:

- Study literature and get familiar with various solenoid fuel injectors and earlier performed research.
- Learn to use a laser doppler vibrometer (Polytec PB OFV-5000).
- Validate (earlier) optical high-speed experiments.
- Perform new measurements with modified needle lift and nozzle geometry.
- Explore the possibility to measure mass and momentum flow of the injector (Cm & Cd).
- (optional) Measure needle lift at high fuel pressure conditions.
- Process and analyze the measured data using Matlab routines.

Recommended courses:

- *Future fuels and clean engines (4AT020)*
- *Experimentation for Mechanical Engineering (4BM20)*
- *Optional: Optical diagnostics for combustion and fuel flow (4BM40)*

Supervisor	Yali Tang
2nd supervisor	Jan Hendrik Cloete (@SINTEF)
Company	SINTEF, Norway
Internal / External	External
Starting date	Any time
Exp./Num./Design	Numerical

Power and Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Wall-particle heat transfer modelling in Two-Fluid Simulations

YALI TANG, JAN HENDRIK CLOETE
EMAIL: Y.TANG2@TUE.NL

INTRODUCTION

Heat transfer between the particle bed and heat exchanger walls plays an important role in numerous fluidized bed reactor concepts that are proposed for more sustainable processes, such as adsorption-based CO_2 capture [1]. Somewhat surprisingly then, there are no suitable models to account for particle-wall heat transfer in the most commonly used computational fluid dynamics (CFD) method for simulating fluidized bed reactors [2], the two-fluid model (TFM). Based on a previous MSc work, this project therefore aims to further develop a recently proposed approach for including particle-wall heat transfer in TFM simulations [3].

Project

The first part of the project will require running TFM simulations in Ansys Fluent for a variety of experimental fluidized bed setups for which particle-wall heat transfer data exist in literature. Data generated from the simulations will then be used to optimize the parameters in a particle-wall heat transfer closure to achieve the best possible match with experimental data. An existing Matlab script will be upgraded and utilised for this purpose. The student will also be expected to improve the formulation of an existing particle-wall heat transfer closure based on physical arguments. This study will be performed in collaboration with researchers from SINTEF Industry, a non-profit research institute based in Norway, and the student will spend part of the project time in Trondheim, Norway. There are possible funding to support the staying of the student in Trondheim.

OBJECTIVES

- To further develop a recently proposed wall-particle heat transfer model
- To improve the model formulation based on physical arguments

APPROACH

- Perform TFM simulations using Ansys Fluent
- Optimize model parameters using Matlab
- Analyse and discuss the results

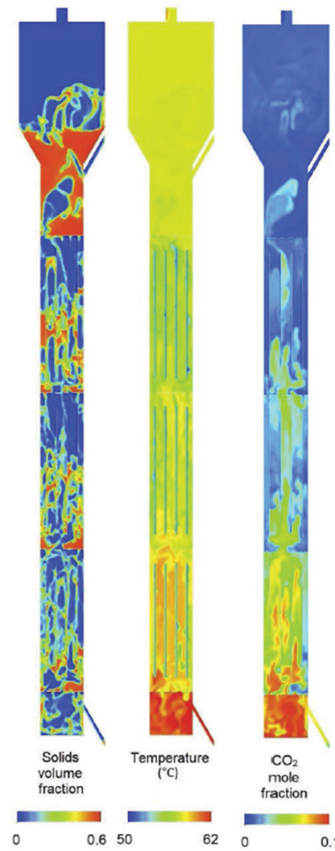


Figure 1: An example of a TFM simulation of a fluidized bed CO_2 adsorption reactor with particle-wall heat transfer included

REFERENCES

1. Dhoke C. et al. Review on Reactor Configurations for Adsorption-Based CO_2 Capture. Industrial & Engineering Chemistry Research. 2021
2. Alobaid F, et al. Progress in CFD Simulations of Fluidized Beds for Chemical and Energy Process Engineering. Progress in Energy and Combustion Science. 2021
3. Cloete J.H, et al. Developing a novel approach for modelling particle-wall heat transfer in fluidized bed reactors for CO_2 capture. In: 24th Fluidized Bed Conversion conference; 2022. Available from: bit.ly/3aaO4t7

Supervisor	Giulia Finotello
2nd supervisor	Dennis Thuy
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Numerical

Power & Flow

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Simulation of breakup and solidification of liquid metal droplets in gas atomization

Dennis Thuy*, Giulia Finotello, Joris Remmers, Niels Deen
*E-mail: d.p.l.thuy@tue.nl

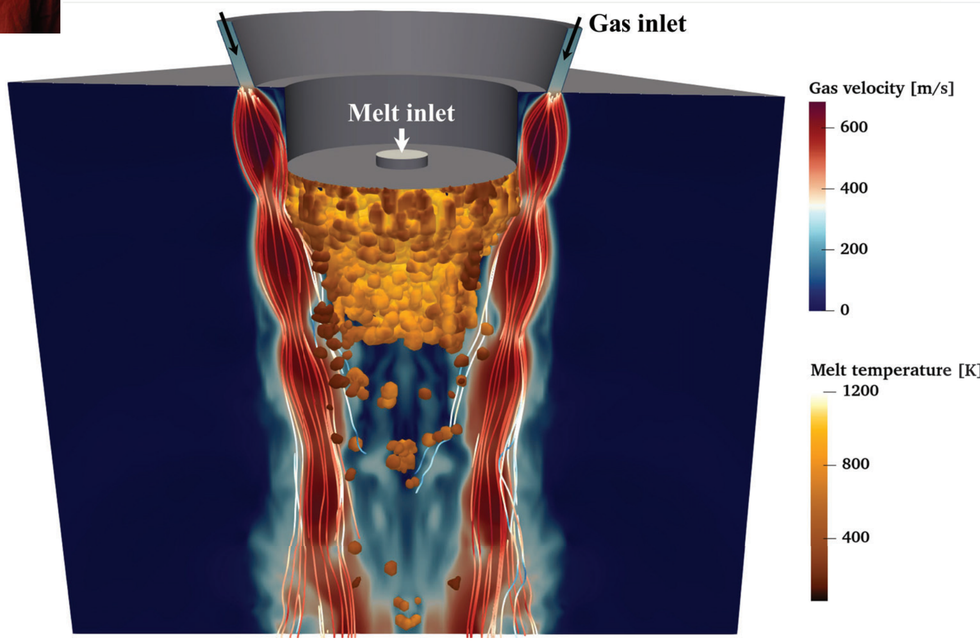


Figure 1: Simulation in OpenFOAM of the breakup of liquid metal in a gas atomizer

INTRODUCTION

Metal 3D printers can be part of a completely circular production environment where scrap metals can be used as the raw material to produce powder for the printers. While metal 3D printers are already in an advanced state, the small-scale production of metal powder is still in need of development.

In a gas atomizer, metal powder is produced by breakup of a liquid metal jet under the influence of high-pressure gas flow, as shown in figure 1. While the metal droplets are breaking up, they also cool down rapidly and solidify in-flight.

After solidification, the metal particles can no longer break up. Therefore, the solidification process is very important for the final properties of the powder. It needs to be accurately modeled in simulations of liquid metal atomization.

TASKS

The project revolves around CFD simulations in which the breakup and solidification of the liquid metal droplets are modelled.

The starting point is the work of a previous master student, who tested breakup models and implemented a simple solidification model in one of the Lagrangian solvers of the CFD software OpenFOAM.

Development of an efficient code that combines the breakup and solidification model for the droplets in the existing solver will be the focus of the project. Of course, it is possible to tune the project to your own interests!

GOAL & BENEFITS

The goal of the project is to develop a numerical model that can show the influence of solidification and breakup on the properties of gas atomized metal powder. This knowledge will be used to finetune powder properties to the requirements of metal 3D printers.

Besides actively contributing to the optimization of additive manufacturing feedstock, other benefits of this project include the freedom to shape the research to your interests, as well as getting to work with OpenFOAM, one of the most popular open-source CFD packages!

Supervisor	Nico Dam
2nd supervisor	Tess Homan
Company supervisor	n.a.
Company	n.a.
Starting date	Any time
Exp./Num./Design	Exp. / Design

EINDHOVEN UNIVERSITY OF TECHNOLOGY

An acoustic bubble scraper

Nico Dam (n.j.dam@tue.nl)
Tess Homan (t.a.m.homan@tue.nl)

INTRODUCTION

Electrolysis, the splitting of liquid water into gaseous hydrogen and oxygen, is one of the ways to store surplus green electricity for later use. During electrolysis, gas bubbles form on a metal electrode surface, and this gives rise to an efficiency trade-off: the more gas is produced, the more electrodes are covered in bubbles, and the less free metal surface is available for more gas production.

Hydrogen bubble formation on a gold wire. As long as they are attached to the wire, they reduce its effective surface area. (Figure from Chen et al. (2018) JPC C122: 15421.)

Obviously, there is a need for an electrode cleaning procedure. In a previous master project (van den Brink, 2023) we have shown that bubbles can be manipulated by sound. In this project, want to employ sound to scrape bubbles of an electrode surface.

TASKS

- Study the literature on acoustic bubble formation
- Design a simple electrolysis setup combined with an acoustic field generator
- Perform experiments to optimize bubble scraping
- Analyze and present the results

GOALS

- Understand the interaction of sound with bubbles
- Construction of a simple electrolysis setup combined with an acoustic generator
- Assess feasibility of bubble scraping
- Assess prospects for upscaling

BENEFITS

Why choose this project?

- Interesting topic
- Relevant application
- Combination of experimental work, design and analysis
- Lots of room for your own input

Why not choose this project?

- *I hate research*
- *I hate to come up with ideas myself*
- *I hate enthusiastic supervisors*
- *I prefer numerical work*

REFERENCES

(A demonstration of acoustic particle manipulation. The movie concerns solid particles in air, but you will employ the method on gas bubbles in liquid.)

- van den Brink, Y. (2023). Design of an an acoustic bubble levitator (Report of the previous master project.)

Supervisors	Nicole Stevens
2nd supervisor	Giulia Finotello
Company	N.A.
Internal/External	Internal
Starting date	2nd quarter 2024
Exp./Num./Design	Experimental

Power & Flow

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number: 2024 Q2-3

Reduction of iron ore in a lab-scale fluidized bed

Nicole Stevens and Giulia Finotello
EMAIL: n.c.stevens@tue.nl

Introduction

Iron powder is a good energy carrier which can be stored and transported in large amount. Energy is released by burning the powder resulting in solid iron oxides. These iron oxides particles can be recycled using green hydrogen via a reduction process. The products of this reduction process are solid iron and water vapor. The combination between these two steps, combustion and reduction, creates a carbon free cycle called the metal fuel cycle.

Initial experiments in a packed bed (thermogravimetric analysis) showed that full conversion to iron is possible using hydrogen, creating porous particles. However, the reaction rate was rather slow and sticking was found to occur. A lab-scale fluidized bed set-up has been designed and built to prevent this by allowing better mixing and heat transfer of the particles.

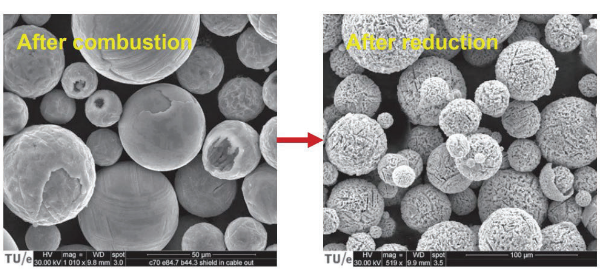


Figure 1: SEM images of combusted iron (left) and direct reduced iron (right) [1]

Project

Reduction experiments at elevated temperatures are executed in the fluidized bed using a hydrogen flow. The experiments can be conducted in two different regimes: bubbly regime and turbulent regime. Sticking behavior during reduction at different conditions can be studied. The particle size distribution of the powder can be analyzed before and after reduction experiments.

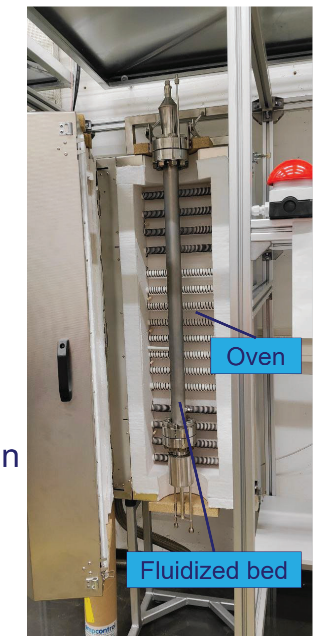


Figure 2: lab-scale fluidized bed

Goals

This project consists of two main goals:

1. Optimize the reduction process.
2. Investigate sticking behavior in both regimes

Tasks

- Study the theory on iron oxide reduction and sticking behavior.
- Investigate sticking behavior in two different regimes and possibly with different conditions (temperature, gas flow, H/D ratio) for reduction in the fluidized bed.
- Assess the powder in terms of size, morphology, porosity, composition and reduction degree.

/POWER AND FLOW

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

Supervisor	Prof Philip de Goey
2nd supervisor	Stijn van Aken
Company supervisor	
Company	Metalot
Starting date	Q3 2023-2024
Exp./Num./Design	Experimental

Power &Flow

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Large-scale cyclicity of iron powder as zero-carbon energy carrier

l.p.h.d.goey@tue.nl
s.a.j.v.aken@tue.nl

INTRODUCTION

Metal fuels can be used to store and transport renewable energy (see fig. 1). By combusting iron powder, heat is released that can be utilized in industrial processes.

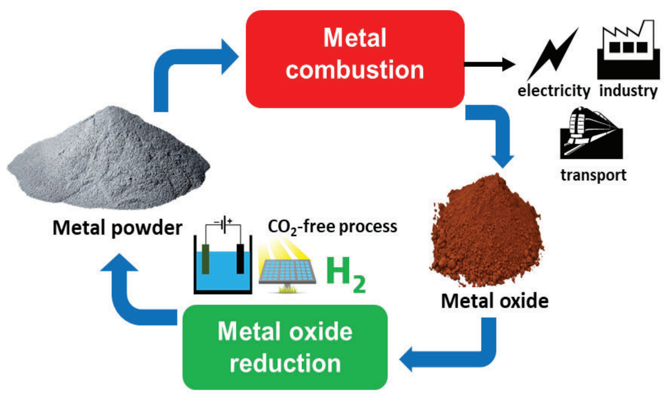


Fig. 1. A Recyclable metal fuels cycle

One of the key features of the metal fuels cycle, is the cyclicity of the iron powder. The cyclicity has been tested for a few cycles on small scale in a lab environment (grams). Currently, Metalot is working together with several partners to set up an experimental program to test the cyclicity of iron powder for many cycles and at large-scale (tons).

Keywords: *Iron Power, metal fuels, cyclicity, large-scale experiments*

TASKS

We are looking for a highly-motivated master graduation student who is interested in the energy transition, process engineering and metallurgy. In this project, you are expected to:

- Conduct experiments using the 500 kW iron powder combustion system (Fig. 2) and the 25 kW hydrogen based powder regeneration setup
- Analyse powder characteristics throughout repeated cyclicity testing

Fig. 2. 500 kW iron powder combustion system operated at Metalot

GOALS

- Determine the cyclicity of iron powder as zero-carbon energy carrier
- Determine the effect of repeated combustion and regeneration on iron powder characteristics

BENEFITS

- Experience to work in the company with experienced researchers on a completely new topic
- You will be working on multi-disciplinary approach (Mechanical Engineering & Chemical Engineering)
- Involved in the development phase of a promising method for green energy storage.

REFERENCES

- https://www.metalot.nl/projecten/
- https://ironfuel.nl/

Supervisor	Helen Prime & Jesse Hameete
2nd supervisor	Yuriy Shoshyn
Company	N/A
Internal / External	Internal
Starting date	Anytime
Exp./Num./Design	Experimental

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Help develop carbon-free energy storage by exploring metal powder combustion

Helen Prime
h.e.prime@tue.nl

INTRODUCTION

Global warming, driven by hydrocarbons, is a well-known problem. We need a carbon free, recyclable energy storage. Metal fuels are proposed as a solution, as they can be used to store energy and help with daily and seasonal energy fluctuations. We need to understand the combustion of metal powder to optimize the process and make the energy transition quicker and easier.

A Bunsen burner was built in McGill, Canada, specifically designed for metal powder combustion. This burner has been optimised for aluminium combustion, shown in Figure 1 for multiple mass flows. We wish to adapt this burner for iron combustion.

OUTLINE

The outline of this project will be the following:

- Set up, calibrate and test metal powder combustion using a Bunsen burner.
- Calibrate using aluminium powder and compare with results from Canada.
- Burn iron powder to measure various parameters and compare with other burners

You have a well-defined project plan, but there is a lot of freedom to make your own planning.

If this sounds like something you want to do for your graduation project, send me a message!

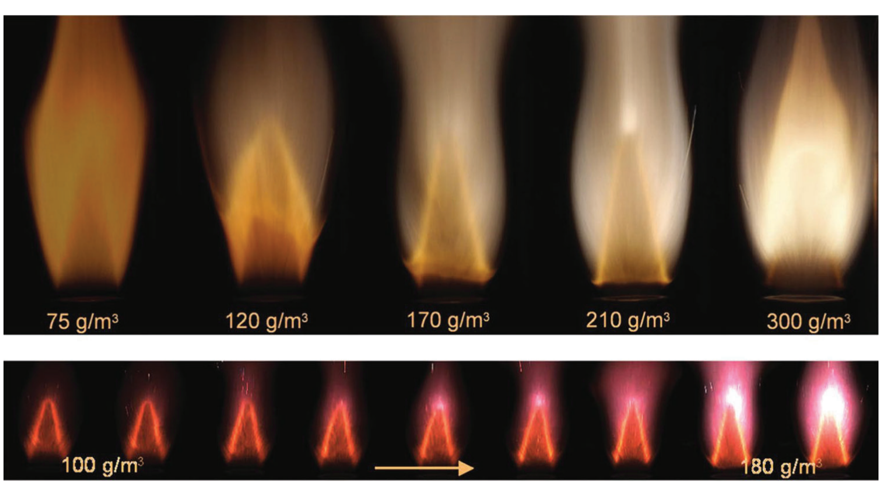


Image 1: Aluminum flame stabilized on a Bunsen burner for multiple mass flows.

Supervisor	Max Peters
2nd/3rd supervisor	Noud Maes, Nico Dam
Company	N.A.
Internal / External	Internal
Starting date	2023/2024 Q2-4
Exp./Num./Design	Experimental Research

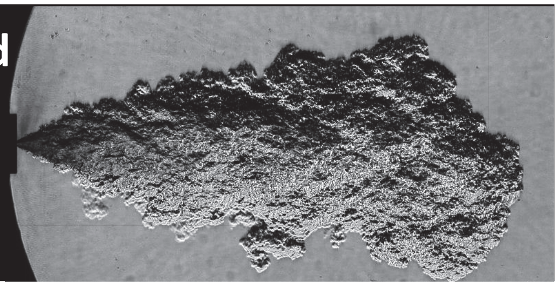
Power & Flow

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

Measuring the Temperature field of a DI (H_2) injector for ICE use using Raman scattering.

M.E.E. Peters*, N.C.J. Maes, & N.J. Dam

*E-mail: m.e.e.peters@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.

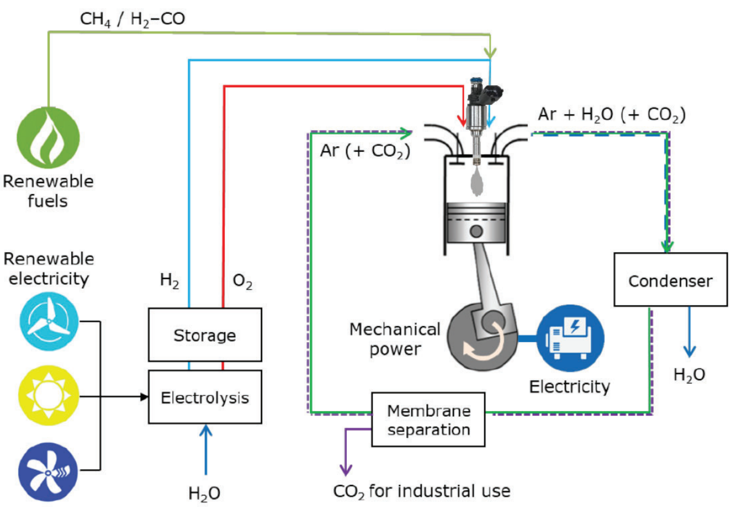


Figure 1: Schematic layout of the Argon Power Cycle.

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.

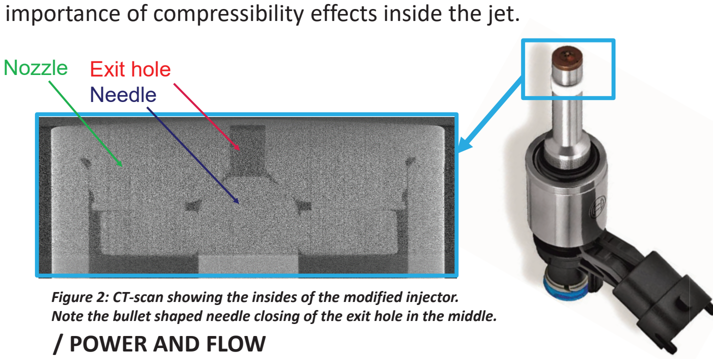


Figure 2: CT-scan showing the insides of the modified injector. Note the bullet shaped needle closing of the exit hole in the middle.

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.

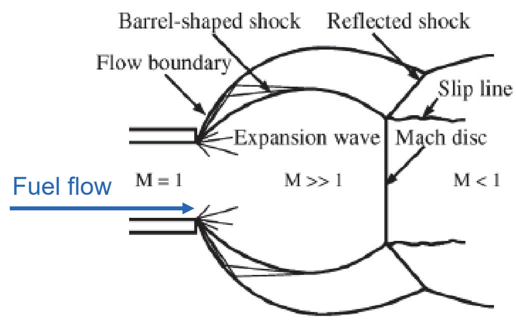


Figure 3: Classical model by Ewan and Moodie[1] on the injection of a compressible fluid.

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])
- HPVB use with hydrogen needs to be explored, single-shot high-pressure setup is also available. (ELPC)
- Validate/compare thermocouple measurements from IFPen.
- Process and analyze the measured data using (existing) Matlab routines.

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]

Supervisor	Marie-Aline Van Ende
2nd supervisor	Niels Deen
Mentor	Nishant Kumar
Company	-
Starting date	Anytime
Exp./Num./Design	Numerical

EINDHOVEN UNIVERSITY OF TECHNOLOGY

CFD Simulation of the pressure drop across a packed bed furnace for the Direct Reduction of Iron

Nishant Kumar, Marie-Aline Van Ende*

***E-mail: m.a.p.p.van.ende@tue.nl**

INTRODUCTION

The direct reduction (DR) of iron oxide, usually followed by electric arc steelmaking, is an alternative route to the conventional blast furnace-basic oxygen route and aims to reduce CO_2 emissions in the iron and steel industry.

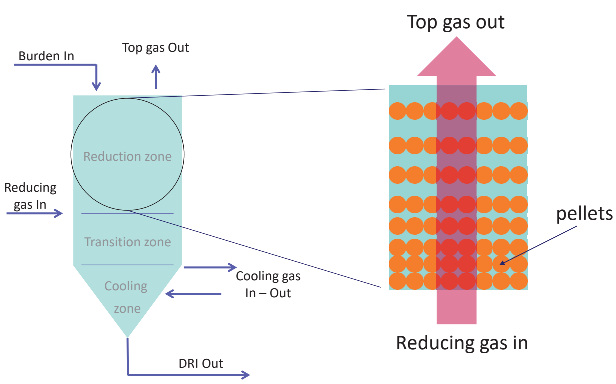


Fig. 1. Simple illustration of the reduction zone in the Midrex process.

The Midrex process is a shaft-type DR process where iron ore, charged atop and descending through a vertical shaft, is reduced to metallic iron by means of an ascending reducing gas (CO, H_2, CH_4). The reduction zone (Fig. 1), where most of the iron oxide reduction takes place, is located between the entry point of the reducing gases and the pellet charging point. The heating of pellets also takes place in this zone due to heat exchange with the hot reducing gas.

The reduction zone resembles a typical packed bed furnace, where hematite pellets are loaded and acts as a packing material. This packing material causes friction in the gas flow, generating friction losses.

Therefore, to ensure adequate pressure drop for the gas flowing through the column at a specified flow rate[1], the relation describing the pressure loss across the length of the packed bed due to the packing material must be determined.

Keywords: *Midrex, Direct Reduction of Iron, Packed column/bed*

OBJECTIVE

Develop a relation for the Pressure Drop across tube length in a packed bed furnace

PRE-REQUISITE

- Familiarity with any CFD software (such as ANSYS Fluent, Star-CCM+, OpenFoam, etc.)

TASK

- Create CFD simulation setup for the furnace
  - Realize geometry and the boundary conditions
  - Generate the flow path geometry
- Conduct parametric study to determine the influence of various input parameters (inlet gas temperature, inlet gas pressure, packing efficiency of bed, size of pellet, etc.)
- Develop the relation for the pressure drop based on those parameters
- Write master thesis report and present main findings

BENEFITS

- Experience working at the High Temperature Thermo-chemistry lab in Seoul National University, Korea
- Experience in design and analysis of numerical models

REFERENCES

[1] R. S. Subramanian. "Flow through Packed and Fluidized Beds." Department of Chemical and Biomolecular Engineering; Clarkson University: Potsdam, NY, USA (2002).

Seoul National University (SNU) campus is located at the feet of the Gwanaksan mountain, south of Seoul (SNU official Facebook page)

Master Thesis Project Proposals
Q2 2023-2024

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	Prof. David Smeulders
2nd supervisor	Yukai Liu
Daily supervisor	Yukai Liu
Company	N.A
Starting date	Any time
Exp./Num./Design	Primarily experimental

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Experimental study of low frequency acoustic transducers

David Smeulders and Yukai Liu
Contact emails: d.m.j.smeulders@tue.nl
y.liu3@tue.nl

INTRODUCTION

The subsurface of the Earth can be used for hydrogen and CO2 storage, but also to extract geothermal energy (hot water). To accurately predict the potential of the subsurface for such applications, permeability assessment of the formation is essential. Borehole acoustic waves can be used to obtain this information. To perform laboratory measurements on borehole waves, acoustic transducers need be designed and constructed. The project is aimed to design, manufacture and test a low-frequency transducer from combined piezoelectric ceramic plate layers.

Fig. 1. Geothermal energy production in Iceland

TASKS

Task1: Build piezoelectric transducer from available ceramic plates.
Task2: Use existing software to predict transducer properties and resonant frequencies
Task3: Perform borehole wave measurements using the new and existing transducers.

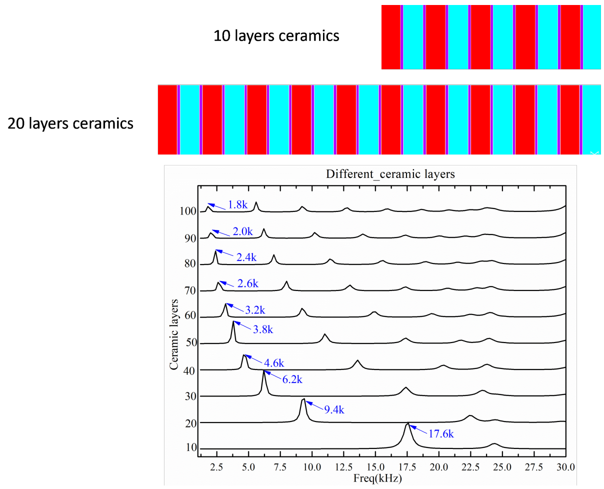


Fig. 2. Schematic of piezoelectric ceramics with 10 and 20 layers and modelling results.

RESEARCH GOALS

1. Determine how the number of piezoelectric ceramic layers affects the resonance frequency.
2. Compare experiment and theory .
3. Measure the waves of the newly constructed transducers in an existing borehole sample.

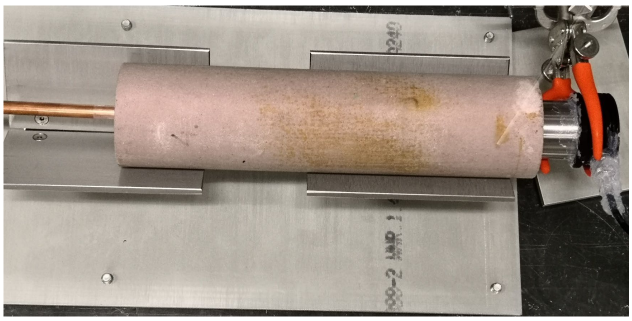


Fig. 3. Photo of borehole wave measurement setup

STUDENT PROFILE

• Interested in experimental studies, especially acoustic or electromagnetic experiments.
• Pro-active attitude.

Supervisor	Dr. Azahara Luna-Triguero
2nd supervisor	Dr. Maja Rucker
Daily supervisor	Mohammad Hossein Khoeini
Company	Internal
Starting date	Anytime
Exp./Num./Design	Numerical and experimental

ETFD

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Advancing analytics for subsurface hydrogen storage: Estimation of the adsorption energy distribution of the solid surface by inverse gas chromatography

Mohammad Hossein Khoeini*, Azahara Luna-Triguero, Maja Rucker

*E-mail: m.h.khoeini@tue.nl

INTRODUCTION

Hydrogen is taking increasingly important role in the energy transition as it has the potential to replace fossil fuels. Hydrogen has higher energy per mass but lower energy per volume than natural gas. Therefore, to meet the energy demand of the industry and society, bulk energy storage systems are required to store produced hydrogen from renewable sources. In this context, underground geological structures, e.g. depleted gas fields, are being considered as a potential option due to their natural large capacity and lower costs. However, underground hydrogen storage (UHS) is still at low level of technological maturity, and there are still several key components of hydrogen storage processes that needs to be fully understood.

One of these poorly understood processes is that during the injection and retraction of hydrogen in the subsurface porous media, hydrogen displaces the existing fluid, e.g., brine, which leads to the complex multiphase fluid flow (figure 1). Two main parameters governing the multiphase flow are surface area and surface energy of the porous rock. These two inherent physiochemical properties determine the extend and strength of the fluids' interactions with porous media and are key to computational models aiming to predict the behaviour of hydrogen in the porous reservoir. In this project, you will advance numerical and experimental methodologies based on iGC to obtain the relevant information needed for future models determining the safety and optimal performance of underground hydrogen reservoirs.

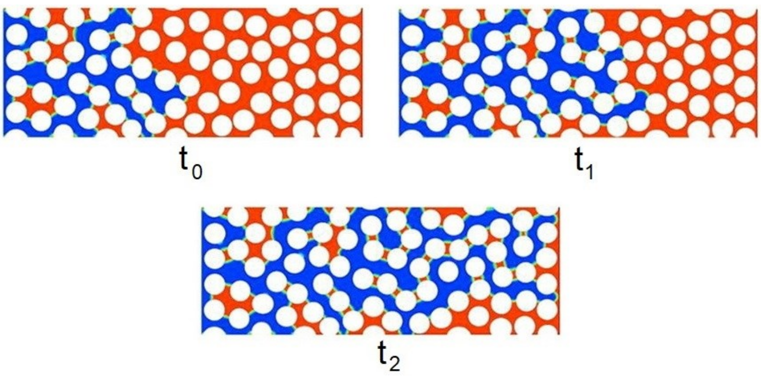


Fig. 1. Schematic representation of existing fluid displacement with hydrogen in porous media (adapted from [2])

Problem statement & Goal

From the theoretical perspective, the surface of a solid is often not composed of interacting sites with a unique energy, but of sites with a range of energy of interaction (figure 2) [2]. In rocks, this is for example due to varying mineral compositions. On the contrary, the surface energy we infer from iGC reflects only the average energy of all interacting sites.

In this study, you will develop an inverse method to obtain the distribution of interacting sites from the average surface energy measured by IGC. Aligned with other projects in our group your work will support steering future decisions on research and implemetation of UHS in NL.

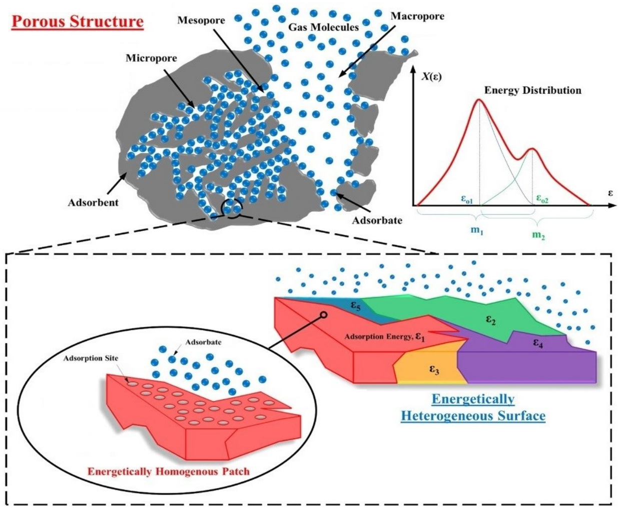


Fig. 2. Adsorption on a surface with different energy sites [3].

TASKS & STUDENT PROFILE

We are looking for the high-motivation student who has creative, solution-oriented, and interest in performing experiments and/or numerical simulations. In this project, you are expected to do:

- Literature review, focusing on different configuration of interacting sites on the surface, and inversion methods of calculating energy distribution functions.
- Developing an inversion numerical method and/or model the method in MATLAB
- Assessing the validity of the developed inverse method by comparing with measured experimental data.

REFERENCES

[1] Aniket S. A., et al. 'Pore-resolved two-phase flow in a pseudo-3D porous medium: Measurements and volume-of-fluid simulations', Chemical Engineering Science, 2021, 230, 116128.

[2] H. Balard, Estimation of the Surface Energetic Heterogeneity of a Solid by Inverse Gas Chromatography, Langmuir 1997 13 (5), 1260-1269.

[3] Ng, K.C., Burhan, M., Shahzad, M.W. et al. A Universal Isotherm Model to to Capture Adsorption Uptake and Energy Distribution of Porous Heterogeneous Surface. Sci Rep 7, 10634 (2017).

Supervisor	Prof. Harald van Brummelen
2nd supervisor	Dr. Stein Stoter
Mentor	Tom van Sluijs
Company	N.A.
Internal / External	Internal
Starting date	Anytime
Exp./Num./Design	Experimental and numerical

Available for ME

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Project number:

Theoretical and numerical investigation of outflow conditions in a multiphase model

Tom van Sluijs*, Stein Stoter, Harald van Brummelen

*E-mail: t.b.v.sluijs@tue.nl

INTRODUCTION

The Cahn-Hilliard equations describe phase separation, such as water-air systems. The Navier-Stokes equations describe the motion of viscous fluids. Coupling these models gives the opportunity to describe the dynamics of multiphase flow. This Navier-Stokes-Cahn-Hilliard (NSCH) model is implemented numerically and used to simulate multiphase problems. See the figure down below or this video for a typical result.

The outflow conditions for pure species are well understood. However, outflow of the mixed phase that contains a phase transition region is more complex. Advanced outflow conditions need to be developed to model the mixed phase outflow and prevented the unphysical phenomena that occur using the general outflow conditions, see Figure 1.

This goal of this project is to develop and implement the mathematical description of appropriate boundary conditions for an outflowing mixed species.

TASKS

In this project, you are expected to do:

- Literature review, focused on multiphase flow, capillarity, the NSCH model and outflow boundaries
- Program a numerical implementation in our existing framework
- Qualitative and quantitative assessment of the numerical results
- Conclusion and advise on both modelling and the numerical implementation

STUDENT PROFILE

We are looking for a motivated MSc student who:

- is curious about the mathematical modeling of fluid dynamics, in particular multiphase flow, interface phenomena and boundary conditions.
- is interested in theoretical modelling, numerical methods, and programming.
- is looking to develop his/her personal skillset: physical modelling, numerical methods, literature study, programming, and data analysis.

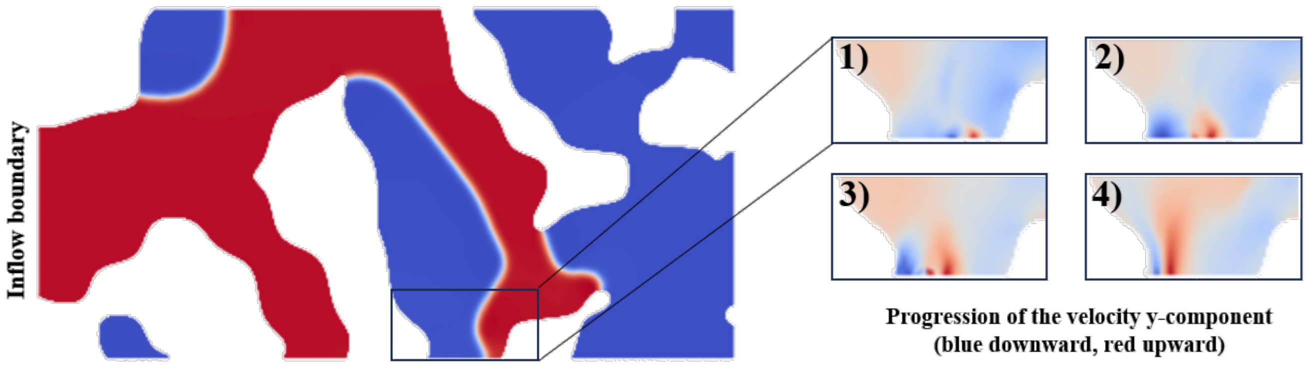


Fig 1.: Multi-phase flow through porous medium. Left: phasefield of two constituents at the time that an outflow boundary is reached. Right: evolution of the velocity field at what should be the outflow boundary. Showing erroneous and unstable velocity inflow (backflow) spikes.

REFERENCES

Abels, H., Garcke, H., & Grün, G. (2012). Thermodynamically consistent, frame indifferent diffuse interface models for incompressible two-phase flows with different densities. Mathematical Models and Methods in Applied Sciences, 22(3). doi: 10.1142/S0218202511500138

Van Brummelen, E. H., Demont, T. H., & van Zwieten, G. J. (2020). An adaptive isogeometric analysis approach to elasto-capillary fluid-solid interaction. International Journal for Numerical Methods in Engineering(March), 1–22. doi: 10.1002/nme.6388

Keywords: Physical modelling, *Numerical methods, Finite Elements, Navier-Stokes, Cahn-Hilliard*

Stein K.F. Stoter, Tom B. van Sluijs, Tristan H.B. Demont, E. Harald van Brummelen, Clemens V. Verhoosel (2023) Stabilized immersed isogeometric analysis for the Navier–Stokes–Cahn–Hilliard equations, with applications to binary-fluid flow through porous media, Computer Methods in Applied Mechanics and Engineering, 2023, 116483, ISSN 0045-7825, https://doi.org/10.1016/j.cma.2023.116483.

Supervisor	Dr. Clemens Verhoosel
2nd supervisor	Dr. Stein Stoter
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

Project number: 2023 Q2-01

Trimmed explicit dynamics: a non-linear Kirchhoff-Love shell model

Clemens Verhoosel, Stein Stoter

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

Keywords: *Explicit dynamics, Trimming, Non-linear Kirchhoff-Love shell, Isogeometric analysis*

INTRODUCTION

Explicit analysis forms the backbone of impact and crash-test simulation software (see Fig. 1). These simulations often involve shell-type components. Trimmed isogeometric analysis streamlines the design-to-analysis pipeline for these types of simulations. In isogeometric analysis, the CAD-based spline geometry representation of the shells is used directly in the analysis software.

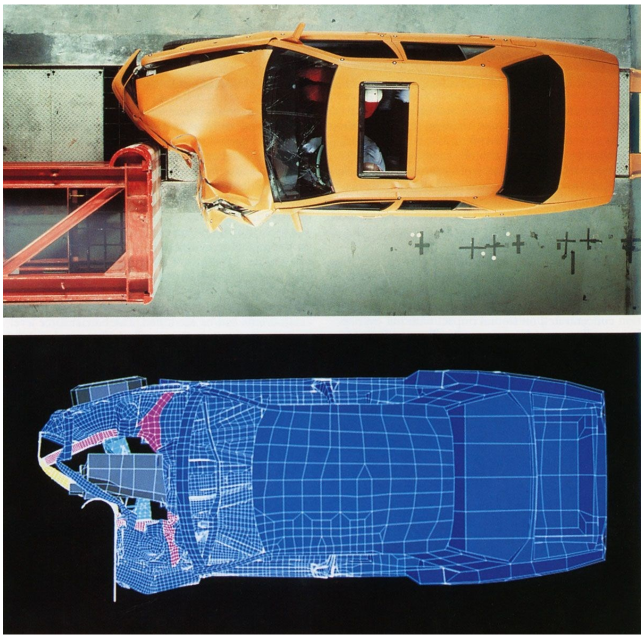


Fig 1: Crash-test simulation. Credit: Cray Research Inc.

PROBLEM STATEMENT

The trimming operation in CAD can lead to elements with very small support. In explicit dynamics, these small cuts may severely limit the permissible time step size. In our group, we have developed methods an analysis procedures for mitigating this adverse effect (see Fig. 2). In this MSc project, you will implement and investigate the performance of this approach for the non-linear variant of the Kirchhoff-Love shell model.

/ ENERGY TECHNOLOGY AND FLUID DYNAMICS

WORK PACKAGE

- Develop a familiarity with shell models and explicit time-stepping methods.
- Extend the existing linear Kirchhoff-Love shell code to a code that can handle the non-linear variant.
- Study the effect of the proposed solution method.
- Depending on the students own learning goals, subsequent research may focus on a shift to the Reissner-Mindlin shell model, or efficient implementation

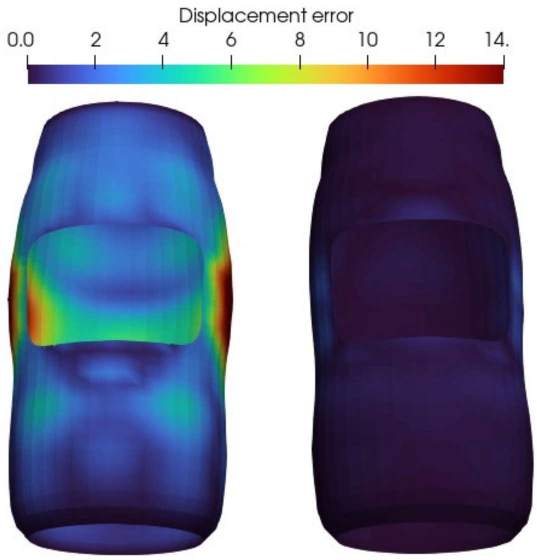


Fig 2: Error in the predicted displacement for the *linear* Kirchhoff-Love shell model, without and with the proposed solution method.

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).

REFERENCES

[1] Stoter, S.K.F. et al. (2022). *Variationally consistent mass scaling for explicit time-integration schemes of lower- and higher-order finite element methods*, Computer Methods for Applied Mechanics and Engineering, 399, 115310.

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

Project number: 2023 Q2-01

Machine learning for scale interaction in advective transport equations

Stein Stoter

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

Keywords: *Machine-learning, Finite element method, Scale interaction*

INTRODUCTION

With the finite element method, we can approximate solutions to partial differential equations. These can be interpreted as 'coarse-scale' representations of the true solution, and the approximation error can be interpreted as the missing 'fine-scale' contribution. For transport equations, the effect of the fine scales must be taken into account (modeled) while computing the coarse scales to obtain stable results (see Fig 1.). The quality of this scale interaction model dictates the quality of the coarse-scale approximation, as seen in the figure below. The ultimate application area of this research lies in the multiscale modeling of turbulent flow (Fig 2.)

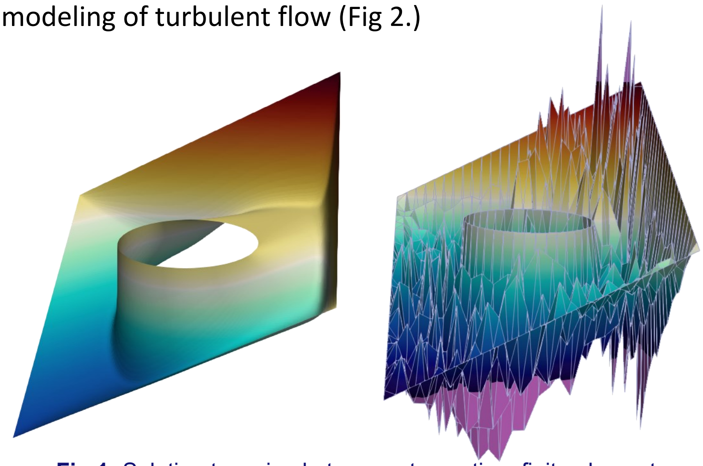


Fig 1: Solution to a simple transport equation. finite element approximation with (**left**) and without (**right**) scale interaction.

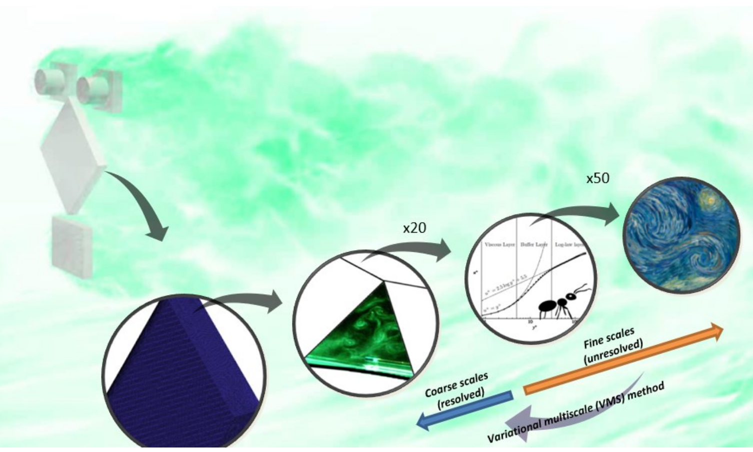


Fig 2: Scale interaction for turbulent flow.

/ POWER & FLOW

PROBLEM STATEMENT

The scale interaction can be computed exactly (Fig 3.), but this costs a lot of computational power, making it unfeasible to do so during the simulation of transport phenomena. Instead, one could learn the scale interaction with a machine learning algorithm during a training phase, and then use the machine learning model during the simulation of the transport problem.

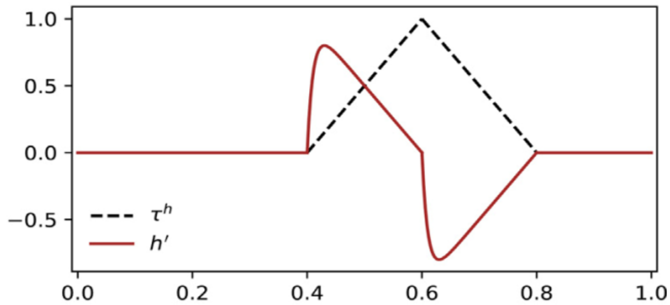


Fig 3: Scale-interaction function for a one-dimensional advective transport equation. For this particular case, the interaction function h' localizes to a single element.

RESEARCH TOPICS

- Perform a literature study to learn about the state-of-the-art of machine learning for scale interaction, and about the types of machine learning techniques used for similar tasks.
- Develop a code that can compute the exact scale interaction function (see one-dimensional example Fig 2.).
- Develop a machine learning code that can predict these functions.
- Study the effectiveness of the machine-learned model of the scale interaction.

STUDENT PROFILE

We are looking for a MSc student who has:

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

REFERENCES

[1] Stoter, S.K.F. et al. (2022). *Discontinuous Galerkin methods through the lens of variational multiscale analysis*, Computer Methods for Applied Mechanics and Engineering, 388, 114220.

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

Project number: 2023 Q2-01

A RANS turbulence modeling framework for means and oscillating flow fields

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

Keywords: *Turbulence modelling, RANS, Fourier representation*

INTRODUCTION

Turbulence modeling frameworks based on the Reynolds-averaged Navier-Stokes (RANS) equations are the main workhorse in CFD-industry turbulent flow solvers. Their popularity stems from computational efficiency and ease of use. Yet they lack in predictive capability, i.e., accuracy. This is due to the underlying models, which necessarily replace the unknown turbulence quantities with expressions based on mean quantities.

Project description

In this project, you will develop a new RANS framework, that not only aims to retrieve the mean quantities, but also the lower-order oscillations of the turbulent flow field. The expectation is that this fuller description of the complete physics alleviates the model deficiencies of classical RANS models.

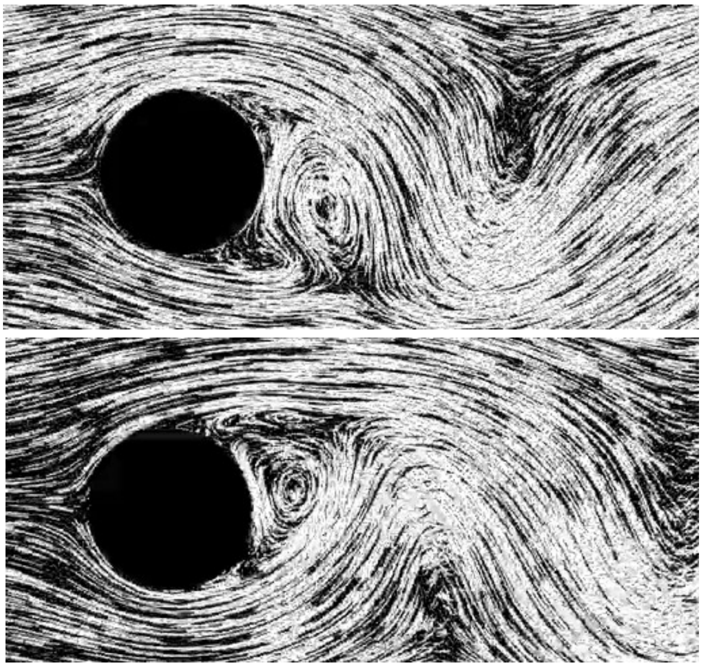


Fig 1: RANS simulation with clear low-frequency oscillation components.

RESEARCH TOPICS

- Perform a literature study to learn about the advanced RANS models.
- Develop a suitable model based on the RANS equations for the means and oscillations.
- Implement this model in an in-house turbulent flow solver, for turbulent channel flow (see below).
- Analyze the accuracy of the obtained solutions in comparison to a typical eddy-viscosity RANS model.

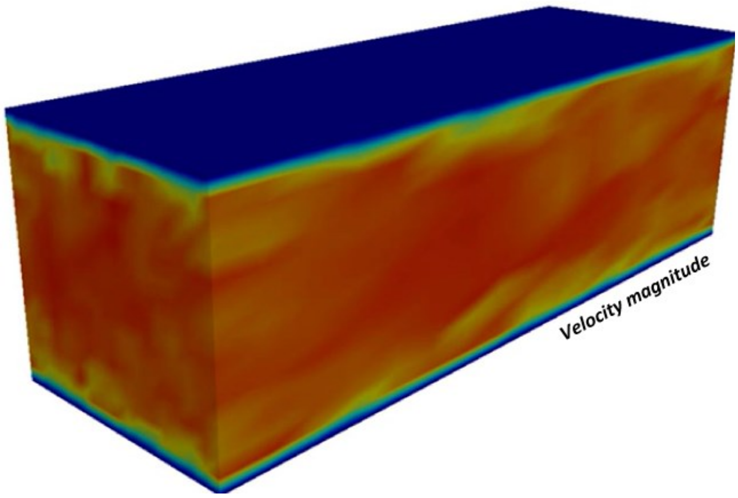


Fig 2: Main testcase: turbulent channel flow

STUDENT PROFILE

We are looking for a MSc student who has:

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

REFERENCES

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

Supervisor	David Rieder
2nd supervisor	
Mentor	
Company	---
Starting date	Asap
Exp./Num./Design	Numerical

ETFD

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Enabling seasonal storage of hydrogen in the Netherlands

David Rieder, Maja Rücker

d.r.rieder@tue.nl

INTRODUCTION

The transition to renewable energy sources inevitably leads to seasonal variations in the EU-wide energy supply. Large-scale storage of hydrogen is considered an attractive approach to buffer such seasonal changes[1]. With the recent end of the extraction of fossil gas in the Netherlands, such a solution may be provided by storing hydrogen in the Groningen gas fields, as shown in figure 1. However, besides the technological challenges, economical feasibility is still subject to high uncertainty.

TASKS

As part of this project, you will:

- Define key performance indicators (KPI) for subsurface hydrogen storage
- Build a process model of the reservoir
- Estimate CAPEX and OPEX of such a facility
- Identify the most influential technical challenges

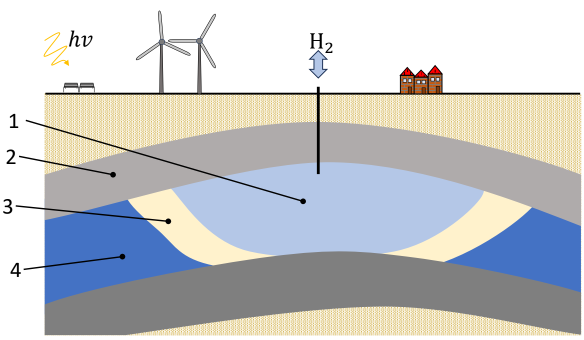


Figure 1: schematic hydrogen storage in Groningen, with (1) hydrogen, (2) cap rock, (3) cushion gas and (4) brine

GOALS

Your goal is to conduct a techno-economical assessment of subsurface hydrogen storage in the Groningen gas fields. As part of this assessment, you will estimate the operational (OPEX) and investment (CAPEX) costs, as well as determine the role of such a storage facility in the transitioning Dutch and European economy. Based on your analysis, you will provide guidance to future research directions, which promise to maximize the chance of success of seasonal hydrogen storage.

STUDENT PROFILE

You are enthusiastic about exploring novel approaches to enable the transition towards renewable forms of energy and would like to play a role in guiding research by identifying the knowledge gaps with the highest impact. Additionally, you have an affinity for numerical modelling of processes and want to develop your analytical skills further.

REFERENCES

[1] European strategy on hydrogen, https://eur-lex.europa.eu/legal-content/EN/TXT/?uri=CELEX:52020DC0301, accessed 20.10.2023

Supervisor	Dr. Azahara Luna-Triguero
2nd supervisor	Shima Rezaie
Mentor	N.A.
Company	Internal
Starting date	Any time
Exp./Num./Design	Numerical

ETFD

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

ASSESSING METAL-EXCHANGED MATERIALS FOR HYDROGEN STORAGE

S. Rezaie, A. Luna-Triguero
E-mail: s.rezaie@tue.nl, a.luna.triguero@tue.nl

INTRODUCTION

There are two main reasons for which hydrogen is considered the energy solution of the future; i) the highest gravimetric energy density known, ii) no carbon dioxide emissions.
Due to its low density under ambient conditions, the storage of hydrogen is challenging energy intensive; some solutions for storing hydrogen are compressed hydrogen gas in stationary tanks or underground cavities, and cryogenic liquid [1].
In this regard, nanoporous materials are being proposed as an alternative storage solution for hydrogen. While porous materials offer the potential for this application, certain limitations, such as adsorption capacity and extreme operating conditions of temperature and pressure need to be addressed. The DOE established targets of binding energy, gravimetric and volumetric capacity, and cost [2,3].

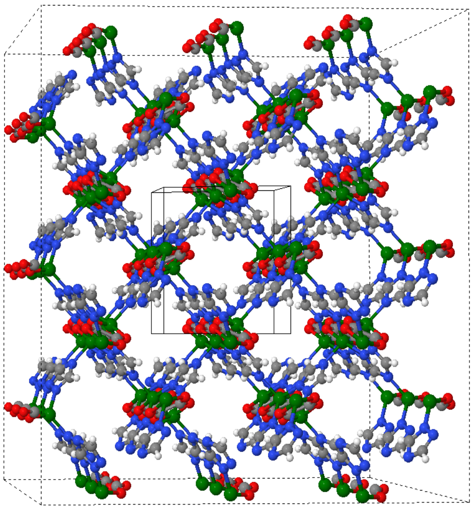
GOAL

Modify porous materials to assess the effect of metal center (and other surface modifications) in the capacity and conditions of hydrogen storage in MOFs.

TASK

In this project, you are expected to:

- Review relevant literature on hydrogen adsorption in porous materials.
- Computational generate hypothetical structures based on reported materials. (Figure 1)
- Test the adsorption capacity and conditions of the new structures and the application range.



STUDENT PROFILE

We are looking for a high-motivated MSc student who has:

- An interest in fundamental and computational work.
- Hands-on attitude toward new challenges.
- Analytical capacity
- Eager to participate as an active member of the group
- Experience with linux os and bash command lines is desired but not mandatory

REFERENCES

[1] Flynn, T. (2004), Cryogenic Engineering, 2nd Ed. Taylor & Francis. ISBN: 0824753674
[2] S. I. Hwang et al., *Metal-Organic Frameworks on Palladium Nanoparticle-Functionalized Carbon Nanotubes for Monitoring Hydrogen Storage*, ACS Appl Nano Mater, 2022.
[3] H. W. Langmi, N. Engelbrecht, P. M. Modisha, and D. Bessarabov, *Hydrogen storage* Electrochemical Power Sources: Fundamentals, Systems, and Applications, 2022, pp. 455–486.

Supervisor	Dr. Azahara Luna-Triguero
2nd supervisor	Dr. Monica E. A. Zakhari
Mentor	
Company	Internal
Starting date	Any time
Exp./Num./Design	Numerical

ETFD

TU/e EINDHOVEN UNIVERSITY OF TECHNOLOGY

CHILLING WITH NANOFLUIDS: Atomistic Insights

A. Luna-Triguero, M. E. A. Zakhari
E-mail: a.luna.triguero@tue.nl, m.e.a.zakhari@tue.nl

INTRODUCTION

The pursuit of energy-efficient and environmentally friendly refrigeration and heat transfer technologies has become paramount due to the escalating demands for cooling in various industrial, residential, and commercial sectors. Conventional refrigerants, such as hydrofluorocarbons (HFCs) and hydrochlorofluorocarbons (HCFCs), have raised significant environmental concerns due to their high global warming potential (GWP) and ozone-depleting properties. As a result, there is an urgent need to explore alternative approaches that can enhance cooling and heat transfer performance and mitigate the environmental impact of refrigeration systems.

One promising alternative in the quest for innovative refrigeration and heat transfer solutions involves the use of nanofluids [1]. Nanofluids are engineered suspensions of nanoparticles in conventional heat transfer fluids, such as water or refrigerants. Incorporating nanoparticles, particularly Metal-Organic Frameworks (MOFs) and zeolites, into these fluids has garnered significant attention for their exceptional thermal properties and potential applications in advanced cooling systems. [2,3]

GOAL

Compute using molecular simulations and ML potentials relevant properties of nanofluids (Fig. 1) for cooling applications.

TASK

In this project, you are expected to:

- Review relevant literature on nanofluids MOFs@Rx pairs.
- Compute relevant properties of the species e.g. heat capacity and thermal conductivity.
- Assess the performance and efficiency of the systems.

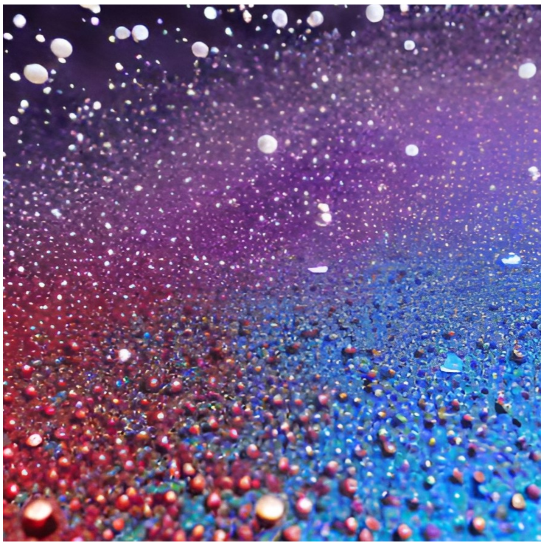


Fig. 1. Nanoparticle suspension in refrigerant. Schematic representation.

STUDENT PROFILE

We are looking for a high-motivated MSc student who has:

- An interest in fundamental and computational work.
- Hands-on attitude toward new challenges.
- Analytical capacity
- Eager to participate as an active member of the group
- Experience with linux os and bash command lines is desired but not mandatory

REFERENCES

[1] McGrail, B. P., Thallapally, P. K., Blanchard, J., Nune, S. K., Jenks, J. J., & Dang, L. X. (2013). Metal-organic heat carrier nanofluids. Nano Energy, 2(5), 845-855.
[2] Nandasiri, M. I., Liu, J., McGrail, B. P., Jenks, J., Schaef, H. T., Shutthanandan, V. (2016). Increased thermal conductivity in metal-organic heat carrier nanofluids. Scientific Reports, 6(1), 27805.
[3] Hu, J., Liu, C., Li, Q., & Shi, X. (2018). Molecular simulation of thermal energy storage of mixed CO2/IRMOF-1 nanoparticle nanofluid. International Journal of Heat and Mass Transfer, 125, 1345-1348.

Supervisor	Henk Ouwerkerk
2nd supervisor	Rick de Lange
Daily supervisor	NA
Company	TU/e & Heat Power
Starting date	TBD
Exp./Num./Design	Experimental

ETFD

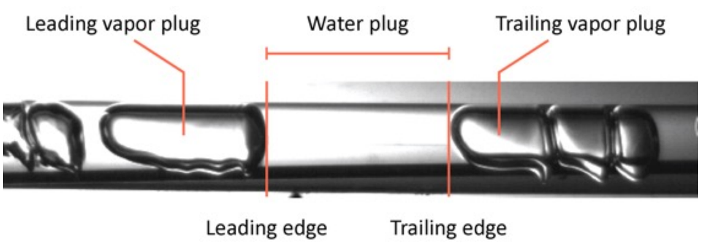
EINDHOVEN UNIVERSITY OF TECHNOLOGY

Study of steam generation phenomena in the once through boiling visualization set-up

Henk Ouwerkerk & Rick de Lange
H.Ouwerkerk@tue.nl & H.C.d.Lange@tue.nl

INTRODUCTION

Heat Power and TU/e together developed the sub critical once through boiler: a fast responding steam generator that can start from cold within minutes. Also, the once through boiler is of modular design and can be easily adapted for different applications. Heat Power has already successfully realized several real-size once through steam boilers (OTB's) in the range 80-2000kW thermal power. The principleworks, however the boiling process can still be unstable. To study and visualize the once through boiling process, an experimental set-up with a quartz boiler tube and high speed cameras was realized, see pictures below.



TASKS & GOALS

- Perform experiments with the setup with- and without the boiling nucleation system (a bubble generator)
- Study the occurring boiling nucleation phenomena and find the reproducibility.
- Study literature for similar experiments or simulations and check both similarities and novelties that might be eligible for publication in a journal

STUDENT PROFILE

- Hands on mentality
- Eager to gain experimental experience
- General Matlab skills
- General scientific writing skills

REFERENCES

Contact
Henk Ouwerkerk: H.Ouwerkerk@tue.nl
and
Rick de Lange: H.C.d.Lange@tue.nl

Supervisor	Henk Ouwerkerk
2nd supervisor	Rick de Lange
Daily supervisor	NA
Company	TU/e & Heat Power
Starting date	TBD
Exp./Num./Design	Experimental

ETFD

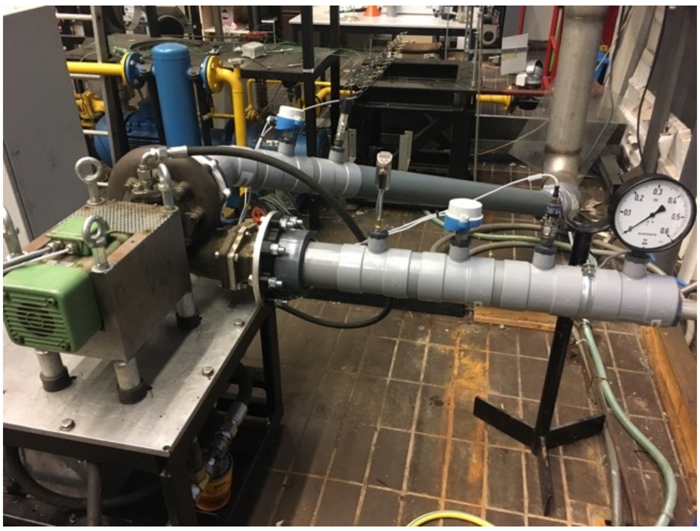
EINDHOVEN UNIVERSITY OF TECHNOLOGY

Experimental set-up for turbine volute testing to increase RCG-turbine output

Henk Ouwerkerk & Rick de Lange
H.Ouwerkerk@tue.nl & H.C.d.Lange@tue.nl

INTRODUCTION

Heat Power & TU/e developed a compact, fast responding and Steam and gas turbine: the Rankine Compression gas turbine (RCG). The RCG implements a radial free power turbine that drives a high-speed generator in direct drive. The maximum allowable speed of the generator is 21000RPM. However, this is still a relatively low speed for the turbine, that limits the pressure ratio and therefore the power output. Numerical studies showed that altering the size of the housing of the expansion turbine, can improve the turbine power output at lower RPM's. An experimental set-up at TU/e was realized to be able to experiment with different turbine housing designs (fig below). The set-up is operated with cold air to make it possible to perform experiments with non-metallic 3D printed turbine houses.



TASKS & GOALS

- Realize an improved experimental set-up that comprises a centrifugal blower so that the turbine volutes can be tested with larger air flows
- Design alternative turbine volutes & realize the designs by 3D printing.
- Implement the printed turbine volutes in the set-up and perform experiments.
- Advise on future steps for altering the turbine housing to increase the turbine output at lower RPM's.
- Pursue goals as described in the above. Note that the scope is flexible and that there will be support of staff with turbo technology, power electronics, utilities, data-acquisition and control. The supervisor and Master student together will assess which of the above goals are feasible within this assignment and where the focus of this assignment will be.

STUDENT PROFILE

- Hands on mentality
- Eager to gain experimental experience

REFERENCES

Henk Ouwerkerk: H.Ouwerkerk@tue.nl
and
Rick de Lange: H.C.d.Lange@tue.nl

Supervisor	Dr. Yousef Damianidis Al Chasanti
2nd supervisor	Dr. Camilo Rindt
Mentor	
Company	
Starting date	January 2024
Exp./Num./Design	Num

ETFD

EINDHOVEN UNIVERSITY OF TECHNOLOGY

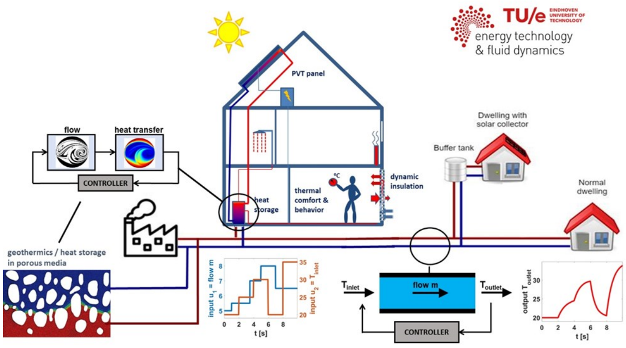
Hydraulic and Thermal Analysis of District Heating Networks

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

Yousef Damianidis Al Chasanti
g.damianidis.al.chasanti@tue.nl

INTRODUCTION

Due to the serious effects of climate change and increasing global temperatures, many countries are actively working to decarbonize their economies. Heating and cooling activities in modern economies account for 50% of global energy consumption and 45% of carbon emissions [1]. District heating networks transport hot fluids, usually water, from the energy source to individual buildings. The centralized heat production in district heating networks is more energy-efficient than individual building heat generation. Our group has developed a numerical model tailored for district heating networks, predicting heat loss and temperature distribution. This model :t.



TASKS

- Develop mathematical formulations that govern the flow in hydraulic piping networks for district heating. These equations will encompass the flow rates for each node and the head losses in each segment of the network. The mathematical formulations must be flexible enough to accommodate any arbitrary number of sources and demand nodes.
- Write a computer code to solve the piping network equations using the Hardy Cross iterative method [2].
- Combine the hydraulic pipe network model with the existing unsteady-state thermal model that calculates the temperature distribution along the pipe segment [3].
- Conduct an analysis using the combined thermo-hydraulic model and test it in real-world scenarios related to district heating networks.

GOALS

- Assess the feasibility of using pipe network analysis for modelling district heating networks.
- Examine how the configuration of the piping network impacts the performance of district heating networks.
- Estimate the head loss and compare it to the thermal losses.

STUDENT PROFILE

- Experience in thermo-fluid engineering with a passion for sustainable energy projects.
- Knowledge of basic programming principles.
- Experience in numerical analysis methods, including finite difference and iterative solvers.
- Experience in pipe flow analysis is preferred but not required.

REFERENCES

1. Mckinsey &Company (2021). Net-zero heat long-duration energy storage to accelerate energy system decarbonization, Technical Report
2. Bhave, P. R. (1991). Analysis of flow in water distribution networks. Technomic Publishing Co., Inc., Lancaster, PA., Book
3. Jiang, M., Speetjens, M. F. M., Rindt, C. C. M., & Smeulders, D. M. J. (2023). A data-based reduced-order model for dynamic simulation and control of district-heating networks. Applied Energy, 340, Article

Supervisor	Camilo Rindt
Company supervisor	Ruben van Gaalen
Mentor	
Company	Philips
Starting date	Any
Exp./Num./Design	Numerical

ETFD

EINDHOVEN UNIVERSITY OF TECHNOLOGY

Optimal cooling topology for MRI gradient coils

Camilo Rindt, Ruben van Gaalen
c.c.m.rindt@tue.nl, ruben.van.gaalen@philips.com

INTRODUCTION

MRI scanners are a crucial tool for diagnosing a wide range of afflictions, most prominently cancer. By magnetizing hydrogen atoms in a body, it becomes possible to visualize the different kinds of tissues that are present.

One important type of electromagnet in a MRI system is the so-called gradient coil. These coils supply a linearly sloped, time varying magnetic field in the three Cartesian degrees of freedom (i.e. x, y, z) by means of a carefully designed electric current pattern. By inducing these gradients on the magnetic field, it becomes possible to distinguish scanned regions from each other.

At Philips, several methods are employed for optimization of the electromagnetic design of gradient coils [1]. However, the resulting conductor patterns also require cooling by means of water flowing through tubes. For the positioning of these tubes, there is still demand for a more rigorous optimization method than those currently in use.

TASKS

Create an axisymmetric thermal model of a gradient coil. By keeping this model as simple as possible it can be used to perform fast optimizations for the positioning of cooling tubes.

Figure 1: Gradient coil

GOALS

- Create an axisymmetric thermal finite element model.
- Use linear/quadratic programming optimization techniques to find optimal positions of cooling tubes.
- Possibly, extend the model with nonlinearities that can be optimized with iterative methods.

STUDENT PROFILE

- Proficiency with MATLAB or similar scripting languages (e.g. Python).
- Knowledge about heat transport and the underlying equations.
- Knowledge about linear algebra.
- Proficiency with numerical methods.
- Knowledge about electromagnetics and/or fluid dynamics is a plus.

REFERENCES

[1] Peeren, G.N. (2003) Stream Function Approach for Determining Optimal Surface Currents. PhD thesis. Eindhoven University of Technology.

Supervisor	Dr. Michael Abdelmalik
2nd supervisor	Dr. Michel Speetjens
Mentors	Joost Prins and Hugo Melchers
Company	N.A.
Internal / External	Internal
Starting date	Any time
Exp./Num./Design	Num./Exp.

ETFD

EINDHOVEN UNIVERSITY OF TECHNOLOGY

NEURAL OPERATORS FOR OPTIMAL CONTROL OF HEAT TRANSFER

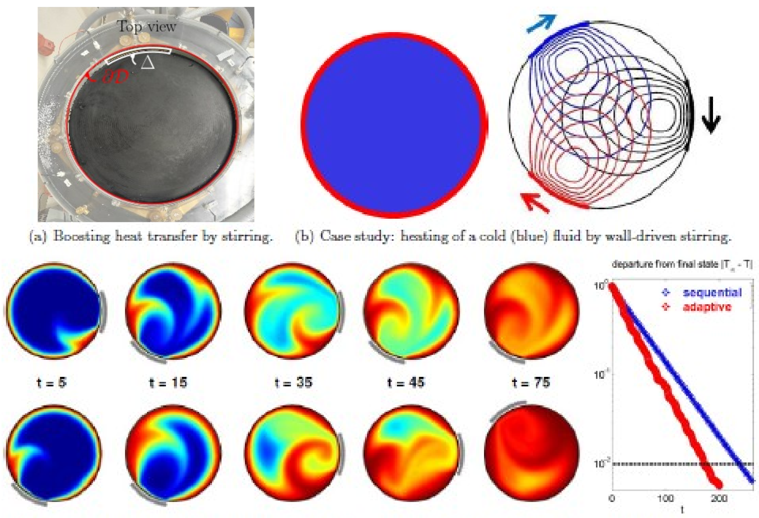
Michael Abdelmalik*, Michel Speetjens, Joost Prins and Hugo Melchers
*E-mail: m.abdel.malik@tue.nl

INTRODUCTION

Simulation of complex physical phenomena has been the primary focus of computational sciences, where methods are developed to solve PDEs. However, conflicting constraints of accuracy and tractability have hindered the application of many of such methods, and their reduced order counterparts, to control problems. In this project we are concerned with controlling the transfer of heat during stirring.

ROTATING ARC MIXER

We consider the heat transfer within a fluid in an industrial Rotating Arc Mixer (Fig 1a) [1]. While fluid is initially at room temperature, it is heated via the mixer walls (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). We ask the question: **what is the optimal stirring sequence for distributing the heat within the fluid?**



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

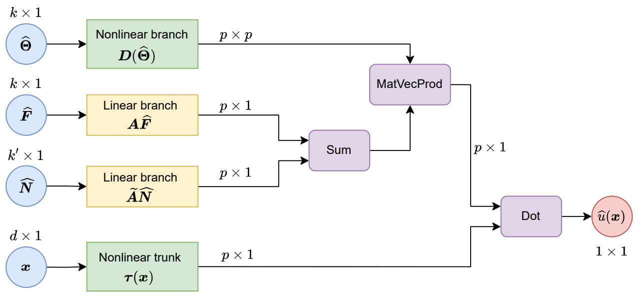
(c) Flow-enhanced heating driven by sequential (top) vs. adaptive (bottom) wall actuation. (d) Heating rate.

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

While counterintuitive, sequential actuation of the sliding walls is suboptimal (Fig. 1c) for distributing heat within the fluid [1]. To develop an optimal/adaptive stirring strategy we would like to design a controller that optimizes the actuation of the sliding walls for optimal heat distribution (eg Fig. 1c, bottom) [1].

NEURAL OPERATORS FOR HEAT TRANSFER

A physics-based heat transfer model predicts the effects of the possible actions of the controller, and the controller bases its decision on these predictions. Although a heat transfer model based on partial differential equations is at our disposal, solving these equations takes too long to be effective when used in hybrid with a controller in an experimental setting. In this project, we want to explore the potential of fast and variationally-based surrogate neural operators, e.g. as shown in Fig. 2 [2], when used in hybrid with a controller.



RESEARCH TOPICS

- The design and training of a neural surrogate [2] for heat transfer testing it in hybrid with a controller in numerical (and possibly laboratory) experiments.
- Refinement of the control strategy in [1] for achieving maximum heat transfer.
- Validation of the heat transfer controller against experiments.

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] Patel, D., Ray, D., Abdelmalik, M. R., Hughes, T. J., & Oberai, A. A. (2022). Variationally mimetic operator networks. *arXiv preprint arXiv:2209.12871*.