

# Infrasound Cooling of Steel Plates

Modeling and Simulation of an Acoustic Cooling System



**LUNDS UNIVERSITET**  
Lunds Tekniska Högskola

Master Thesis

by

**Olle Vahlin and Lisa Lignell**

Supervisor: Christer Fureby

Examiner: Jens Klingmann

May 2026

ISRN LUTMDN/TMHP-26/5704-SE  
ISSN 0282-1990

Department of Energy Sciences  
Faculty of Engineering, LTH  
Lund University  
Lund, Sweden

## Acknowledgements

We would like to express our deepest gratitude to our supervisor, Christer Fureby, for his invaluable guidance and for supporting us throughout this work. Not only did he introduce us to this project, but to the entire world of CFD when we first walked into his classroom in September 2024.

We would like to thank the whole team at Infrasonik for welcoming us at their office for a week and helping us along the way. A special thanks to Jakob Ronnhammar for always answering all our questions. We also want to thank Jonas Engdahl and Anders Carlestam from SSAB, for great discussions and support.

A special thank you to Himani Garg and Jens Klingmann for taking time out of their day to help and encourage us.

Lastly, we would like to thank each other for yet another successful group project!

*Thank you!*  
*Olle Vahlin and Lisa Lignell*  
Lund, May 2026

## Abstract

As global steel demand continues to rise, the steel industry faces increasing pressure to reduce its energy consumption. Conventional cooling is inefficient, and novel technologies have the potential to improve these processes. One promising approach is the use of pulsating flow to enhance heat transfer during cooling of hot steel plates. In this thesis Infrasonik's infrasound cooling solution is simulated using Computational Fluid Dynamics in the software OpenFOAM. A standing wave is established and the coupling of pressure and velocity is explored and documented. Additionally a hot plate is inserted into the domain, and the method of heat transfer from the plate to the surrounding air volume is studied. It is concluded that despite the limitations of the simulation model, using infrasound as a method of cooling in industrial applications has merit and provides a new and improved cooling solution. Making use of simplified boundary conditions in an LES based model, gave excellent qualitative results, which depict the coupling of pressure and velocity. Showing how non-linearity transfers between them, offers more insight into how the pressure driven reversal of flow works. This flow, and its reversal is shown to be able to transfer the heat from a hot plate to the air, and then mix that air which is able to be absorbed into the cooling fins of the surrounding wall. This process yielded a heat transfer rate of  $70 \text{ w/m}^2\text{K}$  for a plate heated to  $700 \text{ K}$ .

# Contents

|          |                                                   |           |
|----------|---------------------------------------------------|-----------|
| <b>1</b> | <b>Introduction</b>                               | <b>1</b>  |
| 1.1      | Background . . . . .                              | 1         |
| 1.2      | Aim . . . . .                                     | 3         |
| 1.3      | Delimitations . . . . .                           | 3         |
| <b>2</b> | <b>Theory</b>                                     | <b>4</b>  |
| 2.1      | Heat transfer . . . . .                           | 4         |
| 2.1.1    | Conduction . . . . .                              | 4         |
| 2.1.2    | Convection . . . . .                              | 4         |
| 2.1.3    | Radiation . . . . .                               | 5         |
| 2.2      | Infrasound . . . . .                              | 5         |
| 2.2.1    | Non-linear acoustics . . . . .                    | 7         |
| 2.3      | Computational Fluid Dynamics . . . . .            | 8         |
| 2.4      | Governing Equations . . . . .                     | 9         |
| 2.4.1    | Conservation of mass . . . . .                    | 9         |
| 2.4.2    | Conservation of momentum . . . . .                | 9         |
| 2.4.3    | Conservation of energy . . . . .                  | 10        |
| 2.5      | Turbulence modelling . . . . .                    | 10        |
| 2.5.1    | LES . . . . .                                     | 12        |
| 2.5.2    | Sub-grid scale modelling . . . . .                | 12        |
| 2.5.3    | Wall modelling . . . . .                          | 13        |
| <b>3</b> | <b>Numerical Theory</b>                           | <b>16</b> |
| 3.1      | Meshing . . . . .                                 | 16        |
| 3.2      | The Finite Volume Method . . . . .                | 16        |
| 3.3      | Discretization Schemes . . . . .                  | 17        |
| 3.3.1    | Time discretization scheme . . . . .              | 17        |
| 3.3.2    | Gradient scheme . . . . .                         | 18        |
| 3.3.3    | Divergence scheme . . . . .                       | 18        |
| 3.3.4    | Laplacian scheme . . . . .                        | 19        |
| 3.3.5    | Courant Number . . . . .                          | 19        |
| <b>4</b> | <b>Computational Setup</b>                        | <b>20</b> |
| 4.1      | Geometry . . . . .                                | 20        |
| 4.2      | Mesh . . . . .                                    | 21        |
| 4.3      | Boundary Conditions . . . . .                     | 22        |
| 4.4      | Solvers . . . . .                                 | 25        |
| 4.5      | Schemes . . . . .                                 | 25        |
| 4.6      | Model evaluation . . . . .                        | 25        |
| <b>5</b> | <b>Results</b>                                    | <b>27</b> |
| 5.1      | Ambient temperature steel plate - 300 K . . . . . | 27        |
| 5.2      | Hot steel plate - 700 K . . . . .                 | 34        |
| 5.2.1    | Heat transfer coefficient and heat flux . . . . . | 39        |



|                 |                                                                                          |           |
|-----------------|------------------------------------------------------------------------------------------|-----------|
| <b>6</b>        | <b>Discussion</b>                                                                        | <b>43</b> |
| 6.1             | Setup evaluation . . . . .                                                               | 43        |
| 6.2             | Ambient temperature steel plate - 300 K . . . . .                                        | 45        |
| 6.3             | Hot steel plate - 700 K . . . . .                                                        | 46        |
| <b>7</b>        | <b>Conclusion</b>                                                                        | <b>49</b> |
| <b>Appendix</b> |                                                                                          |           |
| <b>A</b>        | <b><math>U_x</math> with SLIC in the y-normal plane</b>                                  | <b>54</b> |
| <b>B</b>        | <b><math>P</math> at <math>t = 0.493</math> s in different x-normal cross sections</b>   | <b>57</b> |
| <b>C</b>        | <b><math>U_x</math> at <math>t = 0.494</math> s in different z-normal cross sections</b> | <b>59</b> |
| <b>D</b>        | <b>Miscellaneous</b>                                                                     | <b>60</b> |
| D.1             | Energy spectrum as a function of data frequency . . . . .                                | 60        |
| D.2             | Project Timeline . . . . .                                                               | 61        |
| D.3             | Division of work . . . . .                                                               | 61        |

## List of Figures

|    |                                                                                                                                                                                                                                                                                                                   |    |
|----|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| 1  | Sound wave characteristics. . . . .                                                                                                                                                                                                                                                                               | 5  |
| 2  | Peak sound pressure and rms value. . . . .                                                                                                                                                                                                                                                                        | 6  |
| 3  | Open ended resonance tube with standing pressure and displacement waves. The red and blue curves represents the peak and trough distributions of pressure and displacement in the tube at either extremes. . . . .                                                                                                | 6  |
| 4  | Closed resonance tube with standing pressure and displacement waves. The red and blue curves represents the peak and trough distributions of pressure and displacement in the tube. . . . .                                                                                                                       | 7  |
| 5  | The figure shows a sinusoidal pressure wave, and two extreme cases where non-linear effects turn the wave into a sawtoothed shape. The left image shows a transverse wave travelling to the right, and the right image shows a standing wave just before interference. . . . .                                    | 8  |
| 6  | Cascading effect, where large eddies turn into smaller eddies. . . . .                                                                                                                                                                                                                                            | 11 |
| 7  | Boundary layer for laminar flow (left) and for turbulent flow (right). [36] . . . . .                                                                                                                                                                                                                             | 14 |
| 8  | CAD model of the cooling machine. Image (b) measures 5,7 by 2,7 meters. . . . .                                                                                                                                                                                                                                   | 20 |
| 9  | Simplified representation of the internal volume of the cooling system. Visualised at three angles and rotated using first angle projection. . . . .                                                                                                                                                              | 21 |
| 10 | The simplified geometry, with bodies of influence inserted as dashed rectangles. . . . .                                                                                                                                                                                                                          | 22 |
| 11 | The simplified geometry with boundary surfaces highlighted with red and the potential flow directions shown with arrows. . . . .                                                                                                                                                                                  | 23 |
| 12 | Coded boundary condition for <i>inlet1</i> . . . . .                                                                                                                                                                                                                                                              | 24 |
| 13 | Pressure field of a central slice in the y-normal direction, at time steps 0.480 - 0.550 s. . . . .                                                                                                                                                                                                               | 27 |
| 14 | Velocity magnitude field $U_{mag}$ of a central slice in the y-normal direction, at time steps 0.480 - 0.550 s. Note how the colour gradient shifts with the maximum and minimum value. . . . .                                                                                                                   | 28 |
| 15 | Surface and line plots of pressure as a function of distance and time. Distance is measured from pulsator to pulsator. . . . .                                                                                                                                                                                    | 28 |
| 16 | Line plots of pressure as a function of time and space. . . . .                                                                                                                                                                                                                                                   | 29 |
| 17 | Cross section of the cooling box. X-normal at 1.5 meters from the centre of the cooling box. Pressure distribution at time 0.493 s. . . . .                                                                                                                                                                       | 29 |
| 18 | Velocity in the $U_x$ direction with surface LIC applied. Figures (a) through (e) show time steps 0.472 - 0.514 s, which corresponds to half a pressure cycle. The colour gradient is fixed between -120 and 120 m/s for consistent colouring. . . . .                                                            | 30 |
| 19 | Velocity in the $U_x$ direction. Figures (a) through (e) show time steps 0.467 - 0.514 s, corresponding to half a pressure cycle. The slice is placed 5 mm above the steel plate with the view in the z-normal direction. The colour gradient is fixed between -120 and 120 m/s for consistent colouring. . . . . | 32 |
| 20 | Velocity in the $U_x$ direction. Figures (a) through (e) show time steps 0.485 - 0.501 s. The slice is placed 5 mm above the steel plate with the view in the z-normal direction. The colour gradient is fixed between -120 and 120 m/s for consistent colouring. . . . .                                         | 33 |
| 21 | Line plot showing velocity in x-direction as a function of time and space. Colour gradient depicts the interval 5-10 meters, corresponding to the cooling chamber. . . . .                                                                                                                                        | 34 |
| 22 | Velocity in the $U_x$ direction. Time = 0.493 s. Slice is placed 5 mm above the steel plate. Upper half uses surface LIC. The colour gradient is fixed between -120 and 120 m/s for consistent colouring. . . . .                                                                                                 | 34 |

|    |                                                                                                                                                                                                                                        |    |
|----|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| 23 | Temperature field sliced in the y-normal direction. Time steps distributed over half a pressure cycle. The colour gradient is fixed between 280 - 390 K for consistent colouring. . . . .                                              | 35 |
| 24 | Temperature field slice in the z-normal direction, 5 mm above the steel plate. Time steps distributed over half a pressure cycle. The colour gradient is fixed between 280 - 390 K for consistent colouring. . . . .                   | 36 |
| 25 | Temperature field slice in the x-normal, in the centre of the domain. The colour gradient is fixed between 280 - 390 K for consistent colouring. . . . .                                                                               | 37 |
| 26 | Temperature field normal to the x-direction with slices at different locations. The colour gradient is fixed between 280 - 390 K for consistent colouring. . . . .                                                                     | 38 |
| 27 | Coloured by velocity in the $U_x$ direction, with surface LIC overlayed. Slice normal to the y-direction at time steps between 0.761-0.776 s. Note that the colour gradient is scaled according to maximum and minimum values. . . . . | 39 |
| 28 | Heat flux and the heat transfer coefficient of the steel plate as a function of time. The average and its interval marked in red. . . . .                                                                                              | 40 |
| 29 | Heat transfer rate from the steel plate, upper side. The colour gradient is fixed to 1000 - 70000 W/m <sup>2</sup> for consistent colouring. . . . .                                                                                   | 41 |
| 30 | Time averaged heat transfer rate and coefficient. In these images the colour gradient corresponds with max and min values. . . . .                                                                                                     | 42 |
| 31 | Velocity in the $U_x$ direction with surface line integral convolution applied. Figures (a) through (i) show time 0.485 - 0.562 s. The colour gradient is fixed between -120 and 120 m/s for consistent colouring. . . . .             | 56 |
| 32 | Pressure field at $t = 0.493$ s, normal to the x-direction with slices at different locations.                                                                                                                                         | 58 |
| 33 | Slices with normal in the z-direction, at timestep $t = 0.494$ s. Slices presented in a top down order, showing $U_x$ . . . . .                                                                                                        | 59 |
| 34 | The energy spectrum as a function of data frequency, with the theoretical curve $f^{-5/3}$ and the fitted curve between 20 - 100 Hz. Based on data range 0 - 0.532 s, time steps at 0.004 s interval. . . . .                          | 60 |
| 35 | Energy spectrum split by velocity component. Based on the same data as the figure above. . . . .                                                                                                                                       | 60 |
| 36 | Gant schedule of planned work over a 20 week timeline. . . . .                                                                                                                                                                         | 61 |
| 37 | Gant schedule of the actual work over a 20 week timeline. . . . .                                                                                                                                                                      | 61 |

## List of Tables

|   |                                                                                                                         |    |
|---|-------------------------------------------------------------------------------------------------------------------------|----|
| 1 | Cell sizes and growth rate for each body of influence. . . . .                                                          | 22 |
| 2 | Mesh quality metrics for the two final meshes. Based on values generated by Open-FOAM <i>checkMesh</i> command. . . . . | 22 |
| 3 | Boundary conditions and models. Values can be found in table 4. . . . .                                                 | 23 |
| 4 | Values applied to corresponding models and conditions from table 3. . . . .                                             | 24 |
| 5 | Schemes and their correctors that are used in the simulation setup. [42] . . . . .                                      | 25 |
| 6 | Average heat transfer coefficient and heat flux. Averaged in time and space. . . . .                                    | 40 |

# Nomenclature

## Greek letters

|               |                                           |                                       |
|---------------|-------------------------------------------|---------------------------------------|
| $\beta$       | Bulk viscosity                            | [Pa · s]                              |
| $\Delta T$    | Temperature difference                    | [K]                                   |
| $\Delta t$    | Time step                                 | [s]                                   |
| $\Delta$      | Minimum resolved eddy size / filter width | [m]                                   |
| $\gamma$      | Heat capacity ratio                       | [-]                                   |
| $\Gamma_\phi$ | Diffusion coefficient                     | [-]                                   |
| $\kappa$      | von Kármán constant                       | [-]                                   |
| $\lambda$     | Wave length                               | [m]                                   |
| $\mu$         | Dynamic viscosity                         | [Pa·s]                                |
| $\mu_k$       | Sub-grid scale viscosity                  | [Pa·s]                                |
| $\nu$         | Kinematic viscosity                       | [m <sup>2</sup> /s]                   |
| $\nu_t$       | Turbulent kinematic viscosity             | [m <sup>2</sup> /s]                   |
| $\phi$        | General scalar quantity                   | [-]                                   |
| $\rho$        | Density                                   | [kg/m <sup>3</sup> ]                  |
| $\sigma$      | Stefan–Boltzmann constant                 | [W/(m <sup>2</sup> ·K <sup>4</sup> )] |
| $\tau_w$      | Wall shear stress                         | [Pa]                                  |
| $\varepsilon$ | Emissivity                                | [-]                                   |

## Roman letters

|                |                                  |                     |
|----------------|----------------------------------|---------------------|
| $\dot{q}$      | Heat transfer rate per unit area | [W/m <sup>2</sup> ] |
| $\hat{p}$      | Peak sound pressure              | [kPa]               |
| $\mathbf{D}$   | Rate of strain tensor            | [s <sup>-1</sup> ]  |
| $\mathbf{F}_b$ | Body force vector                | [N]                 |
| $\mathbf{S}$   | Viscous stress tensor            | [Pa]                |
| $\mathbf{v}$   | Velocity vector                  | [m/s]               |
| $A_s$          | Surface area                     | [m <sup>2</sup> ]   |
| $c$            | Speed of sound                   | [m/s]               |

|            |                                      |                                   |
|------------|--------------------------------------|-----------------------------------|
| $C_f$      | Skin friction coefficient            | [-]                               |
| $E$        | Total energy                         | [J]                               |
| $e$        | Internal energy                      | [J]                               |
| $f$        | Frequency                            | [Hz]                              |
| $h$        | Convective heat transfer coefficient | [W/(m <sup>2</sup> ·K)]           |
| $k$        | Thermal conductivity                 | [W/(m·K)]                         |
| $k_{SGS}$  | Sub-grid scale kinetic energy        | [m <sup>2</sup> /s <sup>2</sup> ] |
| $L$        | Tube length                          | [m]                               |
| $P$        | Pressure                             | [Pa]                              |
| $P_{ref}$  | Reference pressure                   | [Pa]                              |
| $Pr_t$     | Turbulent SGS Prandtl number         | [-]                               |
| $R$        | Specific gas constant                | [J/Kg ·K]                         |
| $T$        | Temperature                          | [K]                               |
| $T_\infty$ | Ambient fluid temperature            | [K]                               |
| $T_s$      | Surface temperature                  | [K]                               |
| $u^+$      | Dimensionless wall velocity          | [-]                               |
| $u_\tau$   | Friction velocity                    | [m/s]                             |
| $u_{max}$  | Maximum particle velocity            | [m/s]                             |
| $y^+$      | Dimensionless wall distance          | [-]                               |

## Abbreviations

|         |                                                                   |
|---------|-------------------------------------------------------------------|
| AdTherM | Advanced Computing for Sustainable Thermal Management in Industry |
| CAD     | Computer-Aided Design                                             |
| CFD     | Computational Fluid Dynamics                                      |
| CFL     | Courant–Friedrichs–Lewy                                           |
| DNS     | Direct Numerical Simulation                                       |
| FEA     | Finite Element Analysis                                           |
| FVM     | Finite Volume Method                                              |
| LES     | Large Eddy Simulation                                             |
| LIC     | Line Integral Convolution                                         |

|      |                                    |
|------|------------------------------------|
| PDE  | Partial Differential Equation      |
| RANS | Reynolds-Averaged Navier–Stokes    |
| RMS  | Root Mean Square                   |
| SGS  | Sub-grid Scale                     |
| SI   | International System of Units      |
| WALE | Wall-adapting Local Eddy-viscosity |

# 1 Introduction

The prospect of infrasound cooling offers a novel approach for energy efficient cooling of steel. As a part of the competence center Advanced Computing for Sustainable Thermal Management in Industry (AdTherM) this master thesis is conducted in collaboration with SSAB and Infrasonik AB. The technology that will be the topic of investigation is Infrasonik's InfraCooler. This project aims to give insight into what is happening in the flow field inside the InfraCooler, with the help of Computational Fluid Dynamics (CFD) using the open source software OpenFOAM.

In the following chapter, a background for the project will be provided together with an overview of previous studies done on the subject. The aim of this work will also be presented. In order to limit the scope of the project some delimitations have been made, which will also be presented in this chapter.

## 1.1 Background

Steel is one of the most fundamental materials in our society, serving as a key component in numerous sectors. Construction, infrastructure and the automotive industry are a few examples of industries relying heavily on steel. The global demand for steel has continuously increased since 1970 and will continue to do so as economies grow and build up their infrastructure. The worldwide demand for steel is predicted to further increase by more than a third by 2050. [1]

The production of steel is very emission-and energy-intensive, which has lead to recent efforts of decarbonization. The annual production of crude steel was 1885 million tonnes in 2024, with an average of 1.9 tonnes of CO<sub>2</sub> per tonne of crude steel cast in 2023. In addition to this, over 21 GJ of energy was consumed per tonne of crude steel cast in the same year. This corresponds to around 8% of the global final energy demand. [2] [1]

The cooling of steel is an essential step in the production process. The objective of cooling in the final stages is to enable safe handling further down the value chain. The final step where the processing of the steel is completed, but still hot, is to let it cool down to ambient temperature. If a steel plate is cooled by natural convection, i.e without external measures, the cooling rate is around 0.2 -2°C/s depending on the dimensions of the plate. This takes time, takes up space and the residual heat is wasted. Making this process more efficient by reducing time, space and reusing this heat is a step in the right direction of modernising the steel production process. In the current situation, if a faster cooling rate is desired, liquid cooling may be applied. But this process imposes challenges associated with safety, maintenance and product quality. [3] [4]

Increasing the rate of heat transfer can lower energy costs for cooling, and there are many ways to achieve this. For certain conditions, one way is to use pulsating flows. There are many studies about pulsating flow in a variety of fields, such as in industry and medicine. However, due to the complicated nature of the phenomena included in pulsating flows the mechanisms are not yet fully understood, and the existing research is often specific for its application. Ye et al. [5] provided a comprehensive review of pulsating flow on heat transfer enhancement in 2021. They highlighted the wide range of applications by discussing many different types of pulsating flow such as laminar pulsating flows in different channel geometries, two-phase pulsating flow and pulsating flow in porous media. The authors concluded that there is a lack of relevant theoretical research regarding turbulent pulsating flows due to the difficulty of turbulent theory research. Although this is the



case, turbulent pulsating flow is expected to enhance heat transfer more effectively than laminar flow because of improved mixing, higher flow rate and thinner boundary layer. There are however some research that indicates that a pulsating flow has no effect or a negative effect on heat transfer. The authors discuss that this contradiction suggests that the effects on heat transfer in pulsating flows are closely tied with the working conditions, such as the geometry or the experimental set-up.

In their review, Ye et al. provided a summary on different research methods for pulsating flow. They concluded that experimental studies are the most accurate, but that there are relatively few experiments in this field as the challenges associated with these are significant. The experiments are highly restricted to the conditions and the equipment. The equipment costs are generally high, and it is considered difficult to produce high frequency pulsating flow. Therefore, experimental conditions in the existing researches, focus on very different parameter ranges. [5]

Mikheev et al. [6] conducted an experimental study on heat transfer in pulsating air flow behind a rib in a channel. The study was carried out within the frequency range of 0-15 Hz, which is in the range for infrasound. The experiment showed an enhancement of heat transfer by 25% compared to steady flow. Two years later, Davletshin et al. [7] used a similar experimental set up to investigate the heat transfer and the velocity profile in pulsating flow in frequency range 0-30 Hz after a spanwise rib. The authors found a maximum local heat transfer enhancement of up to three times that of steady flow. Their study also showed a close correlation between the local heat transfer coefficient and the maximum transverse velocity in the channel. Another experimental study done by Davletshin et al. [8] revealed that in a similar channel but with multiple ribs and flow pulsations of frequency range 0-30 Hz, the heat transfer was enhanced with a maximum of 30% for the pulsating flow.

Although heat transfer in pulsating flow is not fully understood, it is not that novel to research. Woods [9] conducted a study in 1992 on using infrasound to enhance heat transfer from a cylinder. The objective of the study was to see if infrasound cooling would be more effective than a fan system, in terms of power consumption in an industrial setting. The set up used in the study consisted of an air compressor generating the infrasound and two resonance tubes, each 4.7 m long corresponding to a quarter of a wavelength, meaning that the frequency used was around 18 Hz. The results from these experiments showed that the heat transfer was more uniformly distributed around the cylinder when using infrasound in comparison to when only the fan was used. Additionally, the infrasound provided significant increases in the heat transfer from the cylinder. Also, when the infrasound cooling was combined with forced convection, the power required decreased with 12-75%.

As has now been established, the use of pulsating flow for heat transfer enhancement can be a solution for more efficient cooling in steel production. Infrasonik has developed a new technology called the InfraCooler, which cools hot steel plates using infrasound cooling. The idea is to have two pulsators generating controlled acoustic pressure oscillations operating in the infrasound frequency range. Resonance tubes are connected to each of the pulsators so that the pressure oscillations propagate through the tubes, with the acoustic resonance of the system allowing the establishment of a standing wave. These resonance tubes are connected to a cooling chamber, in which the hot steel plate is placed. The air remains enclosed in the system, so heat is removed by heat transfer from the steel plate to the oscillating air and from the oscillating air to the water cooled heat sinks. Even though the system is closed, the pulsating flow achieve particle velocities up to 90 m/s in the cooling chamber. The heat transfer between the steel plate and the oscillating air is enhanced as strong turbulence is induced in the cooling chamber, disrupting the thermal boundary layer at the

steel plate surface. This technology facilitates controllable, energy efficient cooling, and will be the topic of this master thesis. [3]

## 1.2 Aim

The aim of this work is divided into several objectives:

- To investigate if it is possible to simulate the InfraCooler with the use of CFD
- To learn what the internal velocity field looks like during a standing wave
- To learn how the temperature field changes during a standing wave
- To approximate a heat transfer coefficient of a hot steel plate in this cooling system

## 1.3 Delimitations

This project is subject to several delimitations that should be acknowledged. Heat transfer by thermal radiation is neglected to limit computational cost. The work is also limited to two cases, one simulation without heat transfer using a cold steel plate and one accounting for heat transfer by using a hot plate. No variations in sound pressure or frequency is made, as both the frequency and the sound pressure magnitudes are based on data from Infrasonik's experimental setup. No other fluid than air is used in the simulations.

Because of the high computational cost and time consuming simulations, no mesh sensitivity study was performed. The mesh generation was time consuming, due to the complexity of the geometry. This required a small cell size, driving up the lower limit of the cell count. The mesh generation was also limited by a lack of computational performance, which imposed a higher limit. These limitations restricted the possible mesh size range, making the potential for a mesh sensitivity study very poor and was thus omitted.

Experimental data was available in form of cooling curves at different pressure amplitudes, frequencies and temperature ranges. Because of resource limitations the simulations in this thesis only reached about one second of simulated time, it was not possible to compare the measured cooling curve with a simulation based equivalent. Therefore no proper experimental validation was feasible, and only minor attempts to validate the results were performed.

The purpose of this thesis is not to evaluate the performance of different models, but rather to evaluate the flow field of a case previously not studied. This coupled with the lack of viable experimental data, means no effort was made in testing what numerical models and setup best modelled the flow field in this study.

## 2 Theory

This thesis involves multiple different physical phenomena. This chapter presents descriptions of these along with derived equations that will be used. The concepts of heat transfer, infrasound and CFD will be explained.

### 2.1 Heat transfer

Heat transfer is the process of thermal energy exchange on account of a spatial temperature difference. This means that whenever there is a temperature difference present, either in a medium or between media, heat transfer will occur. Heat can be transferred by three different means, which are referred to as the three heat transfer modes conduction, convection and thermal radiation.[10]

#### 2.1.1 Conduction

In conduction, the heat transfer is considered governed by molecular motion in solid bodies and fluids at rest. Higher molecular energies are correlated to higher temperatures and lower energies with lower temperatures. Molecules adjacent to high energy regions continually collide. Therefore, transfer of thermal energy from more energetic molecules to less energetic molecules must occur. Heat conduction can be described from Fourier's law of heat conduction as seen in equation (1), here in differential form for 3D heat conduction.

$$\dot{q} = -k\nabla T \quad (1)$$

where  $\dot{q}$  is the heat transfer rate per unit area, also called the heat flux, in  $\text{W}/\text{m}^2$ ,  $k$  is the thermal conductivity in  $\text{W}/\text{m} \cdot \text{K}$  and  $\nabla T$  is the temperature gradient which unit is  $\text{K}/\text{m}$ . The thermal conductivity is material dependent and can be defined as the rate of heat transfer through a unit of thickness of the material per unit area per unit  $\Delta T$ . [10]

#### 2.1.2 Convection

Convection is defined as heat transfer through movement of a fluid. The amount of heat being transferred is dependent on the macroscopic fluid motion, specifically when a fluid is moving adjacent to a solid and the temperatures between the solid surface and the fluid are different. In a situation of no bulk flow of the fluid, heat transfer between the solid and the adjacent fluid is solely by conduction. If there instead exists any bulk flow motion in the fluid, the heat transfer is enhanced. The combined effects from the fluid's motion and conduction are incorporated into convection. [11] [10]

Convection can further be divided into natural and forced convection. It is called forced convection when a fluid flows over the surface forced by external measures, e.g a fan, the wind or a pump. If the fluid motion is caused by density-induced buoyancy forces the convection is called natural convection. Related examples are if a hot steel plate is placed in a surrounding of cooler air without external impact, the heat transfer will be of natural convection. If the plate would be placed in proximity of an active fan, the convection is instead forced. The rate of convective heat transfer can be expressed by Newton's law of cooling in equation (2) below.

$$\dot{Q}_{conv} = hA_s(T_s - T_\infty) \quad (2)$$

where  $h$  is the convection heat transfer coefficient (HTC) in  $\text{W}/\text{m}^2\cdot\text{K}$ ,  $A_s$  is the surface area in  $\text{m}^2$ ,  $T_s$  is the surface temperature and  $T_\infty$  is the temperature of the ambient fluid, both in K. [10]

### 2.1.3 Radiation

In contrast to conduction and convection, heat transfer by radiation is not bounded to have an intervening medium. Energy is emitted as electromagnetic waves from all bodies of finite temperature, as a consequence of changes in the electronic configurations of the molecules or atoms. The phenomenon of thermal radiation is volumetric, meaning that all gases, liquids and solids either emit, absorb or transmit radiation. However, for opaque solids radiation is often seen as a surface phenomenon. [10]

The rate of emitted radiation from a surface of temperature  $T_s$  can be described based on the Stefan-Boltzmann law as

$$\dot{Q}_{emit} = \varepsilon \sigma A_s T_s^4 \quad (3)$$

where  $\varepsilon$  is the emissivity of the surface, in the range  $0 \leq \varepsilon \leq 1$  and  $\sigma$  is the Stefan-Boltzmann constant  $\sigma = 5.670 \cdot 10^{-8} \text{ W}/\text{m}^2\cdot\text{K}^4$ . [10]

## 2.2 Infrasound

Sound below the audible spectrum is called infrasound, and this is commonly defined as frequencies below 20 Hz. Sound waves are characterized by the length of the wave  $\lambda$ , the amplitude, the speed of the wave and the frequency of the sound waves, as shown in figure 1. [12]

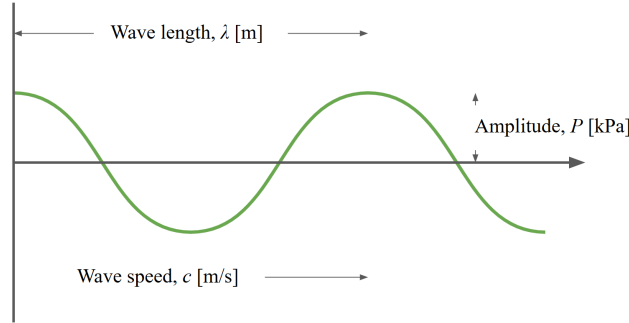


Figure 1: Sound wave characteristics.

A sound wave is a pressure wave, the amplitude of which is commonly measured in decibel (dB), in this thesis the SI unit Pascal (Pa) is used. The relationship between the two units can be seen in equation 4. Wave speed is determined by the medium, in this case ambient air. The relationship between frequency, speed of sound and the wave length can be seen in (5).

$$P = P_{ref} \cdot 10^{\left(\frac{dB}{20}\right)} \quad (4)$$

$$f = \frac{c}{\lambda} \quad (5)$$

Here the reference pressure  $P_{ref} = 20 \mu\text{Pa}$ . This means that if the medium and its properties do

not change, the wave length is inversely proportional to the frequency, and lowering the frequency increases the wave length. [12] [13]

Sound amplitude is measured relative to the ambient pressure, much like gauge pressure, and is referred to as sound pressure. It is not uncommon to talk about sound pressure and refer to the RMS value  $p_e$  and not the peak value  $\hat{p}$  as seen in figure 2. However, in this paper the sound pressure will refer to the peak value unless otherwise specified.

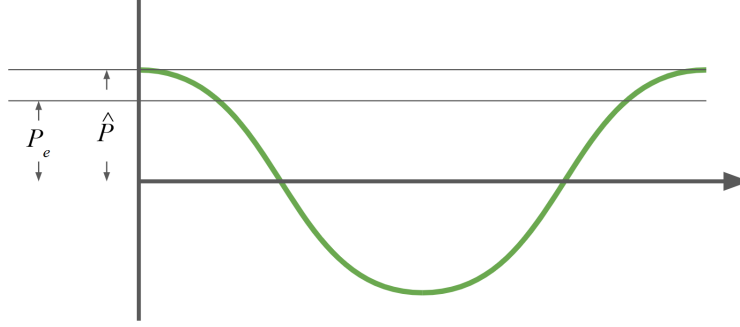


Figure 2: Peak sound pressure and rms value.

In this particular application we are looking at what is essentially a resonance tube. By utilizing the resonance frequency in the tube a standing wave is formed. When considering a sound wave in a open ended tube, as seen in figure 3, only specific frequencies create a standing wave. The frequency corresponds to a wavelength which is a quarter of the length of the tube or a higher order frequency defined by equation (6) where  $n$  is a whole number.

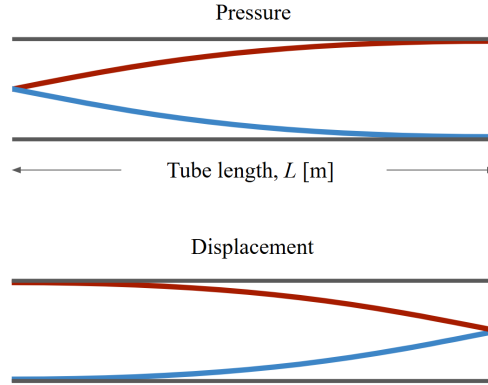


Figure 3: Open ended resonance tube with standing pressure and displacement waves. The red and blue curves represents the peak and trough distributions of pressure and displacement in the tube at either extremes.

$$L = n \frac{\lambda}{4} \quad (6)$$

As we can see in figure [3] there is a displacement node at the wall and its anti-node at the opening. This is mirrored for the pressure curve, where the open end is connected to the ambient atmosphere creating a pressure node and the closed end reflecting the pressure creating the anti-node. This

means that the pressure at the closed end ranges from the peak to the trough value, while the pressure at the open end remains fixed at an ambient pressure level. Conversely, the displacement or particle velocity stays fixed at zero near the closed end, and ranges from peak to trough value at the open end. [14]

In the context of the entire cooling system, the open ended tube only represents half of the total length, but by mirroring the tube and connecting the two open ends together, the system replicates acoustic properties of the cooling solution, like in figure 4. This new construction has the same series of resonance frequencies as the open ended tube even though the length has now doubled. The standing wave is now half a full wave length instead of a quarter, doubling both sides of equation (6), which is of course equivalent, as seen in equation (7).

$$2L = n\lambda/2 \quad \Leftrightarrow \quad L = n\lambda/4 \quad (7)$$

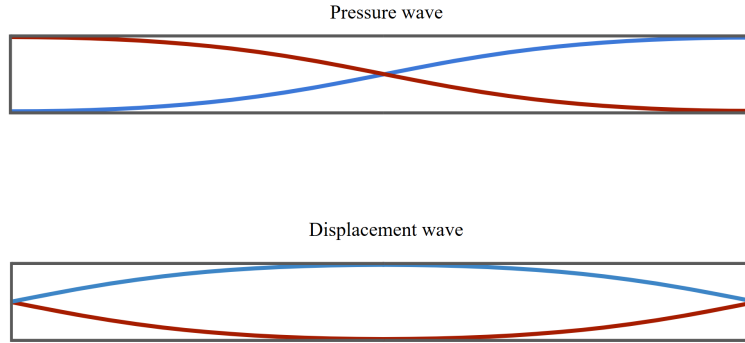


Figure 4: Closed resonance tube with standing pressure and displacement waves. The red and blue curves represents the peak and trough distributions of pressure and displacement in the tube.

In figure 4 the displacement curve shows how there is no movement at the wall and a lot of movement at the centre of the tube. The speed of this displacement of air is called particle velocity, which unlike the speed of sound specifies the velocity of individual molecules of air, instead of the speed of the pressure wave. In this paper, velocity is the same as particle velocity. This particle speed is driven by the pressure and can be calculated using equation 8.

$$u_{max} = \frac{\hat{p}}{\rho c} \quad (8)$$

Where  $\rho$  is the density of the medium in  $\text{kg/m}^3$ . Figure 4 also shows how the displacement wave varies with time relatively to the pressure wave. The peak of the pressure wave is located one quarter wave to the right, or rather the displacement wave is delayed by  $90^\circ$  at a stationary point. Which means that the displacement wave peaks and the flow direction turns as the pressure peaks.[13] [15] [16]

### 2.2.1 Non-linear acoustics

What was previously described is known as linear acoustic, and this paragraph covers non-linear acoustics. This causes changes in the behaviour of the sound waves, which occurs when the pressured amplitude is great enough to distort the shape of the wave. According to Greenspoon the

sound pressure needs to exceed 177 dB, which corresponds to a pressure amplitude of about 14 kPa, which is exceeded in this thesis [17]. A characteristic of non-linear acoustics is the unique shape of the wave. In linear acoustics the shape of the wave is sinusoidal, but due to the high pressure levels, the local temperature in the high pressure zones increase enough to change the local speed of sound  $c = \sqrt{\gamma RT}$ . This causes the peaks of the pressure wave to catch up the troughs of the wave, creating a sawtooth like wave pattern, like in figure 5a. In the standing wave one half of the wave is travelling in the other direction, and will change the overall shape into something more like figure 5b. As this is mathematically non-linear, the numerical simulation needs to be able to handle this as well. Fortunately the governing equations are built using partial differential equations, and are able to capture the nonlinearity of the problem. [17]

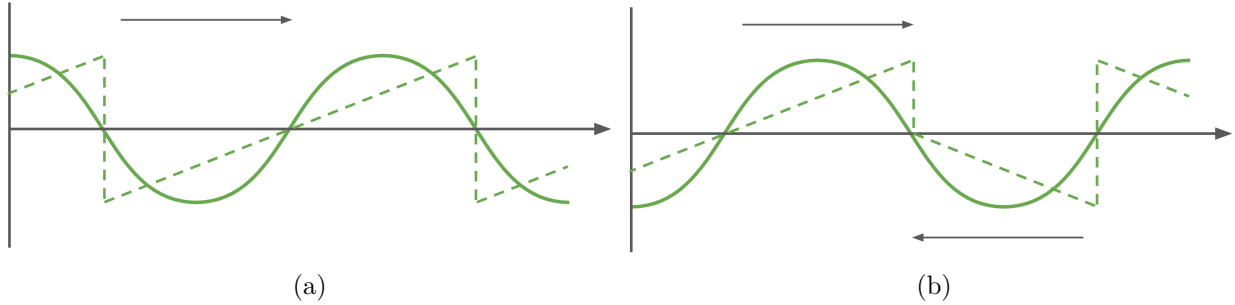


Figure 5: The figure shows a sinusoidal pressure wave, and two extreme cases where non-linear effects turn the wave into a sawtoothed shape. The left image shows a transverse wave travelling to the right, and the right image shows a standing wave just before interference.

Acoustic streaming is another non-linear phenomena which may occur at the present conditions. Although the flow in the domain is oscillatory, due to changes in viscosity near the wall, non-oscillatory flow may be generated, this occurs in and around the boundary layer. For example, flow called Rayleigh streaming is one of these, which are standing vorticities, which can increase the heat transfer rate from a solid surface to the fluid. [18] [19] [20]

## 2.3 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a tool used for analysis of systems involving fluid flow, heat transfer and corresponding phenomena like reaction mechanisms. With starting point in the aerospace and aeronautics industry, CFD has grown to be a well-established tool for development and validation in numerous other fields including chemical, nuclear, biomedical, electronics and building industries. The development of CFD has required more time than other computer-aided engineering tools such as the Finite Element Analysis (FEA) used for solid mechanics. The reason for this is the severe complexity of equations that need to be solved and their underlying behaviour. [21] [22]

The goal of using CFD is to acquire numerical approximations to the governing equations. To do so, the first step is to obtain the partial differential equations describing the flow. The equations are then discretized and the domain is divided into smaller elements or grids. Lastly, the equations are solved with the help of boundary-and initial conditions. CFD-codes are organized so that fluid flow problems can be addressed by the numerical algorithms. The workflow of a CFD analysis

contains three main elements being: (1) Pre-processing for input of the geometry, generating the mesh, defining flow parameters and boundary conditions, (2) solving the flow by discretization and iteratively solving the governing equations for the conditions provided and (3) post-processing to visualize the data. [23] [21]

In the present work, the software used is OpenFOAM. OpenFOAM is an open-source software widely used in both academia and industry, allowing the user to freely use and modify a high-end CFD code. The version used is OpenFOAM 10. [24]

## 2.4 Governing Equations

The governing equations of heat transfer and fluid flow are based on the conservation laws of physics and are referred to as the Navier-Stokes equations. They are mathematical statements derived from the conservation of mass, the conservation of momentum and the conservation of energy. These equations govern the motion of fluids as well as heat transfer, and how that changes in space and time [21]. Since fluid flows typically are three-dimensional and unsteady, the Navier-Stokes equations are non-linear second order partial differential equations. [22]

### 2.4.1 Conservation of mass

The conservation of mass says that a domain will conserve its mass on a local level if there are no mass sources or sinks. In a mass balance, this translates to the fact that the rate of increase of mass is equal to the net rate of flow of mass into the fluid element. The conservation of mass can be described as

$$\frac{\partial(\rho)}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (9)$$

where  $\rho$  is the density of the fluid and  $\mathbf{v}$  is the velocity vector of the fluid. This means that the first term describes the rate of change in density in time. The second term, referred to as the convective term, describes the net outflow of mass. The convective term appears in all governing equations with minor alterations. The continuity equation looks different if the flow is assumed to be incompressible: that is, if pressure changes do not effect density and the density therefore can be considered constant. If that is the case, the density cannot be computed from the conservation of mass, and what is left is only the criteria that the divergence of the velocity field is zero. However, in the present work the flow is considered compressible. [22] [21]

### 2.4.2 Conservation of momentum

Based on Newton's second law, the conservation law of momentum proves that a body maintains its total momentum in the absence of external forces acting on it. Consequently, the change of momentum over time will be equal to the sum of forces acting on the body. It is important to note that momentum is a vector quantity, and therefore its directional components will be conserved as well. When expressing the momentum equation it is common to separate contributions from surface forces and contributions from body forces. Surface forces are for example pressure forces or viscous forces while the gravitational force is considered a body force. The conservation of momentum can be described as

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v}) = -\nabla p + \nabla \cdot \mathbf{S} + \rho \cdot \mathbf{f}_b \quad (10)$$



where  $p$  is the pressure,  $\mathbf{S}$  is the viscous stress tensor and  $\rho \cdot \mathbf{f}_b$  is the body force term. The first term on the left-hand side describes the rate of change in momentum in time and the second term is again the convective term, here describing the total momentum travelling through the volume. [22] [21]

Investigating the right-hand side of equation 10 shows a compact version of the momentum equation. The first term  $-\nabla p$  is the sum of the pressure forces acting on the surface, and the negative sign is a consequence of it being a compressive normal stress by definition. The second term  $\nabla \mathbf{S}$  describes the viscous stresses acting on the fluid. [21] The fluid in the present work is assumed to be Newtonian, and with that assumption the viscous stress tensor is given by

$$\mathbf{S} = \beta(\nabla \cdot \mathbf{v})\mathbf{I} + 2\mu\mathbf{D} \quad (11)$$

where  $\beta$  is the bulk viscosity,  $\mathbf{I}$  is the unity tensor,  $\mu$  is the dynamic viscosity and  $\mathbf{D}$  is the symmetric rate of strain tensor, defined as  $\mathbf{D} = \frac{1}{2}(\nabla \mathbf{v} + \nabla \mathbf{v}^T)$ . By further using the Stokes assumption the bulk viscosity can be written as a function of the dynamic viscosity  $\mu$ :  $\beta = -\frac{2}{3}\mu$ . [25]

### 2.4.3 Conservation of energy

The first law of thermodynamics lays the foundation for the conservation of energy; energy can neither be created nor destroyed during a process. This means that energy can only change from one form to another, and consequently the sum of all different forms of energy in a system remains constant if isolated. For compressible flows, the energy variable used in the equation is commonly  $E$  which is the sum of the internal energy  $e$  and the kinetic energy, defined as  $E = e + \frac{1}{2}\mathbf{v}^2$ . With this, the conservation of energy can be expressed as

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (\rho \mathbf{v} E) = \nabla \cdot (-p\mathbf{v} + \mathbf{S}\mathbf{v} - k\nabla T) + \rho\sigma \quad (12)$$

where  $E$  is the total energy as per the definition above,  $\mathbf{S}\mathbf{v}$  is the work due to viscous stresses,  $k$  is the thermal conductivity and  $\sigma$  describes energy source per unit mass. On the left hand side of the energy equation the change of energy in time and the convective term appears, similar to equation (9) and (10). As the the rate of change in total energy is equal to the rate of heat addition and work extraction, the viscous term  $\mathbf{S}\mathbf{v}$  on the right hand now describes the viscous work done by the velocity gradient. The definition stated for the viscous tensor  $\mathbf{S}$  in equation (11) still stands, but in a context of work one could interpret it as friction inside the fluid generating heat. The third term comes from Fourier's law defined previously in equation (1). The last term on the right hand side,  $\rho\sigma$  is the volumetric energy source, which can be thought of as the energy created internally in the fluid. [25] [21] [22]

## 2.5 Turbulence modelling

In engineering applications, all fluid flows become unstable above a certain Reynolds number. Flows in the laminar regime, where viscous forces dominate, are described as stable and the Navier-Stokes equation can in simple cases be solved analytically. Turbulent flows however, are chaotic with continuous changes in pressure and velocity with time. This is a random and three-dimensional phenomena, bringing additional challenges when solving turbulent flow problems which need to be handled carefully. Characteristically, the diffusive nature of turbulence gives rise to effective mixing of the flow as well as rates of heat, mass and momentum transfer. Because of its unsteady characteristics, statistical methods are preferred over a predetermined description. [26] [21]

Though intensively studied, there is no known way to analytically solve the Navier-Stokes equations for the turbulent case [27]. Turbulence causes large gradients in the flow, which are manifested as rotational flow structures of various length scales called turbulent eddies. The large eddies extract kinetic energy from the mean flow, a process that is called vortex stretching. These eddies contain the majority of the turbulent kinetic energy in the system, around 80% [28]. Throughout the vortex stretching, angular momentum is conserved, which further causes the rate of rotation to increase while the eddies are broken down to smaller eddies. As this is happening, the kinetic energy dissipates from large eddies to progressively smaller ones. This phenomenon is termed the energy cascade, where kinetic energy is eventually converted into thermal internal energy due to viscous effects. The smallest scales of eddies present in turbulent flow are called Kolmogorov microscales and it is the energy associated with these that are lastly converted into heat. [26] [21]



Figure 6: Cascading effect, where large eddies turn into smaller eddies.

In CFD applications, it would be unreasonably expensive to resolve everything in the flow down to Kolmogorov scale. The solution to still being able to incorporate the effects of turbulence into simulations is to make use of turbulence models. There exists a significant amount of models to choose from, and the methods can be grouped into three categories: RANS (Reynolds-Averaged Navier-Stokes), LES (Large Eddy Simulation) and DNS (Direct Numerical Simulation). For RANS, the aim is to obtain equations for the mean flow while not resolving temporal fluctuations, based on statistical methods and time-averaging. The computational cost for RANS is low compared to the other methods, and therefore it has been the cornerstone of engineering flow calculations over the past decades. It is a method widely used in industry since one can obtain results faster and with fewer resources. DNS is the very opposite of RANS, computing the mean flow and all velocity fluctuations. Using DNS, the Navier-Stokes equations are solved on grids fine enough to resolve everything down to Kolmogorov scale. Understandably, this method is very costly, and therefore not normally used in industry. There is a midway between these two categories, LES, which is used in the present work. The reason for this is to obtain more detailed information from the flow behaviour than can be acquired from a RANS simulation, while not using unreasonable amounts of computational power. [21]

### 2.5.1 LES

LES is a method based on spatial low-pass filtering of the Navier-Stokes equations before they are computed. The approach was first proposed by Smagorinsky as early as 1963 [29]. The idea is that the larger eddies pass the filtering while the smaller eddies are rejected. This means that the resolved flow consists of the mean flow and the large eddies, while the small eddies are unresolved and need to be modelled. The information from the smaller filtered-out eddies is destroyed during the the spatial filtering. Interaction effects between the smaller and larger eddies cause SGS (sub-grid scale) stresses, which impact is best described by an SGS model. The scale of what will be filtered out or not can be described by the assigned eddy size  $\Delta$ . It defines the minimum size for which eddies will still be resolved, and is usually the same as the mesh cell size. It is obvious that it would be pointless to choose a  $\Delta$  smaller than the mesh cell size as all finer detail than this is lost anyway. [28] [21]

A filter function defining the spatial filtering operation can be described for any variable  $\phi$ . Each variable can be decomposed into a filtered, resolved component  $\tilde{\phi}$ , and a fluctuating, unresolved component  $\phi''$  as  $\phi = \tilde{\phi} + \phi''$ . The filtering function  $G(\mathbf{x}, \mathbf{x}', \Delta)$  used for finite volume implementations of LES is defined by Versteeg and Malalasekera as:

$$G(\mathbf{x}, \mathbf{x}', \Delta) = \begin{cases} \frac{1}{\Delta^3} & \text{if } |\mathbf{x} - \mathbf{x}'| \leq \frac{\Delta}{2} \\ 0 & \text{if } |\mathbf{x} - \mathbf{x}'| > \frac{\Delta}{2} \end{cases} \quad (13)$$

where  $\mathbf{x}$  is the location vector,  $\mathbf{x}'$  is the location vector for the fluctuating components and  $\Delta$  is the previously mentioned minimum resolved eddy size. [21]

This filter can then be applied to any variable  $\phi$  by the use of

$$\bar{\phi}(\mathbf{x}, t) = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} G(\mathbf{x}, \mathbf{x}', \Delta) \phi(\mathbf{x}', t) dx'_1 dx'_2 dx'_3 \quad (14)$$

Here it also necessary to introduce Favre-averaging which defines  $\tilde{\phi} = \frac{\bar{\rho}\phi}{\bar{\rho}}$  [30]. After applying the filtering to the governing equations stated in 9-12 the filtered Navier-Stokes equations are obtained as:

$$\begin{cases} \frac{\partial \bar{\rho}}{\partial t} + \nabla \cdot (\bar{\rho} \tilde{\mathbf{v}}) = 0 \\ \frac{\partial (\bar{\rho} \tilde{\mathbf{v}})}{\partial t} + \nabla \cdot (\bar{\rho} \tilde{\mathbf{v}} \otimes \tilde{\mathbf{v}}) = -\nabla \bar{p} + \nabla \cdot (\tilde{\mathbf{S}} - \mathbf{B}) + \bar{\mathbf{F}}_b \\ \frac{\partial (\bar{\rho} \tilde{E})}{\partial t} + \nabla \cdot (\bar{\rho} \tilde{\mathbf{v}} \tilde{E}) = \nabla \cdot (-\bar{p} \tilde{\mathbf{v}} + \tilde{\mathbf{S}} \tilde{\mathbf{v}} + \overline{k \nabla T} - \mathbf{b}) + \bar{\rho} \tilde{\sigma} \end{cases} \quad (15)$$

where  $\bar{\phantom{x}}$  denotes filtered and  $\tilde{\phantom{x}}$  denotes Favre filtered. Looking at equations 15 it is obvious that the only change from the unfiltered Navier-Stokes equations 9-12 is the addition of the sub-grid stress tensor  $\mathbf{B}$  and the sub-grid scale energy flux  $\mathbf{b}$ . These terms comes from the filtering of the non-linear convective term. They both represent the effects from the unresolved scales and must therefore be approximated, as these effects have an influence on the resolved scales. This is done by using sub-grid scale models. [21] [26]

### 2.5.2 Sub-grid scale modelling

As mentioned in the above section, the approach of LES is to directly resolve the large eddies while relying on modelling for the smaller, the sub-grid scale, eddies. The small scales are actually assumed to be more isotropic than the larger scales, further motivating this approach [31]. There are

many key SGS-models in use, as for example the Smagorinsky model [29] [32], the Wall-adapting local eddy-viscosity (WALE) model and the k-equation model [33]. In this work the k-equation model and the WALE model is used. The k-equation model will here be described more in depth.

To begin with, the sub-grid stress tensor  $\mathbf{B}$  and the sub-grid scale energy flux  $\mathbf{b}$  that showed up in equation 15 need to be defined as:

$$\begin{cases} \mathbf{B} = \overline{\rho(\mathbf{v} \otimes \mathbf{v})} - \bar{\rho} \tilde{\mathbf{v}} \otimes \tilde{\mathbf{v}} \\ \mathbf{b} = \overline{\rho(\mathbf{v}E)} - \bar{\rho} \tilde{\mathbf{v}} \tilde{E} \end{cases} \quad (16)$$

After this the sub-grid kinetic energy can be defined as  $k_{SGS} = \frac{1}{2}\text{tr}(\mathbf{B})$ . To then proceed towards fully closing the filtered Navier-Stokes equations, Boussinesq's hypothesis can be used to approximate  $\mathbf{B}$  and  $\mathbf{b}$ :

$$\begin{cases} \mathbf{B} \approx \frac{2}{3}\bar{\rho}k_{SGS}\mathbf{I} - 2\mu_k\tilde{\mathbf{D}}_D \\ \mathbf{b} \approx \frac{\mu_k}{Pr_t}\nabla T \end{cases} \quad (17)$$

where  $\mu_k$  is the SGS viscosity and  $Pr_t$  is the turbulent SGS Prandtl number, defined as the ratio between turbulent SGS viscosity and turbulent SGS diffusivity. The k-equation model solves an additional transport equation to estimate the kinetic energy [34]. The transport equation can be derived and is stated as:

$$\frac{\partial(\bar{\rho}k)}{\partial t} + \nabla \cdot (\bar{\rho}k\tilde{\mathbf{v}}) = -\mathbf{B} : \tilde{\mathbf{D}} + \nabla \cdot (\mu_k\nabla k) - c_\varepsilon \bar{\rho} \frac{k^{3/2}}{\Delta} \quad (18)$$

where  $\mu_k$  is again the SGS viscosity, defined as  $\mu_k = c_k\bar{\rho}\Delta k_{SGS}^{1/2}$  both  $c_k$  and  $c_\varepsilon$  are model coefficients. They are defined as  $c_k = 0.05$  and  $c_\varepsilon = 1.00$  [35].[26] [21]

### 2.5.3 Wall modelling

A closely tied topic to sub-grid modelling and of significant importance is wall modelling. Like sub-grid models it uses mathematical modelling to replicate the equivalent results of fully resolving sub-grid phenomena. When approaching the near wall region the boundary layer can be found. [36]

The boundary layer is more complicated than the larger bulk flow of the region and in order to simulate it properly. The mesh needs to be very fine, especially in the direction normal from the surface because the boundary layer is very thin. This increase in cell count raises the computational cost of the simulation significantly. In order to avoid this, wall models have been developed. The models are not as accurate, but are often accurate enough for engineering purposes. [37]

Boundary models can be categorized in many different ways, but it is common to differentiate between standard and continuous models. To understand the difference it is important to better understand the structure of the boundary layer.

This necessitates the introduction of the terms  $y^+$  and  $u^+$ , which are non-dimensional measures of distance from the wall and flow velocity based on the actual flow velocity, fluid properties and

surface smoothness. It is used to help describe different parts of the boundary layer with different types of flow. They are defined below. [37] [36]

$$y^+ = \frac{u_\tau y}{\nu} \quad (19)$$

$$\tau_w = \rho u_\tau^2 \quad (20)$$

$$u^+ = \frac{u}{u_\tau} \quad (21)$$

$u_\tau$  is the friction velocity,  $y$  is the distance from the wall,  $\nu$  is the kinematic viscosity in equation (19).  $\tau_w$  is the wall shear stress in equation (20). In (21)  $u$  is the flow velocity parallel to the wall. Wall shear needs to be estimated, and the formula often used is (22), where  $C_f$  is the skin friction coefficient, and  $U_f$  the free stream velocity. [37] [38] [39]

$$\tau_w = C_f \cdot \frac{1}{2} \rho U_f^2 \quad (22)$$

Using this new dimensionless distance and the near wall flow can be more easily described, what is often called the law of the wall.

The boundary layer can be either be turbulent or laminar. When it is laminar the flow profile is quite simple in nature and there is no need for modelling, and a simple wall shear stress can be calculated, left in figure 7.

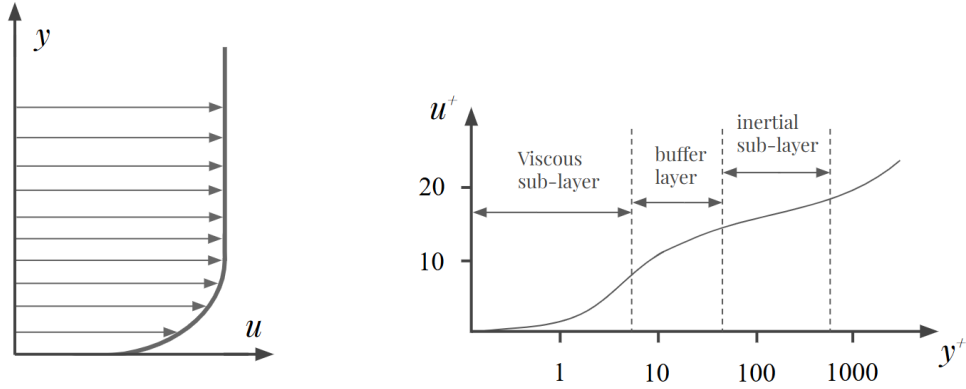


Figure 7: Boundary layer for laminar flow (left) and for turbulent flow (right). [36]

On the right side of figure 7 the turbulent boundary layer is described. The standard boundary layer model calculates the  $y^+$  value of the nearest cell centre from the wall and checks in which regime it is located. If its in the viscous sub-layer then the boundary layer is resolved and no adjustments are needed. But if the cell centre is in the inertial sub-layer or close to it then the turbulent viscosity term  $\nu_t$  is adjusted using (23), which approximates the curve in this range in figure 7. [37]

$$u^+ = \frac{1}{\kappa} \ln(y^+) + B \quad (23)$$

The other common method is the continuous approach, while instead of evaluating the viscous turbulence term based on if the cell centre is located in a certain flow regime, it is evaluated continuously using a function as  $y^+$  goes toward zero  $y^+ \rightarrow 0$ . An example of this is the Spalding wall

model. [37] [40]

These two differentiating approaches are of many variations to modelling the flow in the region near the wall and more can be said, such as how to handle rougher wall surfaces, but that is a topic for another paper.

### 3 Numerical Theory

To be able to solve the governing equations presented in the previous chapter, a linear function must be obtained. In this chapter, the numerical theory that is used to discretize the governing equations is presented.

#### 3.1 Meshing

Before the equations 9-12 can be discretized, a mesh with a number of three dimensional cells must be created. If a dependent variable  $\phi$  is considered, the numerical solution of a partial differential equation consists of finding the values of  $\phi$  at specific points from which its distribution over the domain can be described. These points result from the discretization of a geometry into a set of discrete elements. By relating the values at neighbouring cells to each other, the discrete values of  $\phi$  are computed. The discretization of the geometry results in a mesh, which can be accomplished by a range of different techniques resulting in different types of meshes. Mesh structure, cell shape and orthogonality are examples on different characteristics that are used for classification of meshes. Two different mesh types that can be distinguished here are unstructured and structured meshes, where the difference lies in how neighbouring cells are accessed. [22] [24]

Each cell stores a volume-averaged value of  $\phi$  in the centre of the cell, and this value is here referred to as the nodal value  $\phi_P$ . A mesh cell is bounded by faces that are shared with the neighbouring cells, except at the boundaries of the domain. As the very purpose of the domain discretization is to be a useful platform for the equation discretization, geometric information about the cells must be inferred from the mesh data. In general this compromises element to element relations or face to element relations, but more examples are element volume and normal direction. The interior faces between cells are derived from the geometric information, but faces coinciding with the domain boundary must be prescribed. These faces can be grouped into different patch types. Boundary conditions are then applied to the patches, for each of the fields separately.[24] [22]

A boundary condition is what is applied to a part of the edge of the domain and handles the conditions that applies to flow as it either exits, enters or collides with walls. Unlike flow in a central part of the domain which is handled by the governing equations and their discretization, the boundaries also need to take into account how the different scalars and fields interact with what is outside the domain. This is often coupled with a desire to mimic some kind of real world equivalent. For example, to mimic the outlet of an air duct into a room in a ventilation system, the equivalent of ambient pressure and temperature should probably be applied. However, the velocity of the air flow out of the duct is not usually dictated by what is happening in the room, but rather further up in the duct. A zero gradient condition is therefore applied, meaning that the same speed is applied on the edge of the domain as just inside of this edge. [41] [37]

#### 3.2 The Finite Volume Method

The Navier-Stokes equations 9-12 described in the Theory-section above require approximations because of their non-linearity. There is a need to transform these partial differential equations into linear functions over elements. This can be done by using the Finite Volume Method (FVM). It is a highly flexible discretization model because the discretization is done straightforwardly in the physical space, and it is frequently used for CFD applications. [22] [21]

The FVM is best described with an example. Consider a generic conservation equation for a

general scalar  $\phi$ :

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \phi \mathbf{v}) = \nabla \cdot (\Gamma^\phi \nabla \phi) + S^\phi \quad (24)$$

in this equation the following terms show up: the transient term  $\frac{\partial \rho \phi}{\partial t}$ , the convective term  $\nabla \cdot (\rho \phi \mathbf{v})$ , the diffusive term  $\nabla \cdot (\Gamma^\phi \nabla \phi)$  and the source term  $S^\phi$ . The diffusion coefficient  $\Gamma^\phi$  is of arbitrary dimension. After this, equation (24) is integrated over the control volume, in this case a mesh cell:

$$\int_{\delta V_c} \frac{\partial \rho \phi}{\partial t} dV_c + \int_{\delta V_c} \nabla \cdot (\rho \phi \mathbf{v}) dV_c = \int_{\delta V_c} \nabla \cdot (\Gamma^\phi \nabla \phi) dV_c + \int_{\delta V_c} S^\phi dV_c \quad (25)$$

The transient term can be rewritten in the following way as it is the volume average of the time derivative:

$$\int_{\delta V_c} \frac{\partial \rho \phi}{\partial t} dV_c = \delta V_c \frac{\partial \rho_p \phi_p}{\partial t} \quad (26)$$

where values stored at the cell centre is denoted  $\phi_p$ . Both the convective and the diffusive term can be reformulated as integrals over the bounding surface of the control volume. This is done by the use of Gauss's divergence theorem. After the conversion, the convective and the diffusive term become:

$$\begin{cases} \int_{\delta V_c} \nabla \cdot (\rho \phi \mathbf{v}) dV_c = \int_{\delta A_c} \mathbf{n} \cdot (\rho \phi \mathbf{v}) dA_c \\ \int_{\delta V_c} \nabla \cdot (\Gamma^\phi \nabla \phi) dV_c = \int_{\delta A_c} \mathbf{n} \cdot (\Gamma^\phi \nabla \phi) dA_c \end{cases} \quad (27)$$

Here the  $A_c$  is the surface of the cell and  $\mathbf{n}$  is the surface normal vector. The last term in equation (24), the source term, represents the mean source term in the cell. This is computed directly from other mean quantities, since the integration is not numerically executed. After applying these alterations to equation (25) the following is obtained:

$$\delta V_c \frac{\partial \rho_p \phi_p}{\partial t} + \int_{\delta A_c} \mathbf{n} \cdot (\rho \phi \mathbf{v}) dA_c = \int_{\delta A_c} \mathbf{n} \cdot (\Gamma^\phi \nabla \phi) dA_c + \delta V_c S^\phi \quad (28)$$

The next step is to integrate this equation over a finite time step  $\Delta t$ . After this, discretization schemes can be applied, which is described in the next section. [21]

### 3.3 Discretization Schemes

The numerical solution of PDE's is composed of finding values of the generic scalar  $\phi$  at specified points being grid elements. To be able to solve equation (28) numerically, the next step is to discretize the terms in space and time. There are many available discretization methods and the different terms in the equation will require different discretization schemes. The focus of all discretization methods however is to substitute the exact continuous solution of the PDE's with discrete values. This is done by solving algebraic equations derived from the corresponding conservation equation, and after this the data can be processed. [22]

#### 3.3.1 Time discretization scheme

The transient term of equation (28) involves a time derivative which must be handled by a time discretization scheme. A time stepping approach much be introduced, where an initial condition at time  $t_0$  is introduced as  $n$ . As the solution algorithm marches forward and finds a solution at a later time step, this time step is introduced as  $n + 1$ . This solution is the initial condition for the next time step and so on until the required time is reached. In the same manner,  $n - 1$  is introduced as



a representation of the previous time step. The time derivative of the generic variable  $\phi$  must be expressed using previous nodal values. When only values at the old time step is used to evaluate  $\phi$  at the new time step, the scheme used is explicit. When values at the new time step are used in combination with the old values, the scheme used is implicit instead. [22] [21]

In this work an implicit scheme is used for time discretization, the Euler scheme. The discretization is described as the following

$$\frac{\partial \rho_p \phi_p}{\partial t} = \frac{\rho_p^n \phi_p^n - \rho_p^{n-1} \phi_p^{n-1}}{\Delta t} \quad (29)$$

where  $\phi^{n-1}$  is the previous nodal value and  $\phi^n$  is the current nodal value as previously described. [21]

### 3.3.2 Gradient scheme

The gradients present in the governing equations must be discretized using a gradient scheme. Cell values are extrapolated to faces using the calculated gradient, and therefore the choice of gradient evaluation scheme is crucial. In this work, the linear interpolation scheme is used together with a limiter called *cellMDlimited*. This limits the gradients in the face normal direction so that the face values do not fall outside the bounds of values in neighbouring cells. The limiting is decided with a coefficient from 0 to 1, where 0 corresponds to no limiting and 1 corresponds to full boundedness, meaning that  $\phi$  will not show any forged oscillations. The Gauss linear scheme is based on the application of Gauss' divergence theorem, converting volume integrals to surface integrals. The scheme is explicit and second order accurate. The face values are obtained through the use of linear interpolation between neighbouring cell centres, and the gradient is then approximated as a sum of face fluxes. The approximation can be described as

$$\nabla \phi \approx \frac{1}{V} \sum \phi_f \mathbf{S}_f \quad (30)$$

where  $V$  is the cell volume,  $\phi_f$  is the value of  $\phi$  at each face and  $\mathbf{S}_f$  is here the face area vector composed of the outward normal vector and the face area. [42] [37] [21]

### 3.3.3 Divergence scheme

The divergence schemes are used to discretize the convective terms that are found in each of the equations (9)-(12), as well as in equation (28). Much research has been devoted to the discretization of the convective term due to the difficulty to achieve numerical stability and accuracy, and this has resulted in the development of a large number of divergence schemes [22]. In the present work, the *limitedLinear* scheme is used. This is a limited version of the linear scheme, that can tend towards either the linear or the upwind scheme based on a user input coefficient. The upwind scheme is fully bounded and first order accurate, while the linear scheme is unbounded and second order accurate. [42]

As for the previously discussed discretization schemes, the divergence schemes are based on Gauss integration. As opposed to the upwind scheme which uses the flow direction to express the face values, the linear scheme expresses them in terms of adjacent cells instead. This is done by linear interpolation, where a weight field  $w$  can be defined for each face according to

$$\phi_f = w \phi_P + (1 - w) \phi_N \quad (31)$$

where  $\phi_f$  is here the interpolated face value and  $\phi_N$  is the value of  $\phi$  at an arbitrary neighbouring cell. The linear scheme approximates  $\phi_f$  in accordance with a linear variation between the cell values  $\phi_P$  and  $\phi_N$ , so that the weight field  $w$  can be defined as

$$w = \frac{d_{fN}}{d_{PN}} \quad (32)$$

where  $d_{fN}$  is the distance between the cell face and the neighbouring cell center and  $d_{PN}$  is the distance between the main cell P and the adjacent cell N. [37]

### 3.3.4 Laplacian scheme

Terms including the Laplacian operator  $\nabla \cdot (\nabla \phi)$ , i.e the diffusive term containing the diffusion coefficient  $\Gamma^\phi$  in equation (28), is computed with the help of a Laplacian scheme. In this work, the scheme used is the *Gauss linear corrected* scheme. The surface normal gradient is a part of the Laplacian discretization, and this is also handled with a corrected scheme. The discretization for the diffusive term  $\mathbf{n}_f \cdot (\nabla \phi)_f$  with this choice of scheme is given by

$$\mathbf{n}_f \cdot (\nabla \phi)_f = \frac{\phi_N - \phi_P}{|d_{fN}| \cos \theta} + \left( \mathbf{n}_f - \frac{d_{fN}}{|d_{fN}| \cos \theta} \right) \cdot (\nabla \phi)_{f_{exp}} \quad (33)$$

where  $|d_{fN}|$  is the distance of the vector between the cell face and the adjacent cell N,  $\theta$  is the angle between  $d_{fN}$  and  $\mathbf{n}_f$  and  $(\nabla \phi)_{f_{exp}}$  is an explicit corrector term for the gradient. [37]

### 3.3.5 Courant Number

Stepping away slightly from numerical schemes, the practical discussions on setup further down in this paper requires some understanding of the most common stability criteria used for CFD. This criteria is known as the CFL condition. To fully understand it requires a good understanding of PDE's. However this thesis will not cover any theory of PDE's so a simplified explanation will be presented.

The Courant number or the CFL condition was developed in the late 1920's by Richard Courant, Kurt Friedrichs and Hans Lewy. It concerns the solution to partial differential equations and how the solution depends on propagation of information in a discretized domain. What the CFL condition states is that if the velocity travels further than the distance between two cell centres in the domain, then the solution cannot converge to the exact solution if this continues to be true. This means that fluid from one cell in a one dimensional domain should not travel further than the distance  $\Delta x$  in the time  $\Delta t$ . Thus for a PDE in a one dimensional domain the CFL condition can be expressed as below. [21] [43] [44]

$$u \frac{\Delta t}{\Delta x} \leq 1 \quad (34)$$

As mentioned previously, this is a simplification, and the Courant number does not always have to be less or equal to 1 to be stable. It depends on what type of PDE it concerns and therefore the type of simulation and the kind of physics being recreated. [21]

## 4 Computational Setup

Having established the theoretical and numerical framework, this chapter describes the computational implementation used for the simulations. The domain geometry, meshing strategy, and boundary conditions are all presented together with the discretization schemes and solver used.

### 4.1 Geometry

The geometry consists of two main parts, the cooling chamber and the resonance pipes. The cooling chamber sits between the two resonance tubes. Figure 8 shows two angles of the cooling machine. The internal geometry was extracted from this CAD model. The resonance pipes are simple hollow sections connected to each other, initially circular then transitioning to a rectangular cross section as it connects to the cooling chamber. The inside of the cooling chamber consists of an upper and lower part each equipped with cooling fins. The upper and lower section do not completely seal the chamber volume, instead leaving gaps on the side allowing a steel plate to be inserted into the standing wave. There are also two pipe outlets on the top and bottom parts. These outlets lead out into the open air. Throughout the different sections of the geometry the cross sectional area is equal. Simplified images of this internal volume can be seen in figure 9, showing side views and a top down view.

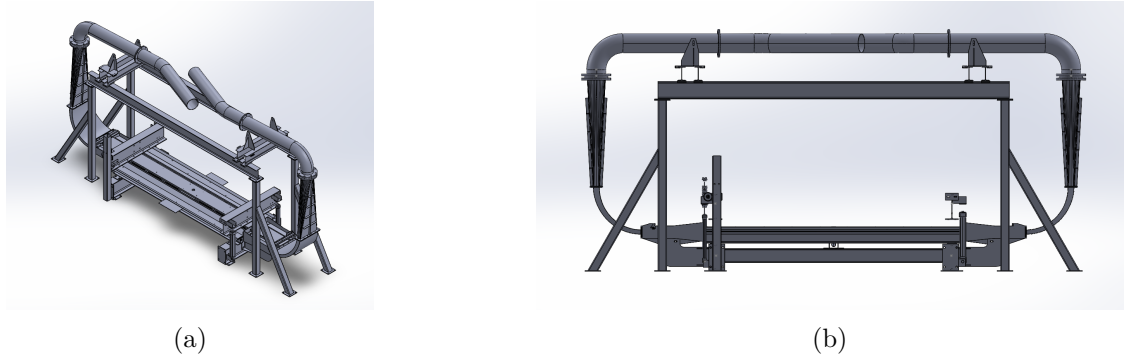


Figure 8: CAD model of the cooling machine. Image (b) measures 5,7 by 2,7 meters.

This specific geometry is tuned for a frequency of 10.7 Hz given by the total length of the system from tip to tail. The geometry can be further split into the coloured sections found in figure 9a. The ends of the resonance tubes are coloured yellow. This is where pulsators would be placed in the real life machine. In this model this is replaced by a purpose built boundary condition. The blue sections represents a circular to rectangular transition. The purple parts curves into the larger cooling area and is shaped much like a diffuser. The red coloured area is not part of the main cooling area, but contain cooling fins. The green section is the cooling box and is wider than previous sections, and also contains the two outlets previously presented. In the cooling box a steel plate is placed, which can be seen sticking out in figure 8a. The steel plate is represented in the internal volume as plate shaped hole.

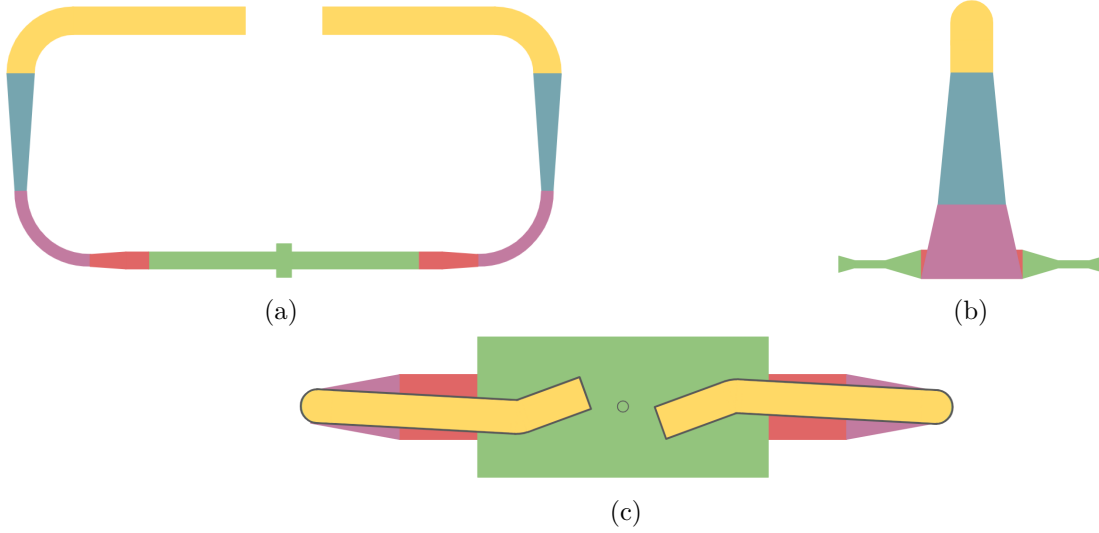


Figure 9: Simplified representation of the internal volume of the cooling system. Visualised at three angles and rotated using first angle projection.

Initially a very detailed internal volume was extracted from CAD files. This contained a number of gaps and thin volume slivers due to the construction based nature of the CAD files. These were removed to represent the internal volume after the tightening of joints screw joints and welding of parts.

Further changes to the volume were necessary in order to make sure the mesh quality was acceptable. The base geometry contains a number of very thin edges and narrow channels, which makes it difficult to build a mesh of good quality. A total of eight different geometries were built, each iteration were created to solve what was perceived as stability issues related to poor mesh quality. This was due to the complexity of the geometry coupled with a lack of resources. The total geometry is both large in size and contain small geometric features. The small features require small cell sizing to properly resolve them. This causes the total cell count to become quite large, and in this thesis a practical limit was reached, particularly with respect to preprocessing.

## 4.2 Mesh

Although meshing is always done after the geometry has been created, in this thesis project the process was iterative because the geometry is of a complex nature, keep this in mind while reading about the mesh.

Meshing was done using a tetrahedral unstructured approach in the Ansys Fluids and Structures 2022 R2. Initial meshes were of poor quality and had poor resolution. They had high maximum and average aspect ratios, resulting in early crashes. This was mostly due to cell sizing being much too large for the size of the geometry features. The cell size was quickly reduced to about 10 mm, which resulted in meshes of a much higher quality. As mentioned in the section on the geometry, some areas were difficult to accommodate but the quality still needed further improvement and so the cell size was reduced down to 3 mm. Throughout this process the meshing process was refined into using what are called bodies of influence as the main method. This lets the user specify meshing requirements for each volume. Figure 10 shows how they were applied. In each box specific size

requirements and growth rates were made. The size specification ensured small enough sizes to model the geometry while maintaining good cell quality, and the growth rate ensured that the size transitions between regions were smooth which improves stability. The specifics can be seen in table 1, and this is how the final meshes were generated. The metrics of the two meshes used can be viewed in table 2.

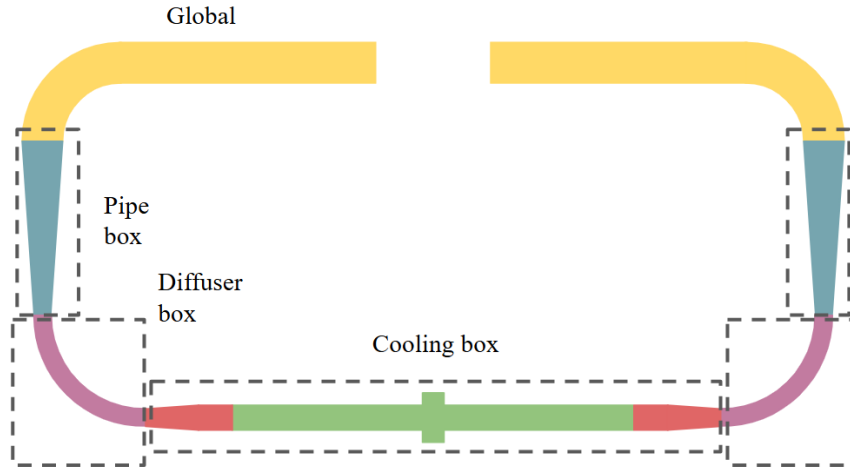


Figure 10: The simplified geometry, with bodies of influence inserted as dashed rectangles.

Table 1: Cell sizes and growth rate for each body of influence.

|                    | <b>Body of influence</b> |          |              |             |
|--------------------|--------------------------|----------|--------------|-------------|
|                    | Global                   | Pipe box | Diffuser box | Cooling box |
| Max cell size [mm] | 12                       | 10       | 8            | 3           |
| Growth rate [-]    | 1,2                      | 1,1      | 1,05         | 1,2         |

Table 2: Mesh quality metrics for the two final meshes. Based on values generated by OpenFOAM *checkMesh* command.

|       | <b>Mesh quality metrics</b> |              |                               |                           |
|-------|-----------------------------|--------------|-------------------------------|---------------------------|
| Mesh  | Max aspect ratio            | Max skewness | Average non-orthogonality [°] | Max non-orthogonality [°] |
| 300 K | 10.19                       | 0.89         | 16.01                         | 71.69                     |
| 700 K | 9.87                        | 1.15         | 16.02                         | 72.95                     |

### 4.3 Boundary Conditions

This simulation studies airflow through a series of pipes and boxes driven by pressure, although the physical models and equations implemented in this setup are quite ordinary, the resulting physics are quite unique and the boundary conditions containing it are as well. In tables 3 and 4 the type or model for each boundary condition are listed, as well as their corresponding values. What sets these boundary conditions apart is that each boundary that allows fluid to pass through, has flow in both directions, either at the same time or alternating over time. Usually a patch has flow in only

one direction, so this is uncommon, especially with all non wall boundaries having this requirement. In figure 11 all boundaries are named and highlighted for the readers convenience.

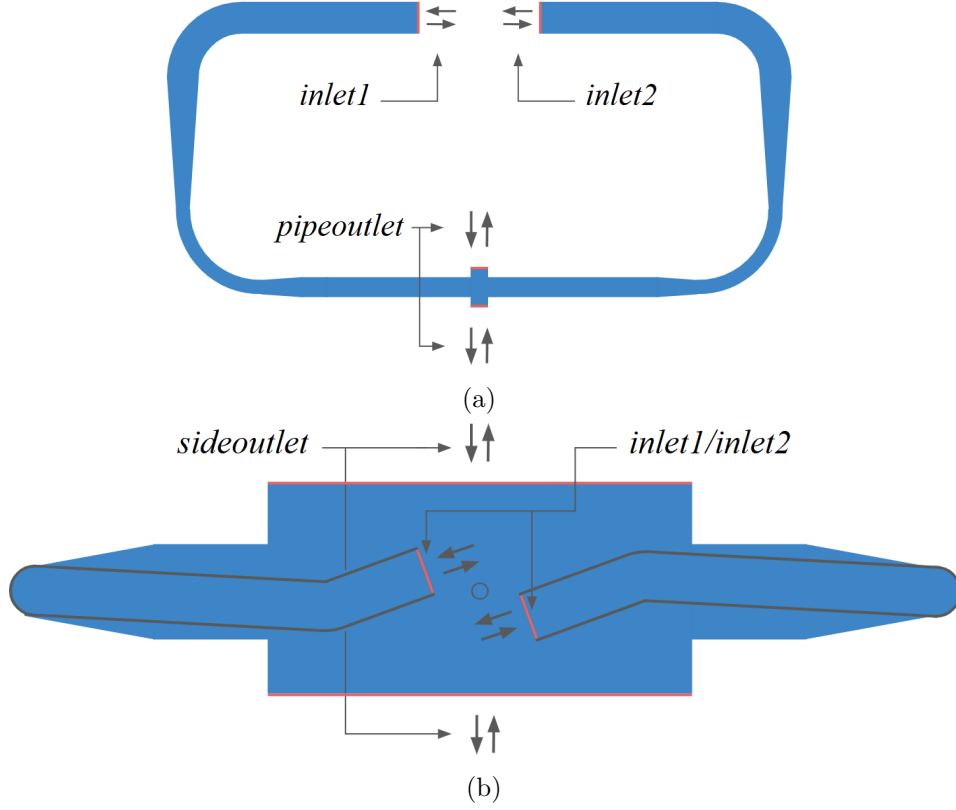


Figure 11: The simplified geometry with boundary surfaces highlighted with red and the potential flow directions shown with arrows.

Table 3: Boundary conditions and models. Values can be found in table 4.

| Patch             | Boundary condition function and type |             |                 |                        |                 |                          |
|-------------------|--------------------------------------|-------------|-----------------|------------------------|-----------------|--------------------------|
|                   | $U$                                  | $T$         | $p$             | $\alpha_{\text{phat}}$ | $k$             | $\text{nut}$             |
| <i>inlet1</i>     | PNIOV                                | inletOutlet | codedFixedValue | zeroGradient           | inletOutlet     | calculated               |
| <i>inlet2</i>     | PNIOV                                | inletOutlet | codedFixedValue | zeroGradient           | inletOutlet     | calculated               |
| <i>pipeoutlet</i> | PNIOV                                | inletOutlet | totalPressure   | zeroGradient           | inletOutlet     | calculated               |
| <i>sideoutlet</i> | PNIOV                                | inletOutlet | totalPressure   | zeroGradient           | inletOutlet     | calculated               |
| <i>wall</i>       | noSlip                               | fixed       | zeroGradient    | zeroGradient           | kqRWallFunction | nutUSpaldingWallFunction |
| <i>steelplate</i> | noSlip                               | fixed       | zeroGradient    | zeroGradient           | kqRWallFunction | nutUSpaldingWallFunction |

Table 4: Values applied to corresponding models and conditions from table 3.

| Boundary condition values |                  |       |                          |                        |                                      |                          |
|---------------------------|------------------|-------|--------------------------|------------------------|--------------------------------------|--------------------------|
| Patch                     | $U$              | $T$   | $p$                      | $\alpha_{\text{phat}}$ | $k$                                  | $\nu_{\text{t}}$         |
| <i>inlet1</i>             | $\nabla U = 0^*$ | 300 K | $P_A \sin(2\pi f)^{**}$  | -                      | $2\text{e-}5 \text{ m}^2/\text{s}^2$ | $0 \text{ m}^2/\text{s}$ |
| <i>inlet2</i>             | $\nabla U = 0$   | 300 K | $P_A \sin(2\pi f + \pi)$ | -                      | $2\text{e-}5 \text{ m}^2/\text{s}^2$ | $0 \text{ m}^2/\text{s}$ |
| <i>pipeoutlet</i>         | $\nabla U = 0$   | 300 K | 101 325 Pa               | -                      | $2\text{e-}5 \text{ m}^2/\text{s}^2$ | $0 \text{ m}^2/\text{s}$ |
| <i>sideoutlet</i>         | $\nabla U = 0$   | 300 K | 101 325 Pa               | -                      | $2\text{e-}5 \text{ m}^2/\text{s}^2$ | $0 \text{ m}^2/\text{s}$ |
| <i>wall</i>               | $U = 0$          | 300 K | $\nabla p = 0$           | -                      | $\nabla k = 0$                       | -                        |
| <i>steelplate</i>         | $U = 0$          | 700 K | $\nabla p = 0$           | -                      | $\nabla k = 0$                       | -                        |

\*For outflow    \*\* $P_A = 25 \text{ kPa}$ ,  $f = 10.7 \text{ Hz}$

The flow in this model is pressure driven, so extra attention should be paid here. The boundaries *inlet1* and *inlet2* represent the pulsators as seen in figure 11a. They force a displacement of the fluid and generate a standing wave. This was simplified by applying sinusoidal pressure boundary conditions which instead generate the standing wave. In figure 12 the code which handles the boundary for *inlet1* and *inlet2* can be found. The two boundary conditions are identical, however the condition for *inlet2* is shifted by half a wave by adding  $\pi$  to time in the operator variable, causing one to pull while the other pushes.

```

inlet1
{
    type                codedFixedValue;
    value                uniform 101325;
    name                 sinusPressure1;

    code
    #{
        scalar pMean = 101325;
        scalar Amp    = 25000;
        scalar f      = 10.7;
        scalar omega  = 2*M_PI*f;

        scalar t = this->db().time().value();

        operator==(pMean + Amp*sin(omega*t));
    #};
}

```

Figure 12: Coded boundary condition for *inlet1*.

This thesis ran two different simulations, one with a plate at room temperature and one with a hot plate. The simulation with a hot plate introduced the boundary patch *steelplate*, in order to increase its wall temperature. Initial attempts used the one dimensional conjugate heat transfer function, but this was discarded for a simpler fixed temperature of 700 K. The *steelplate* boundary had its temperature increased, and the k-equation sub-grid model was swapped for the WALE model.

## 4.4 Solvers

The flow in these simulations handle nonlinear pressure phenomena which involve compressed sections of air. This requires a solver that includes compressibility. The simulation is also transient, this combination means that the *rhoPimpleFoam* solver was used. It is based on the Pimple foam solver but redesigned for compressibility. Pimple foam or PISO-SIMPLE is a combination of the two solvers PISO and SIMPLE and is a transient solver for incompressible flow. [45]

## 4.5 Schemes

The numerical schemes were selected in a practical manner. The simulation setup is not inherently stable and requires careful numerical handling. Table 5 shows each scheme used. Most are bounded or limited in some nature in order to introduce diffusivity and create requisite stability. Initially first order schemes were used for the divergence schemes, but as the simulation matured past initial difficulties it was possible to use higher order divergence schemes to more accurately capture the physics.

Table 5: Schemes and their correctors that are used in the simulation setup. [42]

| Numerical schemes applied |                  |                |                   |             |
|---------------------------|------------------|----------------|-------------------|-------------|
| Category                  | Scheme           |                | Order             | Boundedness |
| Time                      | Euler            |                | 1st order         | Bounded     |
| Gradient                  | cellMDLimited    | Gauss linear 1 | 2nd order limited | Bounded     |
| Surface normal gradient   | corrected        |                | 2nd order         | -           |
| Divergence                | limitedLinear 1  |                | 2nd order         | -           |
| Laplacian                 | linear corrected |                | -                 | -           |
| Interpolation             | linear           |                | 2nd order         | Unbounded   |

## 4.6 Model evaluation

The usual method of evaluating a computational setup is by comparing the results of the simulation to corresponding experimental results. The experimental results that exist for this particular setup are temperature measurements from a steel plate similar to the one in this setup, as it is cooled down in the cooler. This results in a temperature curve which unfortunately cannot be used as validation for this setup.

The total simulation time in this simulation setup is about 1 second. Much too short a time to be able to compare cooling curves, when the expected cooling for that amount of time is a couple of degrees. This means the computational setup requires a different kind of evaluation, of a more qualitative than quantitative nature.

There are other solvers which enable compressibility in the OpenFOAM library. Primarily *rhoCentralFoam*, which is a density based solver, using central differencing. However, it is more sensitive than *rhoPimpleFoam* and is more time consuming as it is density based. For this specific project the simulations have been very time consuming, suggesting that *rhoPimpleFoam* is the appropriate choice. [46] [47] [48] [49]

Choosing a turbulence model was deemed a less inhibiting choice. It is not uncommon that other numerics dominate more than choice of turbulence model [50] [51]. As such the k-equation or



one equation was chosen because it is simple to implement, fairly robust and decently accurate. However when changing the wall temperature of the steel plate to 700 K, the simulation became quite unstable. This was solved by switching sub-grid model to the WALE model.

## 5 Results

In this chapter a selection of the simulation results will be presented. There are two sections which corresponds to the simulation results for the two different simulations. One for the steel plate at an ambient temperature of 300 K, and one for the 700 K steel plate.

### 5.1 Ambient temperature steel plate - 300 K

To get an overview of how the pressure waves propagate through the domain during a cycle, it can be helpful to look at a cross section of the whole domain. In figure 13, the pressure distribution at different time steps through one period of an infrasound cycle is shown. This is done with a sliced plane normal to the y-axis, so that the global behaviour of the pressure going from one side to the other is captured. In figure 13a the left side is the high pressure side and the right side is the low pressure side. As the pressure wave progresses, the higher pressure zone moves through the cooling chamber and approaches the resonance tube on the other side, as seen in figure 13b and figure 13c. At  $t = 0.550$  s the opposite of figure 13a is depicted, where the high pressure and the low pressure sides are switched in figure 13d.

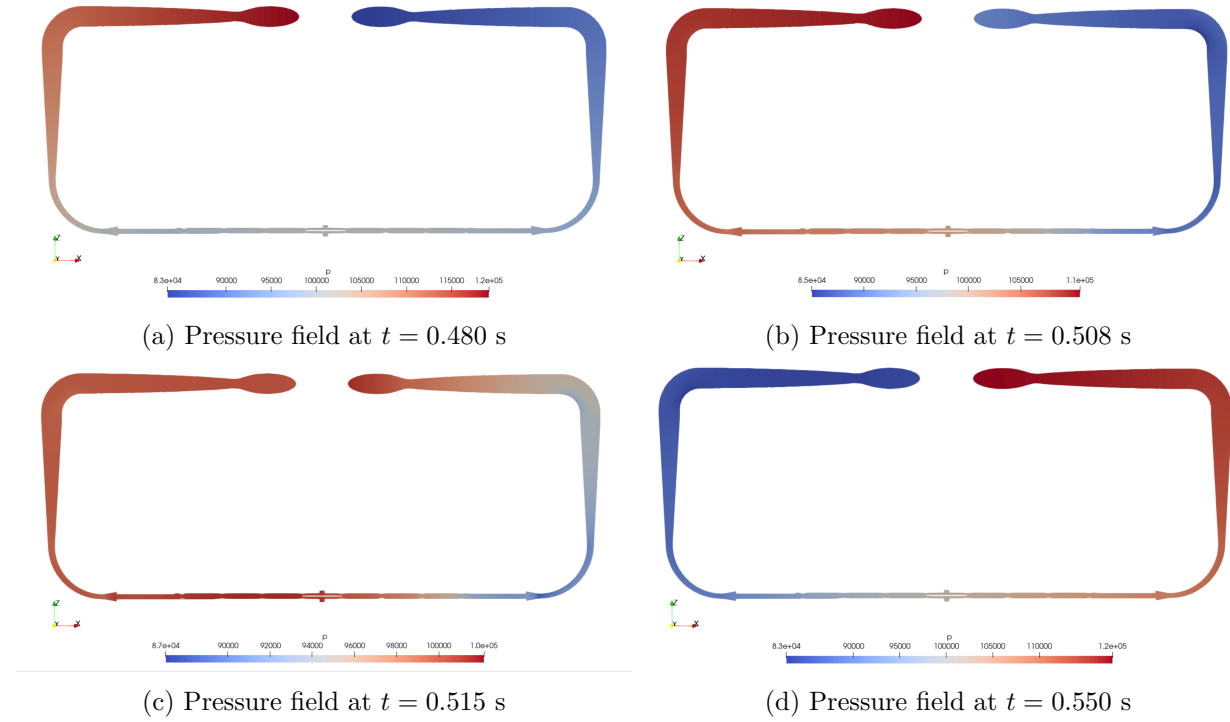


Figure 13: Pressure field of a central slice in the y-normal direction, at time steps 0.480 - 0.550 s.

To further build on the overall understanding of the flow field before going more into detailed phenomena, the velocity field for the same time steps as in figure 13 is shown in figure 14 below. It can be noted that figure 14a and figure 14b are taken at time steps during the first half of the cycle, and show the same trends as figure 14c and figure 14d that are taken from the second half of the cycle, but that the behaviour is mirrored as is expected based on the theory of pressure and particle velocity being  $90^\circ$  out of phase.

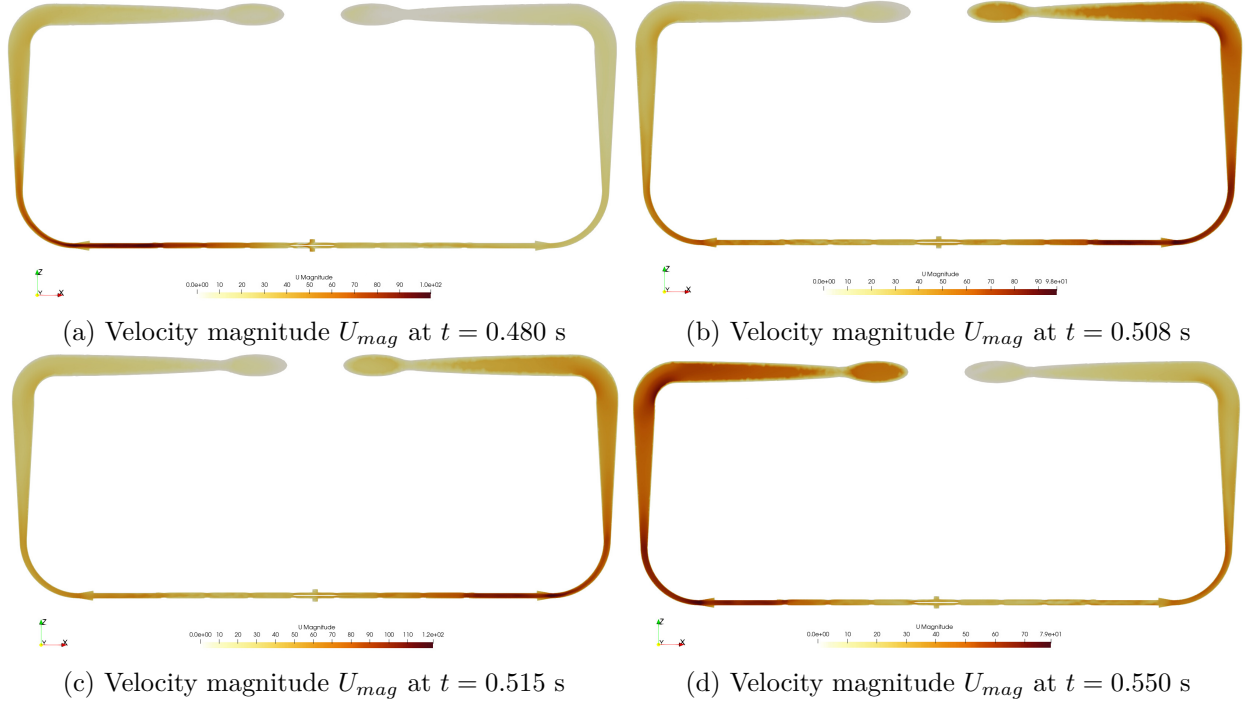


Figure 14: Velocity magnitude field  $U_{mag}$  of a central slice in the y-normal direction, at time steps 0.480 - 0.550 s. Note how the colour gradient shifts with the maximum and minimum value.

Figure 15 shows pressure as a function of time and distance. Both are based on line plotting, but uses a surface representation and a colour based time depiction respectively. It shows the establishment of the standing wave from time step 0 to 0.532 seconds. The points of the line plot follow a central line from pulsator to pulsator, deviating from the centre at the middle where it passes above the steel plate. The blue coloured lines in figure 15b show the early time steps, and the graph shows how they differentiate slightly from later values which are more consistent. The lines are also quite jagged at points, where the pressure wave interacts with the geometry.

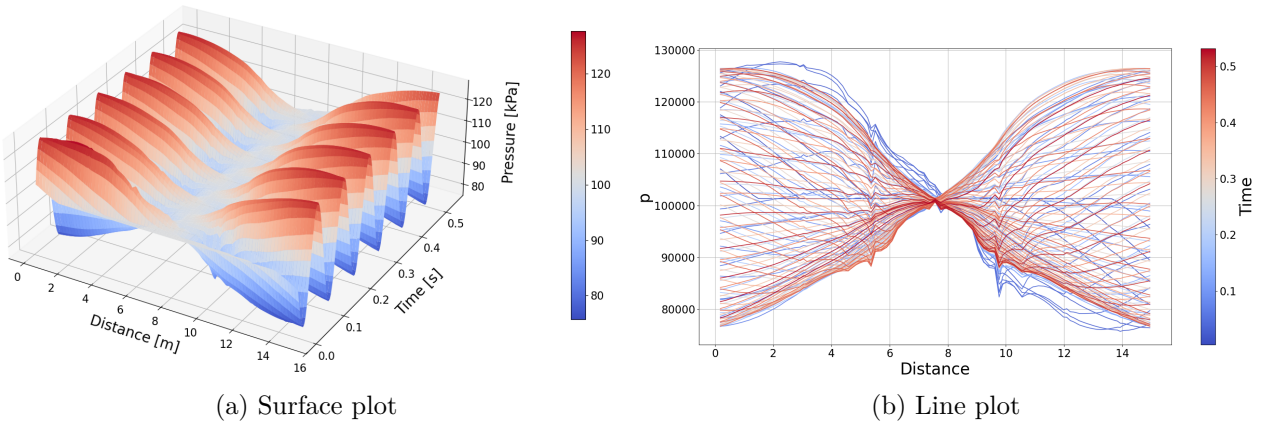


Figure 15: Surface and line plots of pressure as a function of distance and time. Distance is measured from pulsator to pulsator.

Figure 16 shows pressure across the length of the geometry, for a smaller range of time steps, starting at  $t = 0.492$  s and ending at 0.520 s. It depicts what the peak and the trough of the standing wave look like before they overlap and start interfering with each other. As in figure 15b deviations from a smooth curve is due to interaction with the geometry. Note how in figure 16a the curve is not perfectly sinusoidal. The peak seems steeper and the trough seems flatter. In 16b the interference between the peak and trough can be seen as it evolves through time. In the final time steps of figure 16b the back halves of the peak and the trough can be seen, as they have now entered the domain, coloured in dark green. The back half of the pressure wave can be seen, and the back half of the peak seems flatter than the back half of the trough.

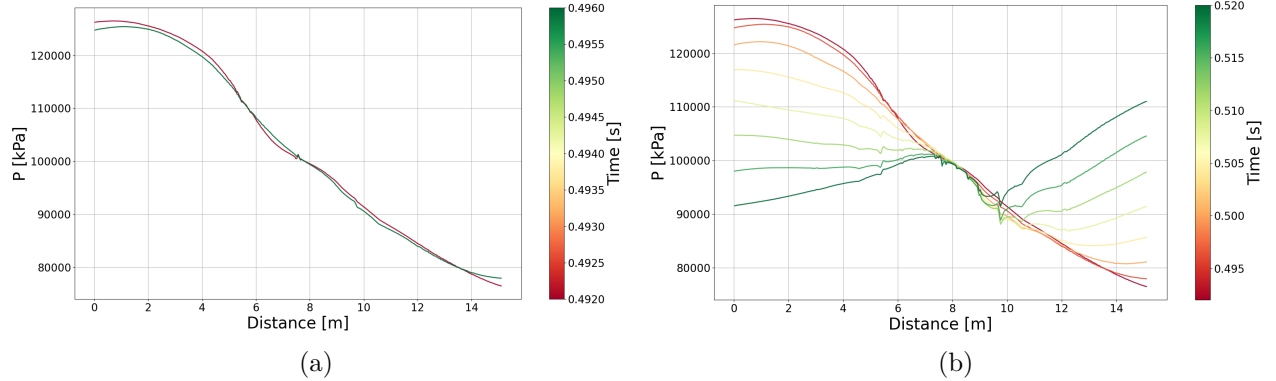


Figure 16: Line plots of pressure as a function of time and space.

Figure 17 shows the pressure distribution in a cross sectional view with the x-axis as the normal. The distribution is very even, showing very little deviation.

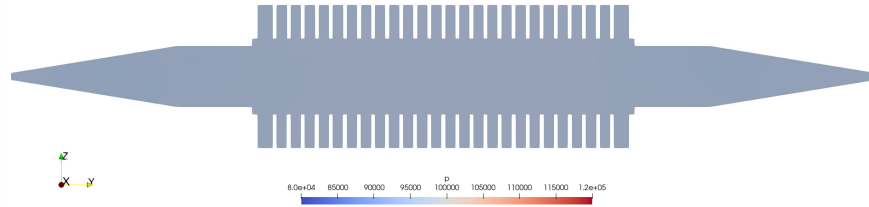


Figure 17: Cross section of the cooling box. X-normal at 1.5 meters from the centre of the cooling box. Pressure distribution at time 0.493 s.

To zoom in on the flow behaviour around the steel plate, figure 18 shows a plane normal to the y-axis but with a slight shift in position of 3 mm in the positive y-direction, meaning that the slice is placed in between the cooling fins. Here the velocity component is in the x-direction and the field is displayed with surface Line Integral Convolution (LIC), in order to look into the directionality of the velocity field. The time steps are chosen to show how the flow field behaves in the beginning of a new cycle, during and after the turning of the flow direction. First, in figure 18a, the flow is going from right to left and two quite symmetric vortices are formed downstream from the pipe outlet. These vortices seem to grow, moving towards the steel plate as can be seen in figure 18b. In figure 18c the flow is more chaotic with multiple vortical structures formed around the steel plates and the ribs. This is a time step taken during the reversal of the flow. In figure 18e, the flow is now going from the left to the right as the flow direction has turned. The behaviour of the streamlines is now generally smooth but the formation of the similar vortices behind the pipe outlet is seen.

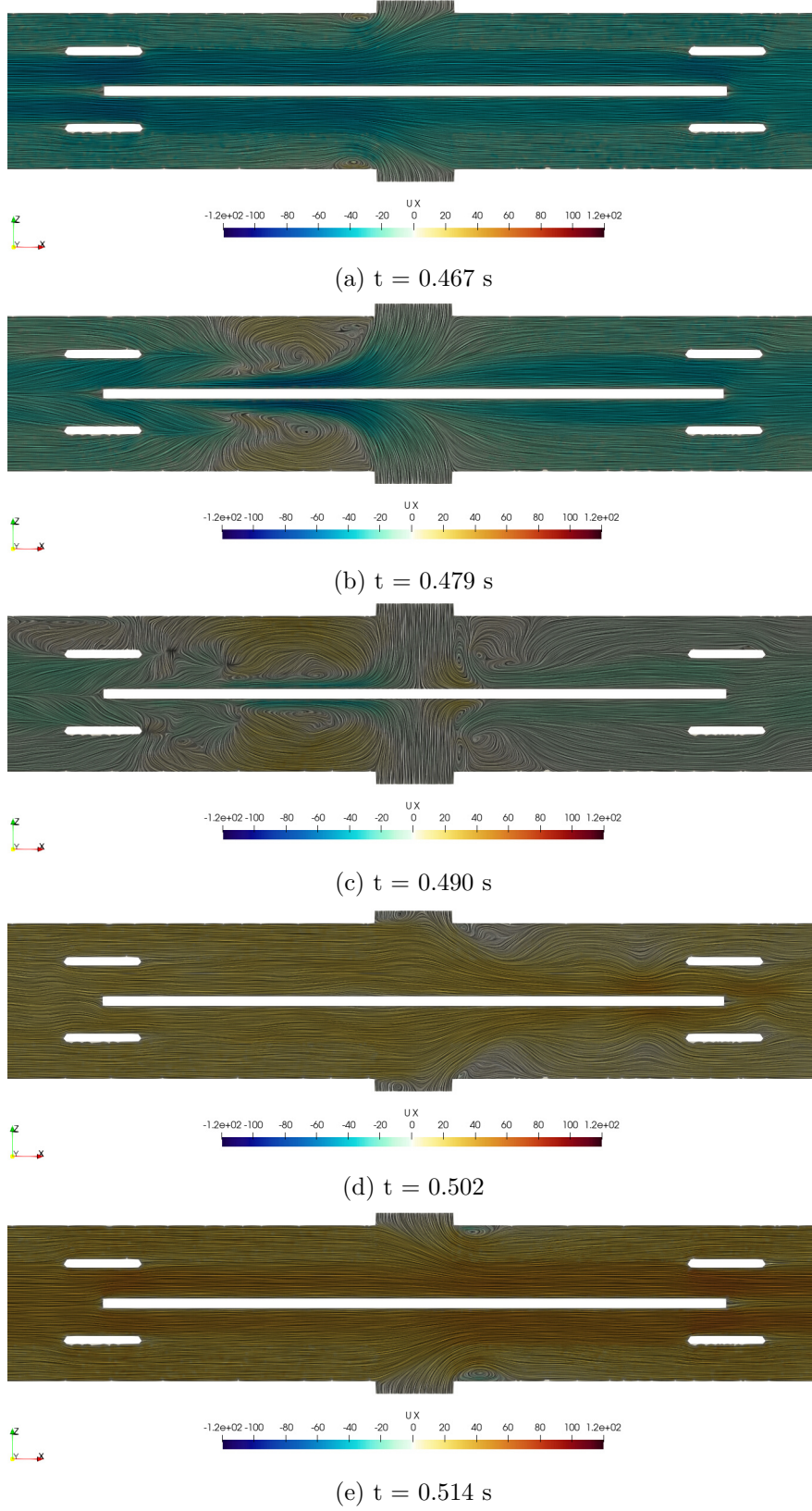
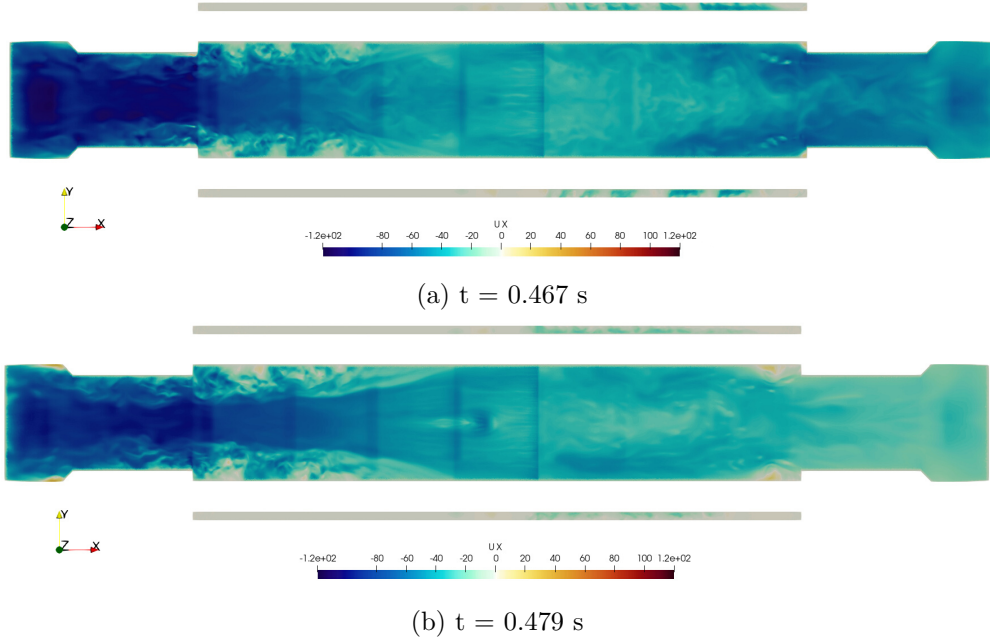


Figure 18: Velocity in the  $U_x$  direction with surface LIC applied. Figures (a) through (e) show time steps 0.472 - 0.514 s, which corresponds to half a pressure cycle. The colour gradient is fixed between -120 and 120 m/s for consistent colouring.

The velocity field in figure 19 displays a slice above the steel plate. It shows time steps during the reversal of the flow direction. As the slice is placed above the steel plate the side outlet shows up as a horizontal lines above and below the main body in the image. Note also that the perturbation in the middle of the slice is located just below the pipe outlet. The time step distribution is linear, yet the only time step with clear flow reversal is figure 19c, suggesting that reversal takes place during a very short time span even relative to the length of the cycle. Figure 19c also show that the air closer to the walls seem to reverse flow first, followed by the central bulk flow.

Figure 20 which uses the same view as figure 19, but across a shorter time interval during the turning of the flow. In this set of sub-figures (a) through (e), it is more clear that the flow starts to reverse closer to the walls.



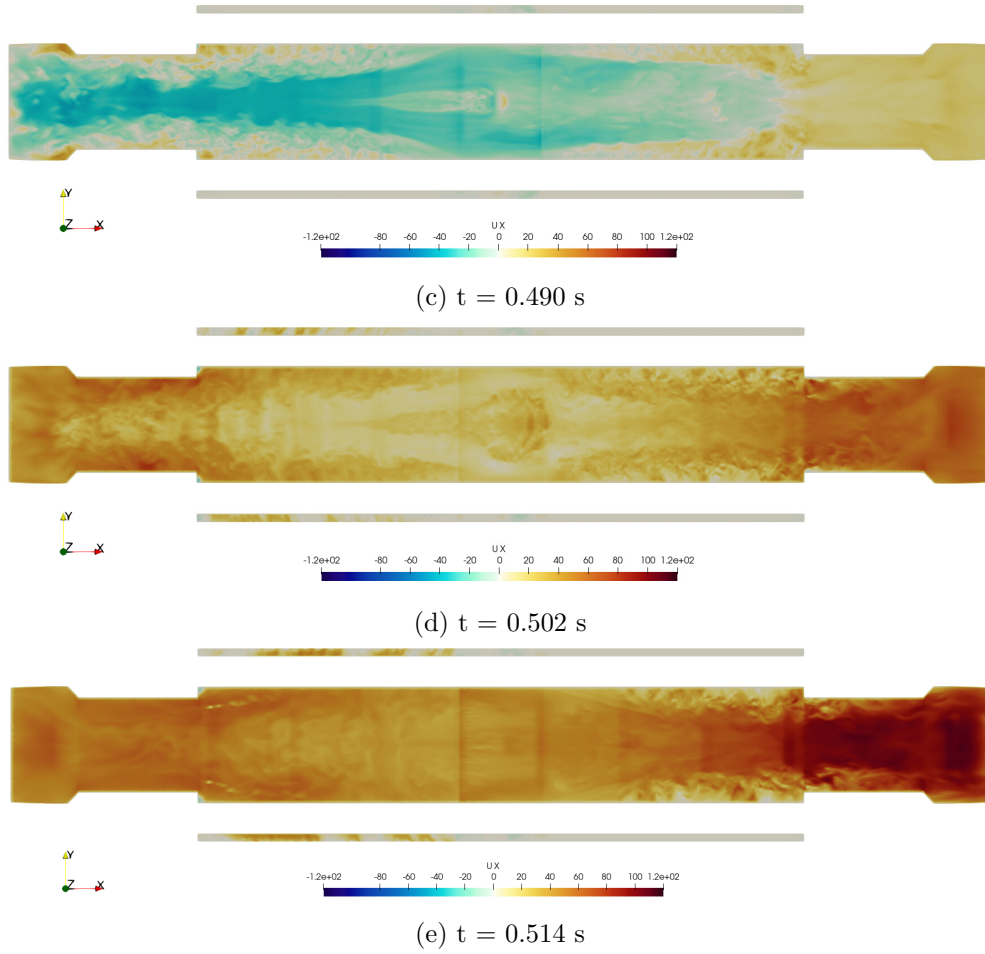


Figure 19: Velocity in the  $U_x$  direction. Figures (a) through (e) show time steps 0.467 - 0.514 s, corresponding to half a pressure cycle. The slice is placed 5 mm above the steel plate with the view in the  $z$ -normal direction. The colour gradient is fixed between -120 and 120 m/s for consistent colouring.



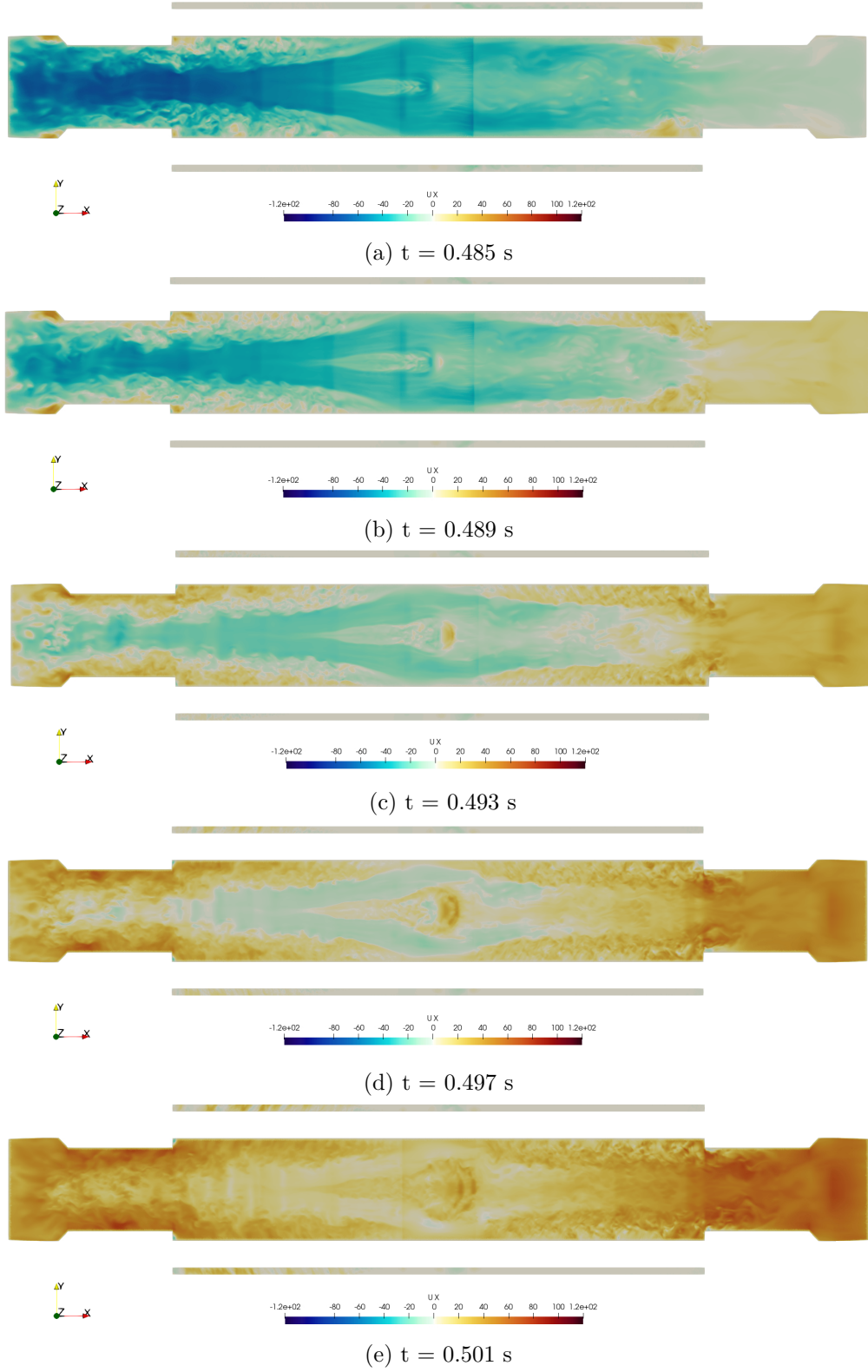


Figure 20: Velocity in the  $U_x$  direction. Figures (a) through (e) show time steps 0.485 - 0.501 s. The slice is placed 5 mm above the steel plate with the view in the z-normal direction. The colour gradient is fixed between -120 and 120 m/s for consistent colouring.



Plotting the velocity as a function of time and the distance from the pulsators can be seen in figure 21. Some time steps have been removed to make it simpler to interpret. There is a deviation in the data with an increase in velocity at about 7.5 meters at time 0.475-0.485 s. This is due to recirculation by pipe outlet at this time. What the figure shows is how the velocity changes at different points in the cooling chamber during the turning of the flow. In other words the slope of the line is the acceleration. The time steps correspond to figure 19 and distance range 5-7.5 meters is the left side in the same figure and the 7.5-10 meter range is the right side. On the left side the flow turns in a sinusoidal fashion and on the right side the lines are more linear in shape.

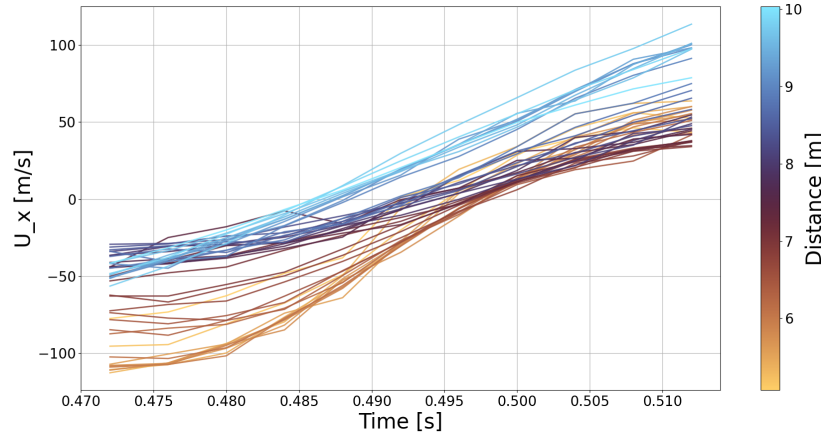


Figure 21: Line plot showing velocity in x-direction as a function of time and space. Colour gradient depicts the interval 5-10 meters, corresponding to the cooling chamber.

Figure 22 shows the velocity in the x-direction, with surface LIC. It illustrates the local flow direction, at the mid point in the turning of the flow. It visualizes how the region in between the two flow directions is effected by the shear stress.

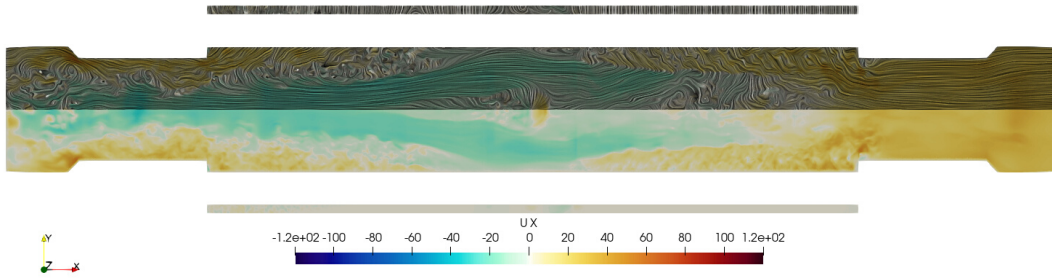


Figure 22: Velocity in the  $U_x$  direction. Time = 0.493 s. Slice is placed 5 mm above the steel plate. Upper half uses surface LIC. The colour gradient is fixed between -120 and 120 m/s for consistent colouring.

## 5.2 Hot steel plate - 700 K

The only change made from the previous ambient temperature case to this case is that the steel plate walls were set to 700 K. In figure 23 the temperature field is visualized in a slice normal to the y-direction. The plate is seen to the right in the figures. The temperature is scaled so that the maximum temperature shown is 390 K, for a better view of the heat propagation. Figure 23a is

taken as the flow direction streams towards the left of the domain. Here the heat is transported with the bulk flow, and it can be seen that the hot air travels with the flow as a coherent structure. As the fluid of higher temperature travels further away from the steel plate, it eventually breaks up as the flow reverses. In figure 23c the flow has changed direction, so flow near the cooling chamber walls moves faster to the right than the bulk flow. Now the hot air is going back towards the steel plate as seen in figure 23d, with more dispersed high temperature spots. The pattern then repeats itself, in the other side of the domain.

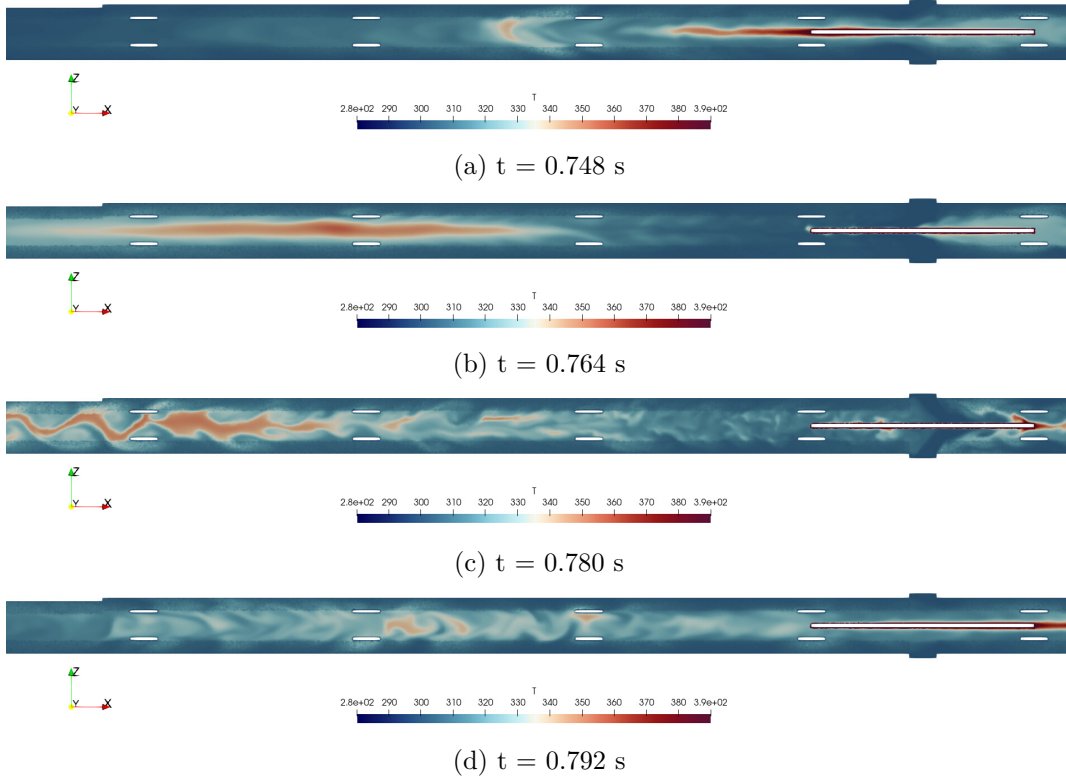


Figure 23: Temperature field sliced in the y-normal direction. Time steps distributed over half a pressure cycle. The colour gradient is fixed between 280 - 390 K for consistent colouring.

To further understand how the temperature field changes, figure 24 shows the same time steps as in figure 23 but for a z-normal slice. This is the same slice previously used for the 300 K case, located 5 mm above the steel plate. This view shows more of how heat is distributed right above the plate. At the time step in figure 24a, the heat is quite concentrated above the plate as it is taken when it returns after having reversed in the right side of the domain. Figure 24b shows the same behaviour as in figure 23b, with an elongated high temperature region in the flow direction. Thermal energy is here advected downstream as the flow is moving to the left. The high temperature region is again more scattered in figure 25c, as a consequence of the flow reversal. Stepping further in time to figure 24d, the peak temperatures are again around the centre, slightly to the right of the steel plate. The overall behaviour at this time step looks much like in figure 24a but mirrored, as is expected.

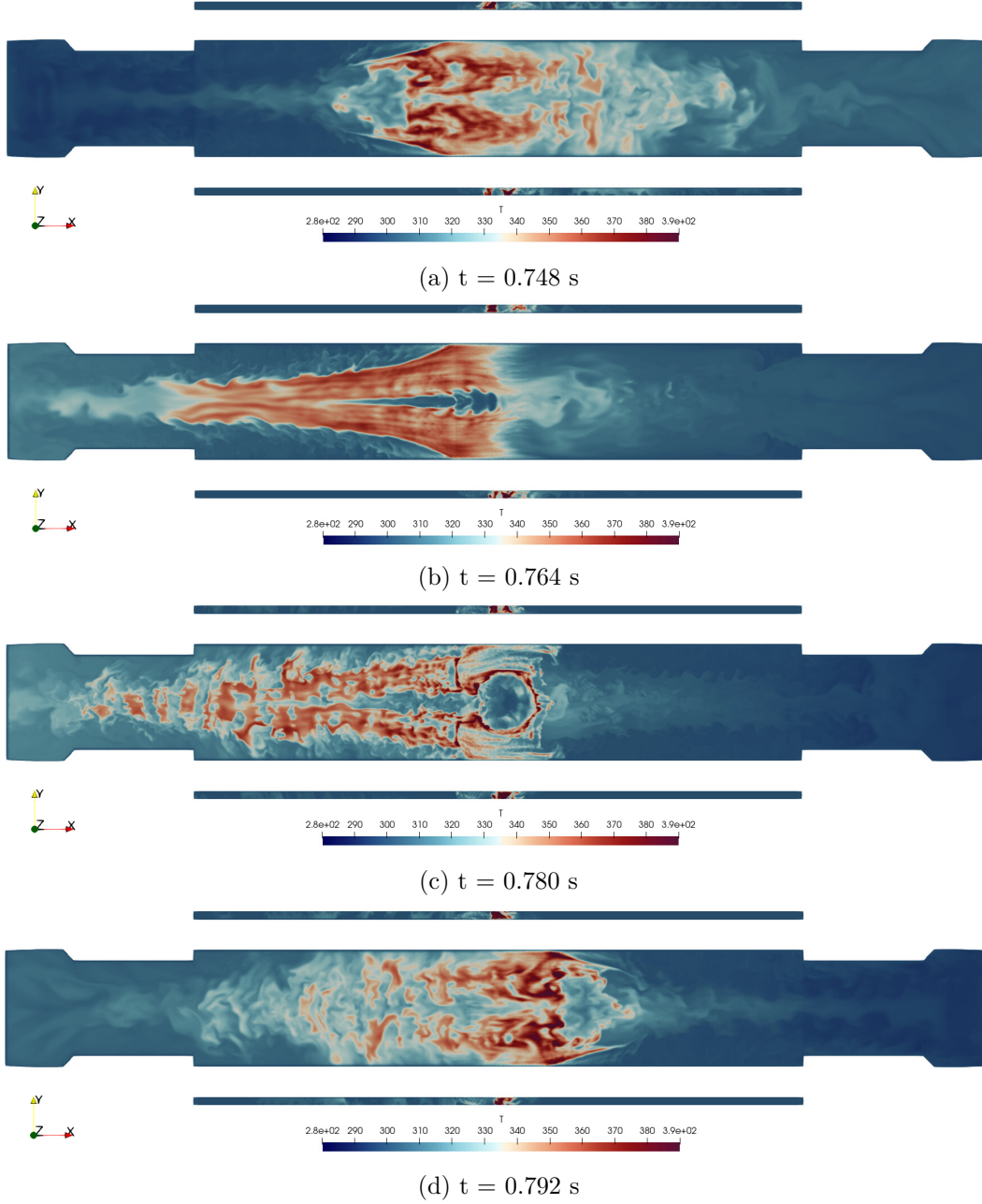


Figure 24: Temperature field slice in the z-normal direction, 5 mm above the steel plate. Time steps distributed over half a pressure cycle. The colour gradient is fixed between 280 - 390 K for consistent colouring.

As the propagation of the heat from the plate to the air in the domain is a three-dimensional phenomenon, it is necessary to look at the x-direction as well. Figure 25 illustrates again the same time interval as previously, but with a slice taken normal to the x-direction. The time steps that correspond with figure 25a, figure 25c and figure 25d all had a more dispersed temperature field in the previously looked at figures. In this x-normal view it is evident that this corresponds to more heat spread away from the plate of the domain, i.e more heat is transferred towards the cooling fins in figures 25a, 25c and 25d. At the time of figure 25b, the coherent structure previously seen is

shown as a quite uniform high temperature distributed over the plate.

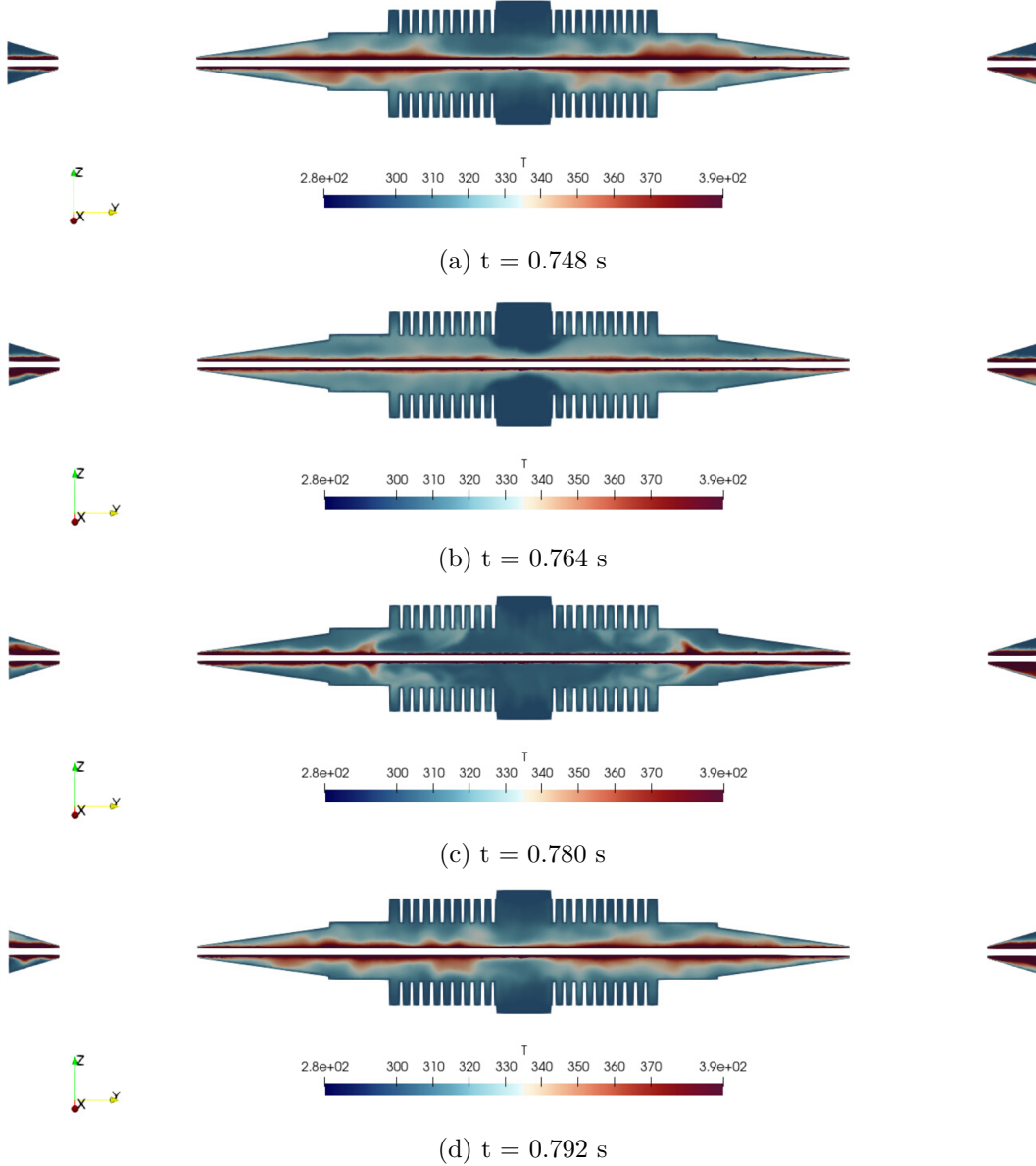


Figure 25: Temperature field slice in the x-normal, in the centre of the domain. The colour gradient is fixed between 280 - 390 K for consistent colouring.

After looking at figure 23, figure 24 and figure 23 the temperature field in three dimensions are outlined. However, the behaviour of the thermal field also changes depending on what cross section is displayed. Figure 26 displays the x-normal direction but for slices placed on different measurements from the centre of the domain. The figures are all taken on the same time step, 0.776, which corresponds to a time step during the reversal of the flow. As can be seen, the flow is going in the negative x-direction and very little heat is propagated in the opposite direction. Here again the unsteady characteristics of this propagation is captured as the thermal field evolves spatially in the domain. The cross section closest to the plate in figure 26b, shows strong thermal activity spreading almost throughout the spanwise direction of the slice. The corresponding figure in the

opposite direction 26a show some regions of higher temperature. When going further away from the steel plate in the positive x-direction, the temperature field is going towards a uniform ambient temperature as can be seen in figure 26g and figure 26i. Looking again in the negative x-direction the temperature field farther away from the plate is more diffused, show lower maximum temperatures and the thermal activity is more concentrated to the centre of the domain, as seen in figures 26f and 26h.

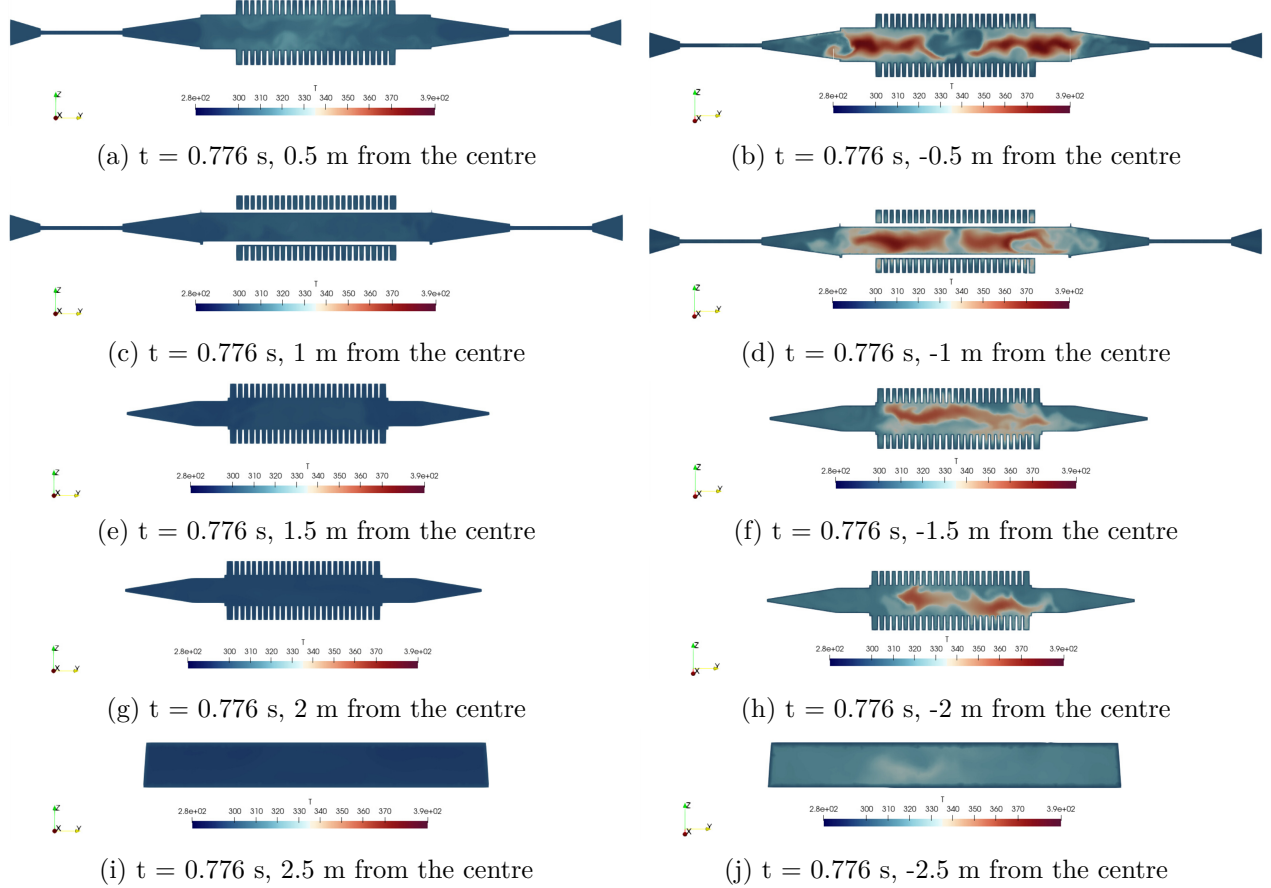


Figure 26: Temperature field normal to the x-direction with slices at different locations. The colour gradient is fixed between 280 - 390 K for consistent colouring.

To further understand how the reversal of the flow takes place, figure 27 shows a zoomed in view of velocity in the x-direction at the edge of the cooling chamber. The slice is taken in the y-normal direction, and depicts a zoomed in area between the red section and the purple section of figure 9a. The time steps are chosen to show the vortex formation and its interaction with the bulk flow.



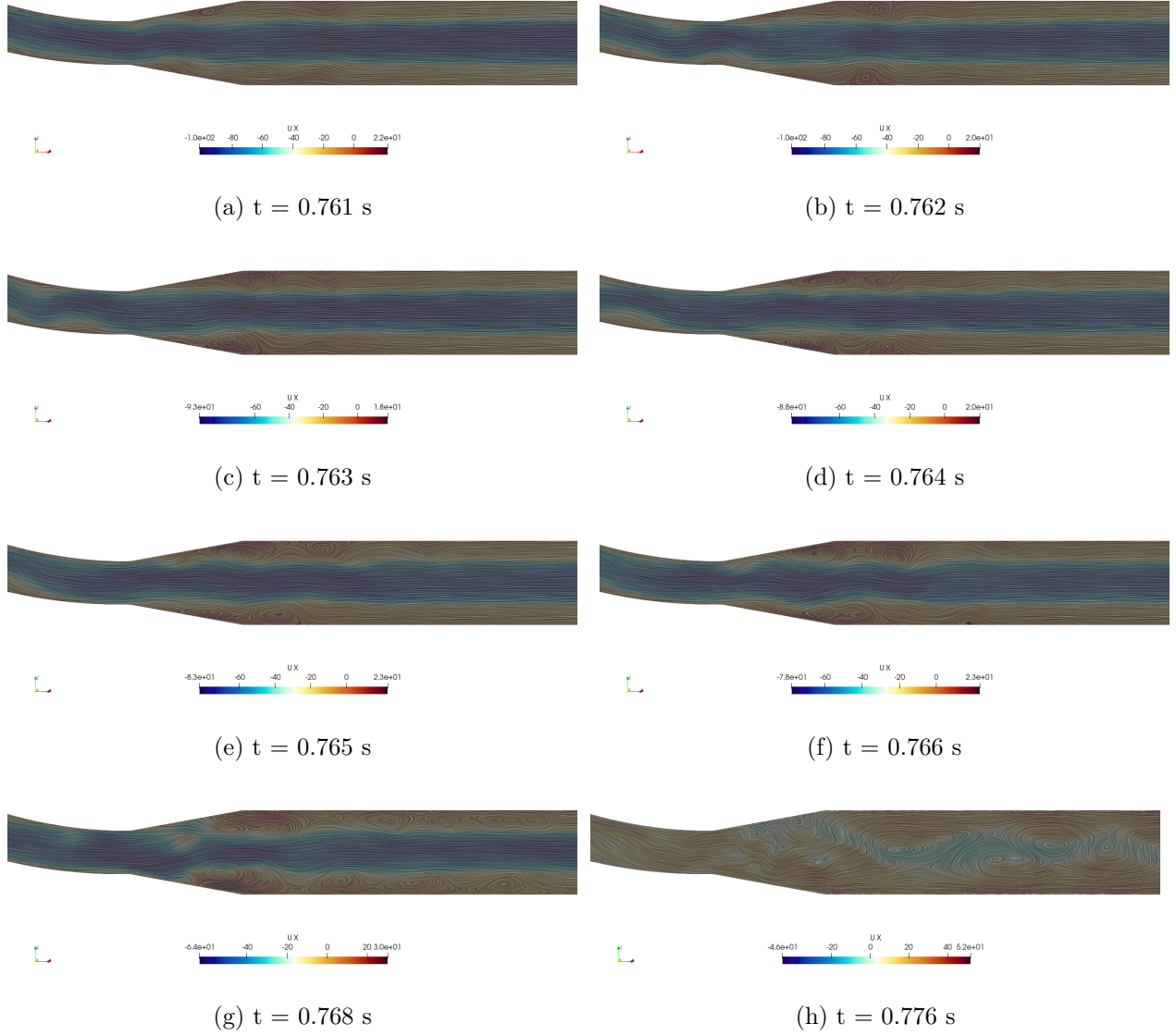
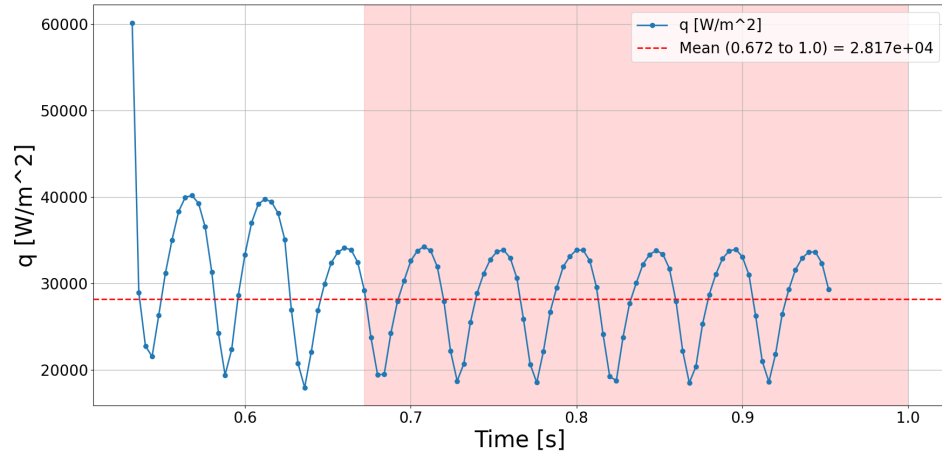


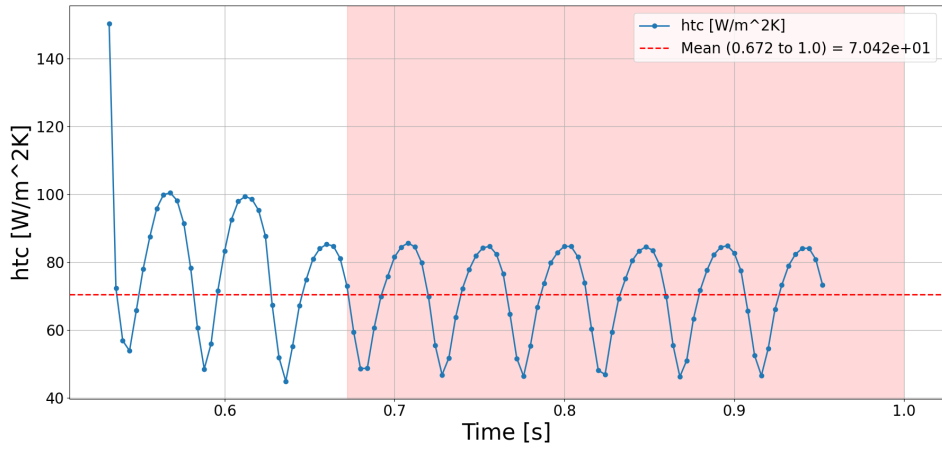
Figure 27: Coloured by velocity in the  $U_x$  direction, with surface LIC overlaid. Slice normal to the y-direction at time steps between 0.761-0.776 s. Note that the colour gradient is scaled according to maximum and minimum values.

### 5.2.1 Heat transfer coefficient and heat flux

In figure 28 the HTC, as well as the heat flux between the steel plate and the air, both as functions of time. It varies in a pulsating shape, with the peak value rounded and the trough value sharp. The initial spike stems from the instant change in the wall temperature applied to the steel plate, and should be treated as a numerical effect and not something physically accurate. The average of each value is calculated in the delta 0.672 - 0.952 s, which can be found in table 6.



(a)



(b)

Figure 28: Heat flux and the heat transfer coefficient of the steel plate as a function of time. The average and its interval marked in red.

Table 6: Average heat transfer coefficient and heat flux. Averaged in time and space.

| HTC and Heat flux |                         |                        |
|-------------------|-------------------------|------------------------|
|                   | HTC                     | Heat flux              |
| Average value     | 70.4 W/m <sup>2</sup> K | 28.2 kW/m <sup>2</sup> |

In figure 29 the instantaneous heat transfer rate over the steel plate can be seen. The time delta is the same as in figure 24. The figures show how the heat flux varies with the turning of the flow, as well as how the pipe outlet affect the heat transfer rate. Figure 30 shows the time averages of the heat transfer rate and the HTC and how it changes over the surface. Although the spread is fairly even, notable features is the circular pattern in the middle from the pipe outlet, and the striations aligned with the flow direction.

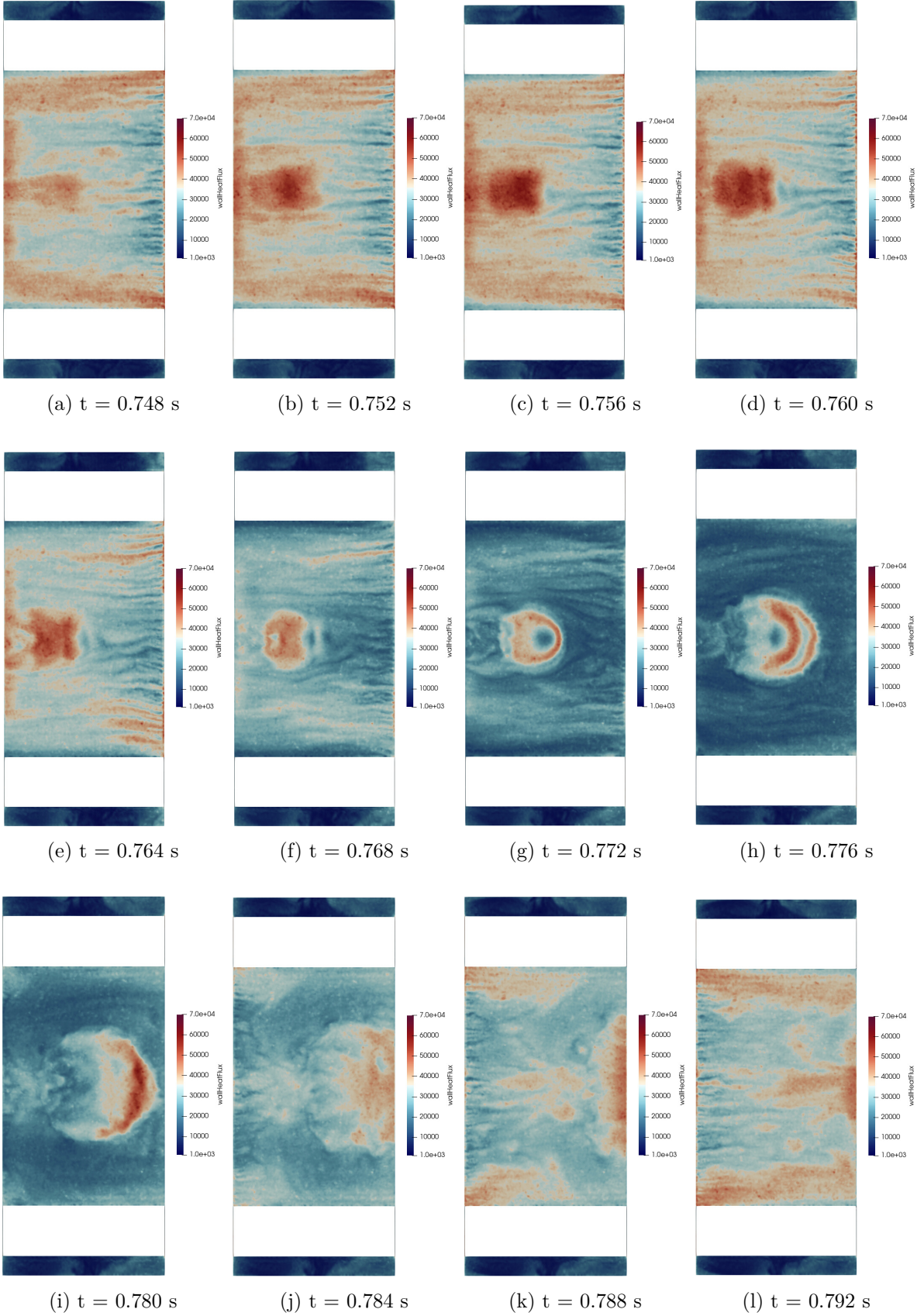


Figure 29: Heat transfer rate from the steel plate, upper side. The colour gradient is fixed to 1000 - 70000 W/m<sup>2</sup> for consistent colouring.



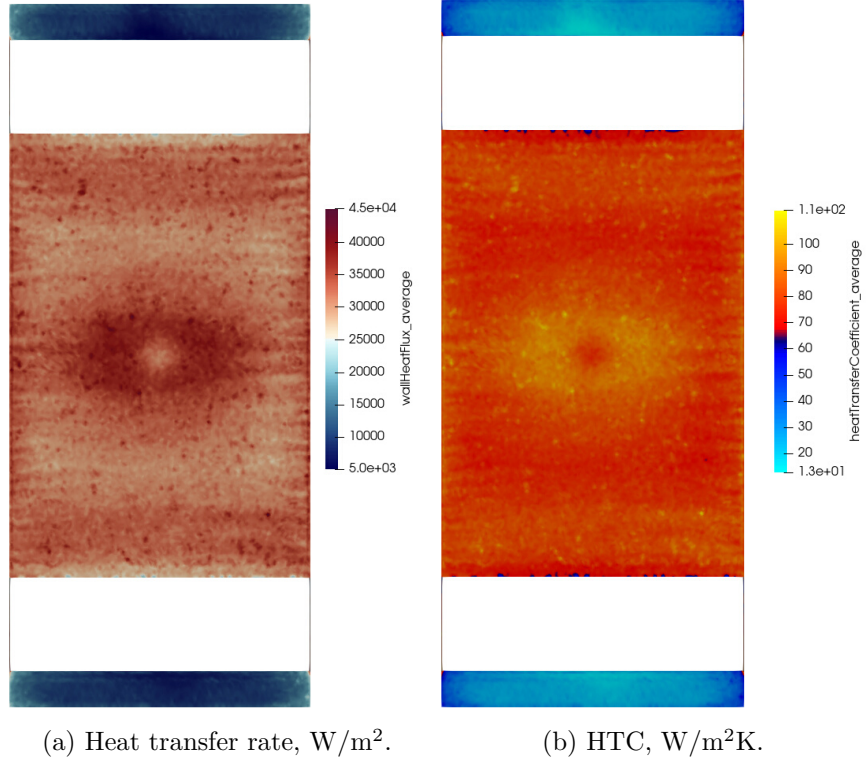


Figure 30: Time averaged heat transfer rate and coefficient. In these images the colour gradient corresponds with max and min values.

## 6 Discussion

After the results have been presented in the previous chapter, the following discussions aim to interpret the observed flow and thermal behaviour. The setup, both in numerical and more practical terms, is evaluated and error sources are identified.

### 6.1 Setup evaluation

A lot of the setup involves practical trade-offs. Balancing numerical stability and accuracy, such as divergence schemes where the choice landed on *limitedLinear* which is a second order scheme but tends towards first order near sharp gradients. Similar concessions have been made when considering boundary conditions. The pulsator boundary conditions are significant simplifications of their real life counterparts. Instead of moving a physical boundary to generate the pressure required, a simple pressure condition function was used. It is indeed possible to accurately model a pulsator by creating a dynamic mesh which would move the physical boundary back and forth. However, it was assumed that the increase in time it would take to model this would not yield an equivalent increase in the accuracy of the result, and therefore the idea was discarded. The purpose of this simplification was to scale down the complexity of the simulation. While the goal of this boundary condition was to build a standing wave over the length of the geometry. In figure 15a the standing wave is visible as a function of time, verifying that this setup has indeed produced a standing wave.

In the real application the pulsators establish a hard boundary which air cannot move through, and the displacement is generated by the oscillation of the pulsator wall. Although the standing wave generated seems to be established as intended, there are no measurements to compare them with other than the maximum and minimum pressure values which are manually assigned in the model. Meaning there are no pressure values other than the shape of the pressure wave to verify the physics against. However, it is possible to make a few simple comparisons between theory, reality and the simulation to check if any of the results can be deemed reliable.

The real pulsators have an amplitude of about 40 - 45 mm, which when accounting for the sinusoidal shape of the velocity profile and their increased cross-sectional area, comes out to a maximum displacement of 0.41 - 0.46 m<sup>3</sup>/s. In the simulation the corresponding inlet volumetric flow is 1.2 m<sup>3</sup>/s, which is an increase of 160 - 190 %. This is a non-trivial difference, but the impact of it is difficult to evaluate. The volumetric flow at the pulsator and the inlets would ideally be the same or comparable, but whether or not this suggests that the rest of the flow is inaccurate cannot be determined from this alone. The pulsator is a solid boundary, while the boundary condition is more similar to attaching a large reservoir with fluctuating internal pressure. How this affects the physics inside the domain, especially in the cooling box which is far away from the inlets is very difficult to judge.

Another pressure boundary condition that is important is the pressure outlet conditions. Initially a fixed pressure was applied here but it was found that the condition *totalPressure* was more stable. Simulations using the fixed condition created unphysical low pressure vorticities just inside of the boundary, where there is inflow. This points to the fact the fixed pressure condition made the pressure gradient too sharp. This in turn suggests that the inflow was larger than what the simulation could handle. The total pressure condition should be applied where inflow velocities are large enough to cause a pressure drop at the boundary. This resolved the numerical issue that the fixed boundary condition caused, but it does not mean that either boundary conditions are un-

physical in nature, or unrealistic compared to a real setup. To determine that, velocity or pressure measurements from an experimental rig would be required.

It is also prudent to compare a theoretical particle velocity with the simulated particle velocity in the setup. The theoretical particle velocity for ambient conditions and a pressure wave of 25 kPa, should be about 60 m/s. The corresponding value in the simulation is found by extracting a central plane at the middle point of the geometry and reading the maximum value in the lengthwise direction  $U_x$ . This turns out to be 53 m/s, which is a deviation of about 12%. This is a good indication, however there are lot of variations between the theoretical value and the simulation. The theoretical value is one dimensional and does not account for geometry or friction. The theory is also based on linear acoustics, and the simulation shows that a 25 kPa is not linear in nature. In this case the geometry in the middle part of the cooler includes outlets in the z-direction and y-direction. The pipe outlet is very close to the central flow and as seen in 18 a significant amount of flow is sourced from this outlet. This implies that the geometry plays an important part in determining the flow structure and velocity.

The particle velocity in the middle part corresponds fairly well with theory, according to which the largest displacement and thus particle velocity should be in the centre of the geometry. In the simulation however, the largest velocity is found in the red section in figure 9, which is just outside of the main cooling chamber and it is significantly larger at about 120 m/s. The difference is large compared to 60 m/s and does therefore not align well with the theory. On the other hand, the pressure levels achieved in this simulation introduces non-linear phenomena into the pressure curve, which in turn will affect the velocity, and invalidates the linear theory. Another part of the difference can be explained by the geometry. In figure 9c we can see that the red sections are not as wide as the main cooling chamber, which means that the cross sectional area decreases as flow passes through. As such the red section acts as a nozzle, which should speed up the flow. Another source for this high velocity and unexpected placement may be a system that is not really resonating. In practical experiments the resonance is found by finding the frequency which generates the greatest pressure, but because this model fixes the pressure and the frequency directly, there is no feedback. This essentially enables the creation of a standing wave at a desired pressure without actual resonance. If the resonance tubes are too short, but the standing wave is well established as it would be for a longer system, then the peak velocity could be placed downstream of the centre, which is the case in these results. It turns out it may be difficult to properly tune the simulation without regulating the frequency. Since the speed of sound is dependent on the temperature, choosing an appropriate ambient temperature is also important. The system was originally tuned for 10.7 Hz. At 20°C this requires a length of 16 meters, which this domain is not. It is in fact 15.1 meters, and the ambient temperature is 300 K (= 27 °C), which increases the speed of sound and the required length which extends to 16.2 meters. This highlights a source of error, and something that needs to be accounted for when designing this kind of system. As the internal temperature may rise during operation despite cooling functions, the frequency would need to be regulated. With that being said, this may not be a debilitating mistake. Although the frequency for this system is not correct, it does not have to be perfect, in practice there is not one resonance frequency but rather a span in which there is resonance. That coupled with a standing wave that is fully formed, though it may be compressed due to the shorter tube length, means that the flow behaviour should in large part still be representative. The theoretical velocity is flawed so it is not possible to know exactly what to expect, and even though the resonance tubes are too short, the turning of the flow and how the hot and cold air is mixed should still be informative.

## 6.2 Ambient temperature steel plate - 300 K

In figures 20c and 22 it is clear that the flow direction is both in the positive and the negative x-direction. What figure 22 is able to show specifically is the interaction in the area between the two flow directions. The surface LIC image shows how the shear stress causes large vortices to appear. This coupled with the vortex structures in figure 18c show that during the turn complex flow structures are generated. The direction of the rotational structure appears when looking at slices from all three directions, and at many different points in the cooling room, as seen in figures 20, 18 + en till (x-normal). This suggests that the oscillating air flow creates potential for great mixing in the domain.

As mentioned in the results section, the flow seems to turn quite rapidly. This is supported by the short time frame in which it happens. This should not come as a surprise because of the sinusoidal shape of the pressure function. The rate of the turning of the flow should be a function of the magnitude of the pressure, which pulls the flow in the opposite flow direction. The force of this pressure peaks at a short interval, at the top of the sinusoidal curve. This coincides with the turning of the flow, which matches the theory. The time at which the pressure curve peaks corresponds to the point in time where the displacement curve peaks, like in figure 2.

During the turn, the flow seems to turn first at the edges and then in the centre, see figure 20. This is probably due to how the flow speed is distributed prior to the turn and how the turn is generated. In figure 19a almost the entire field is flowing in one direction, and in the next few time steps it will turn and change flow direction. Figure 19b shows that the flow slows down and starts to turn at the parts of the flow that were previously the slowest. When considering the mechanism with which the flow is turned, this seems reasonable. The pressure distribution in figure 17, is very even, which means that the force exerted on the fluid during the turn is the same no matter how much momentum it has. If the velocity in a cross section of the flow varies in speed, it will take different amounts of time for each fluid particle to change direction, because the same pressure is applied. This results in flow in both directions at the same time.

Figure 17 shows the pressure distribution in a cross section of the geometry, and figure 13 shows how the pressure deviates with the length of the cooling chamber. Building on the arguments from the previous paragraph, if the pressure pulling or pushing on the flow is greater closer to the pulsators than at the centre of the cooling chamber then it follows that the flow should turn first at the beginning and the end of the cooling chamber and last in the centre. Looking at figure 20d the flow does indeed behave as suggested. This is further supported by figure 21, where the line plots from the middle of the cooling chamber are flatter than the plots from further downstream and upstream.

Figure 21 also shows that the flow downstream of the steel plate turns in a sinusoidal fashion, while the flow upstream turns in a more linear fashion. A sinusoidal shape is not strange given how the pressure driving this flow varies sinusoidally. The more flat shape of the line plot upstream of the plate is more strange. Looking at the pressure driving the flow reversal a similar shape can be found. Figure 16a shows the peak and the trough of the pressure profile. The low pressure side of the standing wave is much more linear in shape compared to the high pressure side. This seems to be due to the high pressure of this standing wave. As mentioned in the theory section, high pressure waves exhibit non-linear behaviour, where peaks catch up to the troughs of a wave. This results in the peak arriving earlier at the middle than the trough. It also changes the shape of the wave. The peak should be sharper and the trough flatter which is clear in the figure. Figure 16a shows the

first quarter of each wave and figure 16b shows time steps as the waves interfere with each other. In the second image two things can be seen. The trough has not yet reached the centre and the second quarter of the peak is flatter and the trough is steeper, which is reasonable according to the theory presented.

### 6.3 Hot steel plate - 700 K

When looking at the temperature field for when the steel plate is set to 700 K it is evident that it is strongly coupled with the velocity field. The thermal structures do not seem to move independently, but rather follows the acceleration, deceleration and the turning of the flow. It is almost as if the temperature behaves as a passive scalar, closely following the redistribution of momentum caused by the pulsating flow. One observation from the three views depicted at time  $t = 0.764$  s in figures 23b, 24b and 25b has in common is that all of them show a more cohesive temperature field with a stretched plume-like appearance. For example in figure 24b, the corresponding velocity field appears more organized, with the bulk flow accelerating in the streamwise direction. This clearly enhances advection of thermal energy away from the plate. Stepping further in time from here, the flow starts to turn, disrupting the previously seen coherent structure. It begins with the flow near walls turning so that the contour of the temperature structure is interrupted, before the disruptions reach further and the whole structure gets more scattered as seen in figure 24c. After this the now dispersed high temperature region moves towards the steel plate again and the pattern repeats itself.

The z-normal direction highlights the thermal distribution above the plate and the formation of the elongated structure advected downstream. When looking at the temperature field in the y-direction in figure 23 instead, more can be discovered about how the thermal energy is spread along the height in the domain. In figure 23a the heated region leaving the plate is relatively confined near the vertical centre. As the propagation along the flow direction continues the heated area is slightly thicker, reaching closer to the walls of the domain. It is however not until the turning of the flow that the heat is extended along almost the full chamber height, as seen in figure 23c. Here the sinusoidal behaviour of the pulsating flow generates strong vertical redistribution of the heat, showing that the turns induce strong thermal mixing in the domain.

The three-dimensional phenomenon of thermal transport inside the cooling chamber is completed by looking at the x-normal slice in figure 25. While the z-and y-normal views showed streamwise transport the x-direction emphasizes how the pulsating flow redistributes the heat spanwise across the chamber width. As previously noted the advection of heat at time  $t = 0.764$  s is seen in this view as well. Notably the temperature field in figure 25b when this is happening shows that the thermal energy is more evenly distributed rather than concentrated in isolated high temperature structures. In the other time steps, the trend is that the irregular flow structures trap thermal energy locally, while the areas without these structures remain comparatively cool. It seems as though the more cohesive velocity field at  $t = 0.764$  s prevents the formation of the hot regions, and instead distributes more moderate thermal energy over a larger area. At the next time step  $t = 0.780$  s after the turn the velocity field is more non-uniform, and the previously uniform thermal distribution is broken down into the localized high-temperature regions again. Here in figure 25c another notable feature is found, as the heated fluid is now located further away from the centre and the pipe outlets. This pattern can be traced back to what was seen in the z-normal direction in figure 24c. Looking at the centre of the domain there, a distinct region is seen where the temperature remains comparatively cool. This region coincides with the pipe opening, serving both as inlet and outlet. The pipe is shown in the x-normal slice, with the majority of the air in it keeping

ambient temperature for all time steps in figure 25. What can be specifically seen in figure 25c is that a broader region in proximity of the pipe remains cool. Since figure 24c is taken 5 mm above the steel plate, the ring-like region of cold fluid corresponds to what is seen in figure 25c. The behaviour suggests that the temperature field is not only governed by the heated plate and the main chamber flow, but also by the exchange through the pipe. Since the boundary condition for temperature here is an *inletOutlet*, with an inlet value of 300 K, any backflow through the pipe introduces relatively cold fluid into the chamber. As previously discussed, the flow after the turn is highly unstable and vortex-dominated, something that could promote local flow reversal near the pipe opening. This means that the 300 K inlet temperature naturally cools the surrounding thermal field, which explains the colder core appearing in figures 24c and 25c. This observed cooling region is physically plausible, as ambient air would be drawn into the chamber through the pipe when the pressure in the chamber drops locally. However, from previous discussions it is known that the centre of the chamber is expected to experience very little pressure fluctuations as it corresponds to a pressure node of the acoustic field. The boundary condition at the pipe outlet is set to *totalPressure*, allowing the patch pressure to adjust according to the local velocity. Although this limits the inflow from the ambient surrounding, there is still sufficient inflow at 300 K to produce the colder region observed.

Figure 26 shows how the thermal field changes with distance away from the plate. Unlike the previous slices that all has been taken on a plane in the centre of the domain these reveal how the influence of the heated plate decays as the distance from the plate increases. The slices taken in the negative x-direction move sequentially with the flow direction in 0.5 m intervals, and the images are taken at  $t = 0.776$  s which is between the time steps in figures 25b and 25c. This is why the figures in the right column displays higher temperatures than the left column. In figure 26b the temperature field shows two asymmetric higher temperature regions, separated by a colder region in the middle. This is again the effect of the pipe outlet in the middle, something that is still present in the next slice further away in figure 26d. It is not until reaching -1.5 m away from the plate that the thermal field is coherent without the separation in the middle, indicating that the pipe outlet affects the thermal structures in this view as well. It is clear that the peak temperatures are lower going further away from the plate, reaching an almost uniform temperature field at -2.5 m away. Furthermore, these slices can give some insight into how the heated fluid interacts with the fin geometry. The heated structures do not remain confined to the chamber centreline, but expand in the z-direction towards the fins. Especially in figure 26f, thermal contours penetrate the finned region both on the upper and the lower side of the chamber.

Understanding how this hot air moves towards the fins requires a closer look at this area as the flow reverses. During the turning of the flow in the y-normal view, the velocity field shows that the process starts at the very end of the cooling chamber, so in this case further left than what is seen in figure 23. In a corner in the red section seen in figure 9a, a vortex structure is built on the upper and lower wall before the completion of flow reversal. This is due to the geometry disrupting the flow there and the vortex can be seen in figures 27a and 27b, forming at  $t = 0.761$  s i.e before 23b. These vortex structures seem to grow as the flow reverses, propagating downstream in the new direction. Not only do they move towards the steel plate in the middle of the cooling chamber, they also grow and drift towards the centre line, as seen in the following sub-figures of figure 27. This is not strange, as the near wall air changes direction first, moving the near wall vorticities. The shear stress between these opposite flows seems to promote these structures creating more of them. The layer that has changed flow direction grows as the rest of the bulk flow has reversed which causes the vorticities to enlarge as well. At the final stages of the reversal of the flow these vorticities now

occupy most of the bulk flow as seen in figure 27h. These vorticities push the bulk flow into the sinusoidal pattern illustrated in figure 23c, with the hot air closely following the oscillating shape. It seems as though the vortical structures help transport heated fluid toward the fins, preventing heat from being confined in the centre and increasing contact between the hot fluid and the cold surfaces. This demonstrates that the heated air reaches the fin geometry successfully.

Now that a qualitative understanding of how the temperature redistribution is affected by the flow reversal has been provided, a quantitative analysis is desirable. Quantifying the cooling is done by measuring the heat transfer rate and the heat transfer coefficient. The time averages can be found in table 6. A heat transfer coefficient of  $70.4 \text{ W/m}^2\text{K}$  is not very far from the value calculated by Infrasonik. Their calculated value is about  $90 \text{ W/m}^2\text{K}$ . It should be noted that the value given by the simulation in this thesis cannot be fully accurate, because the boundary layer is not resolved by the mesh. This can be seen in figure 23. In part because resolving the boundary layer does give more accurate heat transfer results, but also because in this case there should be Rayleigh streaming on the surface of the walls and steel plate at the centre of the cooling chamber, which should affect the heat transfer results. From figure 29 it is clear that the pipe outlet significantly affects the cooling rate, by the circular shape in the middle of the plate.

Although figure 25 only shows the boundary layer over the centre of the plate, some correspondence between them and figure 29 can be established. For time step  $t = 0.780 \text{ s}$  in figure 25c the boundary layer is thinnest in the middle and thicker about halfway to the edge. This should correspond to better cooling in the middle compared to the further towards the edges. Looking at figure 29i the centre line from top to bottom does indeed have worse heat transfer rate away from the middle. Looking at another time step,  $t = 0.764 \text{ s}$ , there is high flow rate as can be seen in figures 23b and 24b, but the boundary layer is mostly even. However, in the middle just beyond the boundary layer, the temperature is noticeably cooler in figure 25b, which should translate to better heat transfer rates in this area. Looking at figure 29e one is reminded that the flow field is three dimensional. The flow inserted into the domain by the pipe outlet introduces colder air, but it seems to hit the plate slightly downstream of the centre of the plate.

The source of the striation pattern on the plate is not entirely clear. One potential source is the previous pressure and flow cycle. In figures 24a and 24d the hot air that has been broken up passes over the steel plate. Figure 26 shows how that broken up air non-uniformly is distributed in the x-normal cross section. The impact of it can be seen in figures 25a and 25d where the hot broken up structures pass over the plate again. The non-uniform thickness above and below the plate can clearly be seen. This can potentially be the source of the striations seen in figure 29, as hot regions above the plate should result in worse heat transfer at those points, and the opposite for points where there is less hot air passing over. Although the striations are quite prominent at instantaneous points in time, the time averaged field in figure 30 shows that the field is much more even overall, and the deviations from left to right has been evened out as is expected. It is also clear that the cooler air from the pipe outlet does increase the heat transfer rate in the middle of the plate.

## 7 Conclusion

This thesis has studied the flow in an infrasound cooling machine. It has shown that it is indeed possible to simulate the cooling system of the InfraCooler using CFD. The set sinusoidal pressure boundary conditions establish a standing wave in the system, which drives the flow. The flow field inside the cooler is complex and non-linear. The pressure driving the flow, forces the air through the cooling chamber as it passes over the steel plate. The pressure curve is non-linear which is what causes the flow, especially as it reverses direction to also do this non-linearly. The peak of the pressure curve is steeper and forces the corresponding side of the flow to turn faster than the other side which has a flatter pressure curve. Conversely, the transverse reversal of the flow shows how the flow near the wall changes direction before the more central bulk flow, creating high shear stress regions which generate vortical structures. These structures grow and move towards the centre as the boundary between the two flow directions moves inward.

The thesis also shows how the temperature distribution develops from a hot steel plate. The air passing over the steel plate is heated as it moves downstream of the plate. As the hot air is moved downstream, it is de-accelerated to a stop and then returned to the middle of the cooling chamber. As the flow is turned, a great deal of mixing occurs and the heat is distributed through advection and spread through the geometry. The air is then reused as it cools the steel plate in the second half of the cycle. The heat transfer coefficient of the 700 K steel plate was approximated to an average of  $70.4 \text{ W/m}^2\text{K}$  in space and time. This spatial distribution is fairly even across the plate. It exhibits a striation-like pattern due to the previous flow cycle, which is averaged out across time. The heat transfer rate is also heavily affected by the cooling chambers access to ambient air through the pipe outlets on either side, which increases cooling in the centre of the plate.

The work in this thesis is by no means comprehensive for the field in general nor this specific setup in particular. There are many areas of future work that this type of cooling solution would benefit from. In this work, there was not time to answer all potential questions, such as: Why is the peak velocity not located in the centre of the chamber? As lower pressure levels do not have the same non-linear effects on the flow as high sound pressure levels have, how does lower pressure levels affect the flow field? Or, as discussed, some of the boundaries may cause the field to be less accurate than possible. How does a moving boundary, which would better replicate the real life pulsator, compare with the results in this thesis? Or, is closing outlets in the cooling chamber beneficial to the cooling? Moreover, as this project is subject to limitations in computational cost, the boundary layer is not resolved. It would be interesting to see how a fully resolved boundary layer would impact the heat transfer rate. A more resolved cooling chamber would also be able to answer questions about the flow field near walls and in between the fins. These matters might seem detailed but to fully understand and quantify the heat transfer from the plate to the fluid accurately, the boundary area needs to be studied in detail.



## References

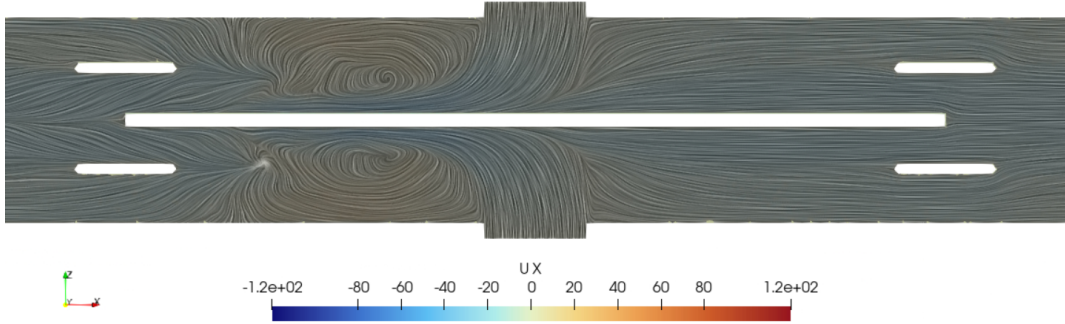
1. Agency IE. Iron and Steel Technology Roadmap: Towards More Sustainable Steelmaking. International Energy Agency, 2020. Available from: <https://www.iea.org/reports/iron-and-steel-technology-roadmap> [Accessed on: 2026 Apr 19]
2. World Steel Association. World Steel in Figures 2025. Accessed: 2026-04-19. World Steel Association, 2025. Available from: <https://worldsteel.org/data/world-steel-in-figures/world-steel-in-figures-2025/>
3. Infrasonik AB. InfraCooler [Internet]. 2026. Available from: <https://www.infrasonik.se/infracooler> [Accessed on: 2026 Apr 24]
4. Jernkontoret. Järn- och stålframställning: Låglegerade stål (Del 11) [Internet]. 2019. Available from: [https://www.jernkontoret.se/globalassets/publicerat/handbocker/utbildningspaket/jarn-och-stalframstallning\\_del11.pdf](https://www.jernkontoret.se/globalassets/publicerat/handbocker/utbildningspaket/jarn-och-stalframstallning_del11.pdf)
5. Ye Q, Zhang Y, and Wei J. A comprehensive review of pulsating flow on heat transfer enhancement. *Applied Thermal Engineering* 2021; 196:117–275. DOI: <https://doi.org/10.1016/j.applthermaleng.2021.117275>
6. Mikheev NI, Gazizov IM, and Davletshin IA. Heat transfer in pulsating flow behind a rib in the channel inlet region. *Journal of Physics: Conference Series* 2018 Nov; 1105:12–8. DOI: [10.1088/1742-6596/1105/1/012018](https://doi.org/10.1088/1742-6596/1105/1/012018)
7. Davletshin I, Mikheev A, Mikheev N, and Shakirov R. Heat transfer and structure of pulsating flow behind a rib. *International Journal of Heat and Mass Transfer* 2020; 160:120–73. DOI: <https://doi.org/10.1016/j.ijheatmasstransfer.2020.120173>
8. Davletshin IA, Gazizov IM, and Paereliy AA. Heat transfer in the flow with forced pulsations in a rib-roughened channel. *Journal of Physics: Conference Series* 2018 Nov; 1105:012022. DOI: [10.1088/1742-6596/1105/1/012022](https://doi.org/10.1088/1742-6596/1105/1/012022)
9. Woods BG. Sonically enhanced heat transfer from a cylinder in cross flow and its impact on process power consumption. *International Journal of Heat and Mass Transfer* 1992; 35:2367–76. DOI: [https://doi.org/10.1016/0017-9310\(92\)90079-8](https://doi.org/10.1016/0017-9310(92)90079-8)
10. Cengel YA and Ghajar AJ. Heat and mass transfer. Penerbit McGraw-Hill Education, New York, 2020
11. Sundén B. Introduction to heat transfer. Wit Press, 2012
12. Berg RE. sound [Internet]. Encyclopedia Britannica. Available from: <https://www.britannica.com/science/sound-physics> [Accessed on: 2026 Feb 13]
13. Olsson M. Ljud. Presented on 2026-01-26, during the introduction of Infrasonik to the Master students.
14. K. Forinash WC. Tube Resonance [Internet]. Available from: [https://phys.libretexts.org/Bookshelves/Waves\\_and\\_Acoustics/Sound\\_-\\_An\\_Interactive\\_eBook\\_\(Forinash\\_and\\_Christian\)/11%3A\\_Tubes/11.01%3A\\_Standing\\_Waves\\_in\\_a\\_Tube/11.1.01%3A\\_Tube\\_Resonance](https://phys.libretexts.org/Bookshelves/Waves_and_Acoustics/Sound_-_An_Interactive_eBook_(Forinash_and_Christian)/11%3A_Tubes/11.01%3A_Standing_Waves_in_a_Tube/11.1.01%3A_Tube_Resonance) [Accessed on: 2026 Feb 25]
15. Standing wave [Internet]. 2025 Dec. Available from: <https://www.britannica.com/science/standing-wave-physics> [Accessed on: 2026 May 14]
16. Russell D. Standing Sound Waves, Longitudinal Standing Waves [Internet]. 2012 [2020]. Available from: <https://www.acs.psu.edu/drussell/demos/standingwaves/standingwaves.html> [Accessed on: 2026 May 14]

17. Greenspon JE. Acoustics, Linear. *Encyclopedia of Physical Science and Technology (Third Edition)*. Ed. by Meyers RA. Third Edition. New York: Academic Press, 2003 :129–67. DOI: <https://doi.org/10.1016/B0-12-227410-5/00009-0>
18. Riley N. Acoustic Streaming. *Theoretical and Computational Fluid Dynamics* 1998; 10:349–56. DOI: <https://doi.org/10.1007/s001620050068>
19. Cooper P, Sheridan J, and Flood G. The effects of sound on forced convection over a flat plate. *International Journal of Heat and Fluid Flow* 1986; 7:61–8. DOI: [https://doi.org/10.1016/0142-727X\(86\)90044-5](https://doi.org/10.1016/0142-727X(86)90044-5)
20. Vainshtein P, Fichman M, and Gutfinger C. Acoustic enhancement of heat transfer between two parallel plates. *International Journal of Heat and Mass Transfer* 1995; 38:1893–9. DOI: [https://doi.org/10.1016/0017-9310\(94\)00299-B](https://doi.org/10.1016/0017-9310(94)00299-B)
21. Versteeg H and Malalasekera W. An introduction to computational fluid Dynamics: The finite volume method. 2nd ed. Prentice Hall, 2007
22. F. Moukalled L. Mangani MD. The finite volume method in computational fluid dynamics. Springer, 2016. DOI: 10.1007/978-3-319-16874-6
23. Zawawi MH, Saleha A, Salwa A, Hassan N, Zahari NM, Ramli MZ, and Muda ZC. A review: Fundamentals of computational fluid dynamics (CFD). *AIP conference proceedings*. Vol. 2030. 1. AIP Publishing LLC. 2018 :020252. DOI: <https://doi.org/10.1063/1.5066893>
24. Mooney K, Maric T, and Höpken J. The OpenFOAM Technology Primer [Internet]. 2021. DOI: 10.13140/2.1.2532.9600
25. Fureby C and Yu R. Introduction to MVKN70. Course lecture slides. Lecture 1, MVKN70 Advanced Methods within Numerical Fluid Mechanics and Heat Transfer. Faculty of engineering Lund University. 2025
26. Fureby C and Yu R. Turbulence and Introduction to Reacting Flow [PowerPoint presentation]. Course lecture slides. Lecture 11, MVKN70 Advanced Methods within Numerical Fluid Mechanics and Heat Transfer. Faculty of engineering Lund University. 2025
27. Scheffel J. On analytical solution of the Navier-Stokes equations [Internet]. 2001 Apr. Available from: <https://www.osti.gov/etdeweb/servlets/purl/20199349>
28. Rodriguez S. LES and DNS turbulence modeling. *Applied computational fluid dynamics and turbulence modeling: Practical tools, tips and techniques*. Springer, 2019 :197–223. DOI: <https://doi.org/10.1007/978-3-030-28691-0>
29. Smagorinsky J. General circulation experiments with the primitive equations: I. The basic experiment. *Monthly weather review* 1963; 91:99–164. DOI: [https://doi.org/10.1175/1520-0493\(1963\)091<0099:GCEWTP>2.3.CO;2](https://doi.org/10.1175/1520-0493(1963)091<0099:GCEWTP>2.3.CO;2)
30. Wang F and diMare L. Favre-averaged nonlinear harmonic method for compressible periodic flows. *AIAA Journal* 2019; 57:1133–42. DOI: <https://doi.org/10.2514/1.J057523>
31. Zhiyin Y. Large-eddy simulation: Past, present and the future. *Chinese journal of Aeronautics* 2015; 28:11–24. DOI: <https://doi.org/10.1016/j.cja.2014.12.007>
32. Lilly D. On the application of the eddy viscosity concept in the inertial sub-range of turbulence. NCAR report 1966. DOI: 10.5065/D67H1GGQ
33. Nicoud F and Ducros F. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, turbulence and Combustion* 1999; 62:183–200. Available from: <https://doi.org/10.1023/A:1009995426001>

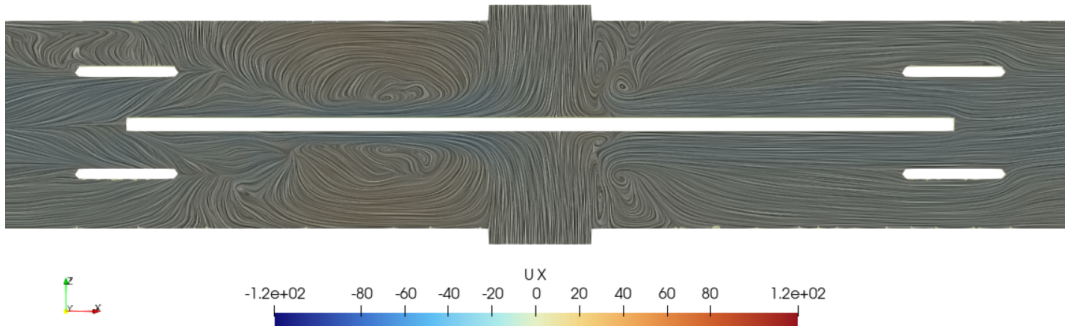
34. Kim S, Wilson PA, and Chen ZM. Effect of turbulence modelling on 3-D LES of transitional flow behind a circular cylinder. *Ocean Engineering* 2015; 100:19–25. DOI: <https://doi.org/10.1016/j.oceaneng.2015.03.014>
35. Fureby C, Tabor G, Weller H, and Gosman A. A comparative study of subgrid scale models in homogeneous isotropic turbulence. *Physics of fluids* 1997; 9:1416–29. DOI: <https://doi.org/10.1063/1.869254>
36. Hermann Schlichting (Deceased) Klaus G. *Boundary-Layer Theory*. Springer, 2017. DOI: 10.1007/978-3-662-52919-5
37. Greenshields C and Weller H. *Notes on Computational Fluid Dynamics: General Principles* [Internet]. Reading, UK: CFD Direct Ltd, 2022. Available from: <https://doc.cfd.direct/notes/cfd-general-principles/> [Accessed on: 2026 May 20]
38. Y plus wall distance estimation. CFD Online. 2011. Available from: [https://www.cfd-online.com/Wiki/Y\\_plus\\_wall\\_distance\\_estimation](https://www.cfd-online.com/Wiki/Y_plus_wall_distance_estimation) [Accessed on: 2026 Apr 1]
39. Dhari R. Y+ Calculator [Internet]. Omni calculator. 2024. Available from: <https://www.omnicalculator.com/physics/yplus> [Accessed on: 2026 Apr 1]
40. SPALDING DB. A Single Formula for the "Law of the Wall". *Journal of Applied Mechanics* 1961. Available from: <http://www.dicat.unige.it/guerrero/turbulence2021/additional/Spalding2.pdf>
41. Greenshields C. *OpenFOAM v10 User Guide - 5.2 Boundaries* [Internet]. 2022. Available from: <https://doc.cfd.direct/openfoam/user-guide-v10/boundaries> [Accessed on: 2026 Mar 31]
42. CFD Direct Ltd. *OpenFOAM v10 User Guide: fvSchemes* [Internet]. 2025. Available from: <https://doc.cfd.direct/openfoam/user-guide-v10/fvschemes#x20-1450004.5> [Accessed on: 2026 Apr 9]
43. Willcox K and Wang Q. Upwinding and the CFL condition. Massachusetts Institute of Technology. 2014. Available from: <https://dspace.mit.edu/bitstream/handle/1721.1/143656/16-90-spring-2014/contents/numerical-methods-for-partial-differential-equations/upwinding-and-the-cfl-condition/1690r-the-cfl-condition/index.htm> [Accessed on: 2026 Apr 13]
44. The Courant–Friedrichs–Lewy (CFL) Condition. 80 Years After Its Discovery [internet]. Ed. by Carlos A. de Moura CSK. Birkhäuser Boston, MA, 2012 Oct :vii–ix, 209–28. DOI: <https://doi.org/10.1007/978-0-8176-8394-8>
45. Weller H. *OpenFOAM Foundation repository for OpenFOAM version 10* [computer program]. CFD Direct Ltd. 2022. Available from: <https://github.com/OpenFOAM/OpenFOAM-10> [Accessed on: 2026 Apr 14]
46. Greenshields C. *OpenFOAM v10 User Guide - 3.5 Standard solvers* [Internet]. CFD Direct Ltd. 2022. Available from: <https://doc.cfd.direct/openfoam/user-guide-v10/standard-solvers> [Accessed on: 2026 Apr 21]
47. Gaur N, Srinadhi K, and T. KB. Comparative studies of solvers on compressible axisymmetric jet using OpenFOAM. *AIP Conference Proceedings* 2022 Nov; 2516:30–6. DOI: 10.1063/5.0109490
48. J R Martins J V Barbosa FPS and Silva LFLR. Computational analysis in underexpanded jets simulations. *Fluid Dynamics Research* 2022 Dec; 54. DOI: 10.1088/1873-7005/aca357

49. Zadeh N and Paoli R. Numerical Experiments of Subsonic Jet Flow Simulations Using RANS with OpenFOAM. Open Journal of Fluid Dynamics 2022 Jun; 12:230–47. DOI: 10.4236/ojfd.2022.122011
50. Sagaut P. Large Eddy Simulation for Incompressible Flows. An Introduction. Springer Berlin, Heidelberg, 2005 Dec :124, 129, 174–5. DOI: <https://doi.org/10.1007/b137536>
51. E. Garnier N. Adams PS. Large Eddy Simulation for Compressible Flows [Internet]. Springer Dordrecht, 2009 Aug :30–1, 193–5. DOI: <https://doi.org/10.1007/978-90-481-2819-8>

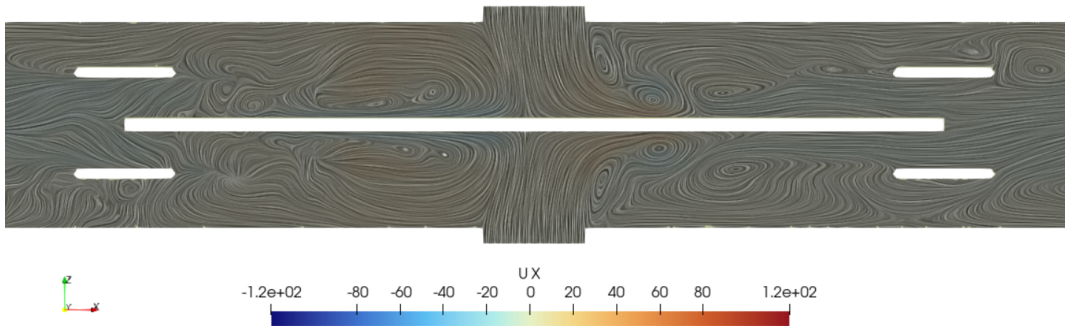
# A $U_x$ with SLIC in the y-normal plane



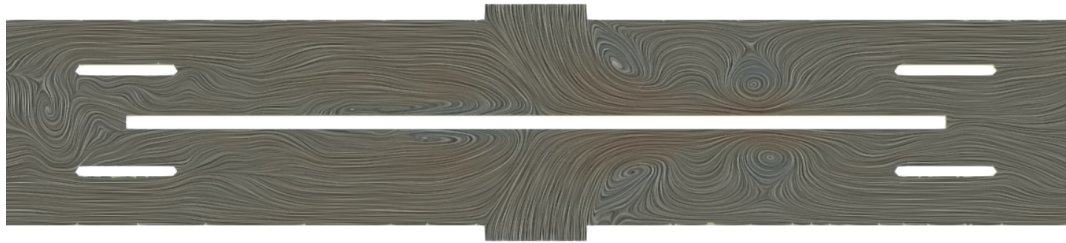
(a)  $t = 0.485$  s



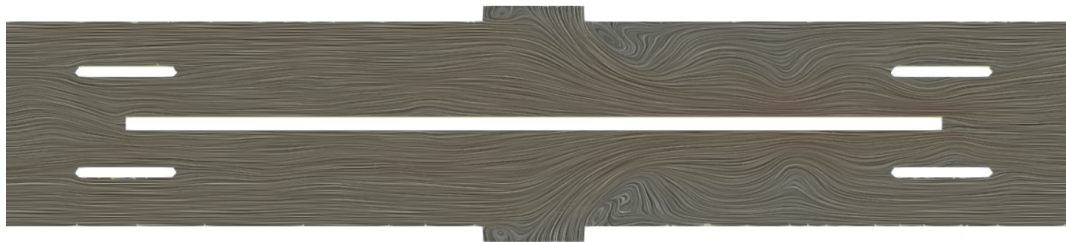
(b)  $t = 0.489$  s



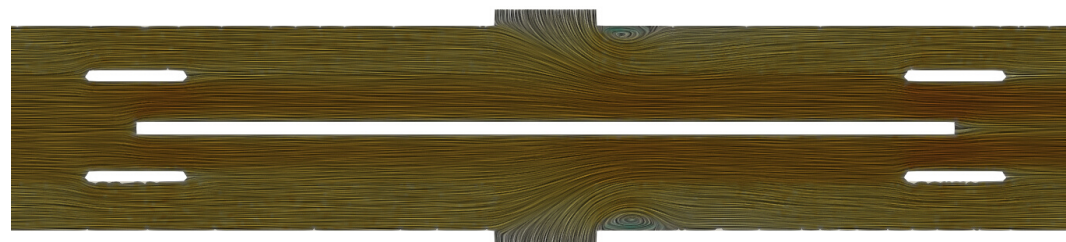
(c)  $t = 0.493$  s



(d)  $t = 0.497$  s



(e)  $t = 0.501$  s



(f)  $t = 0.514$  s

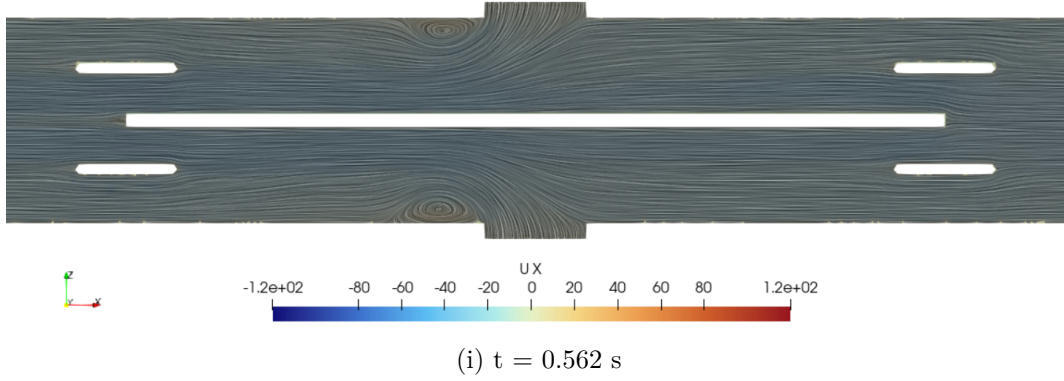
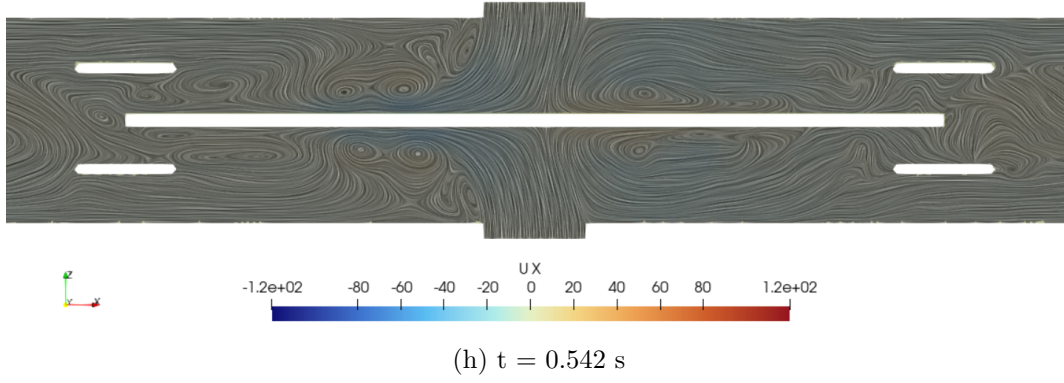
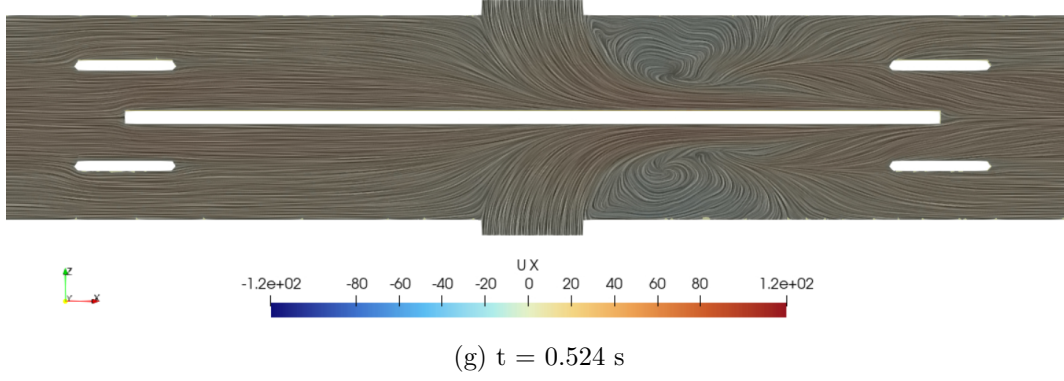
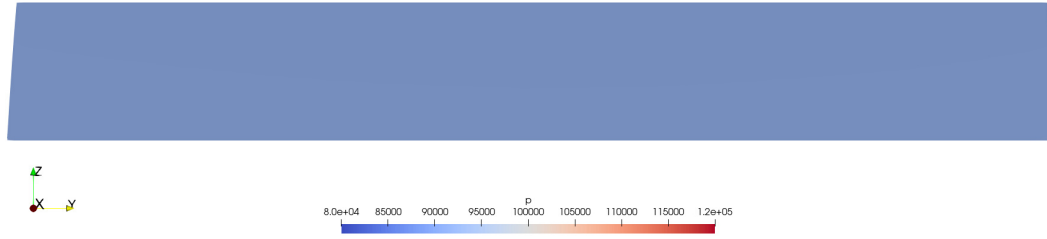


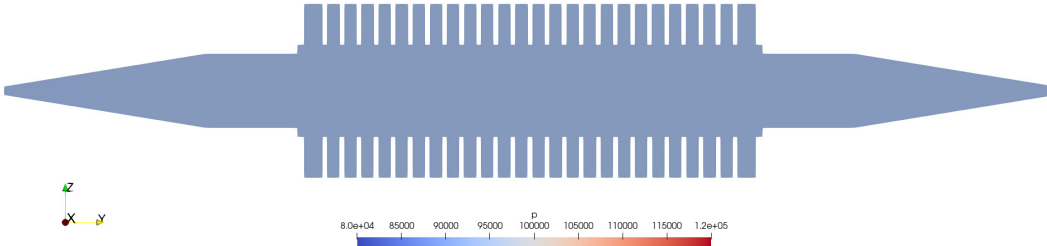
Figure 31: Velocity in the  $U_x$  direction with surface line integral convolution applied. Figures (a) through (i) show time 0.485 - 0.562 s. The colour gradient is fixed between -120 and 120 m/s for consistent colouring.



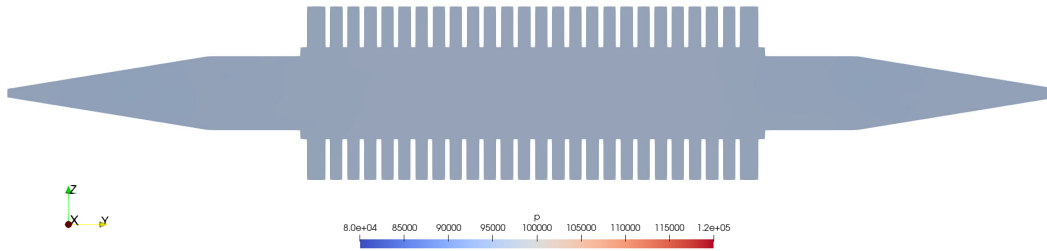
## B $P$ at $t = 0.493$ s in different x-normal cross sections



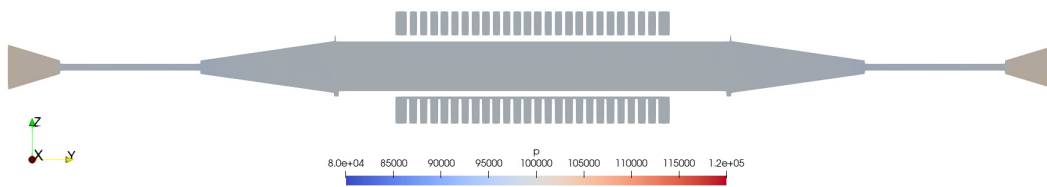
(a)  $t = 0.493$  s, pressure, 2.5 meters from centre.



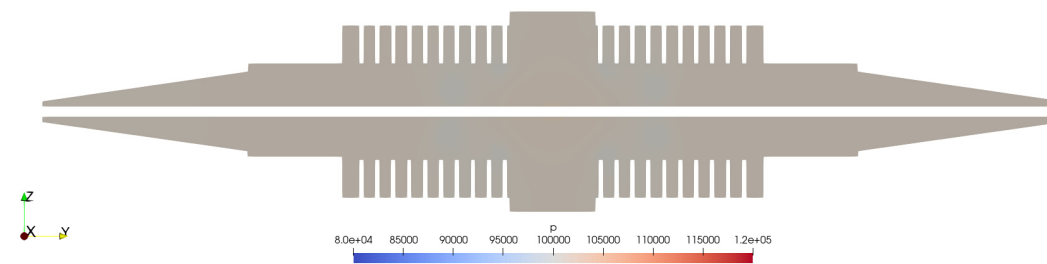
(b)  $t = 0.493$  s, pressure, 2 meters from centre.



(c)  $t = 0.493$  s, pressure, 1.5 meters from centre.

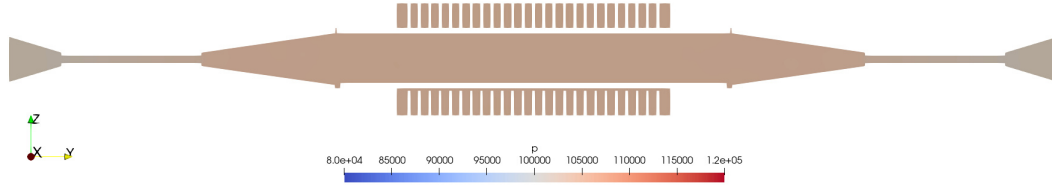


(d)  $t = 0.493$  s, pressure, 0.5 meters from centre.

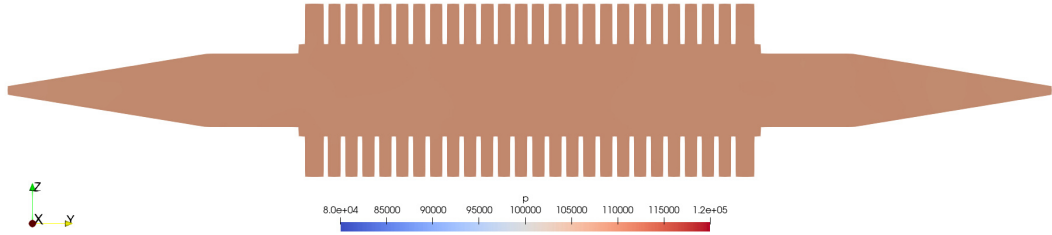


(e)  $t = 0.493$  s, pressure, at the centre.

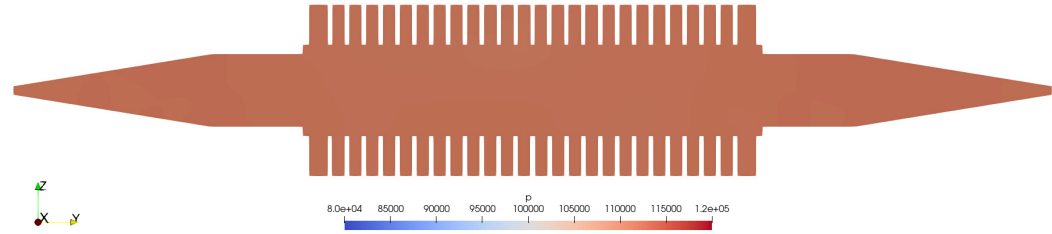




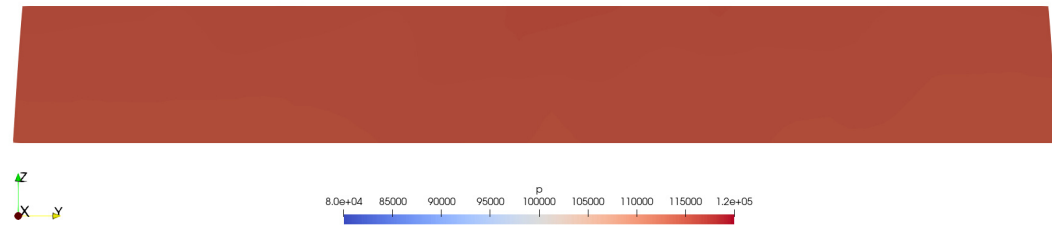
(f)  $t = 0.493$  s, pressure, -0.5 meters from centre.



(g)  $t = 0.493$  s, pressure, -1.5 meters from centre.



(h)  $t = 0.493$  s, pressure, -2 meters from centre.



(i)  $t = 0.493$  s, pressure, -2.5 meters from centre.

Figure 32: Pressure field at  $t = 0.493$  s, normal to the x-direction with slices at different locations.

# C $U_x$ at $t = 0.494$ s in different z-normal cross sections

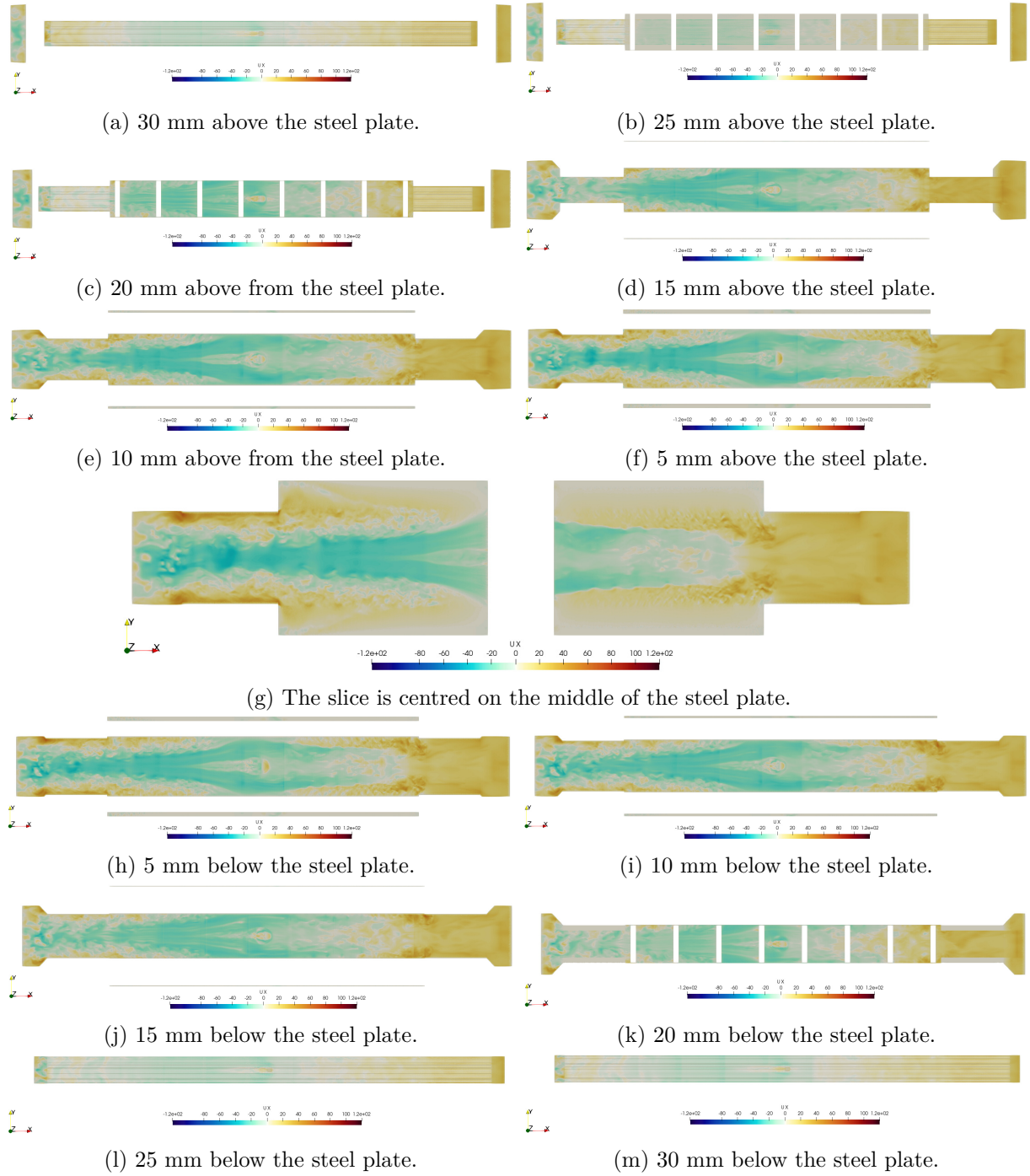


Figure 33: Slices with normal in the z-direction, at timestep  $t = 0.494$  s. Slices presented in a top down order, showing  $U_x$ .

## D Miscellaneous

### D.1 Energy spectrum as a function of data frequency

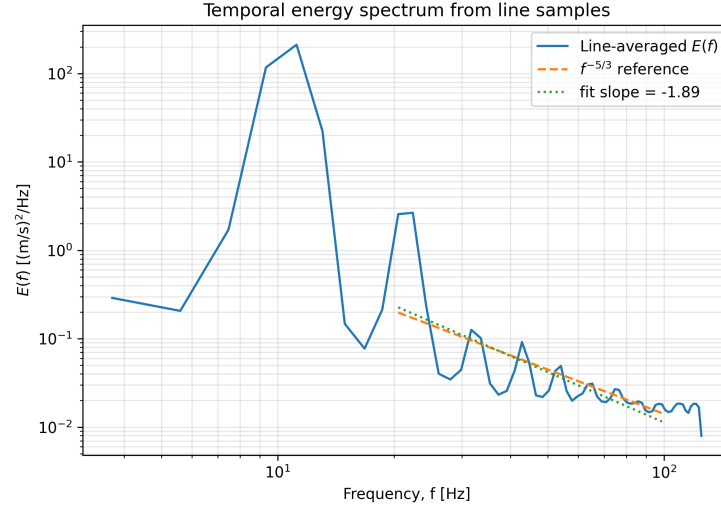


Figure 34: The energy spectrum as a function of data frequency, with the theoretical curve  $f^{-5/3}$  and the fitted curve between 20 - 100 Hz. Based on data range 0 - 0.532 s, time steps at 0.004 s interval.

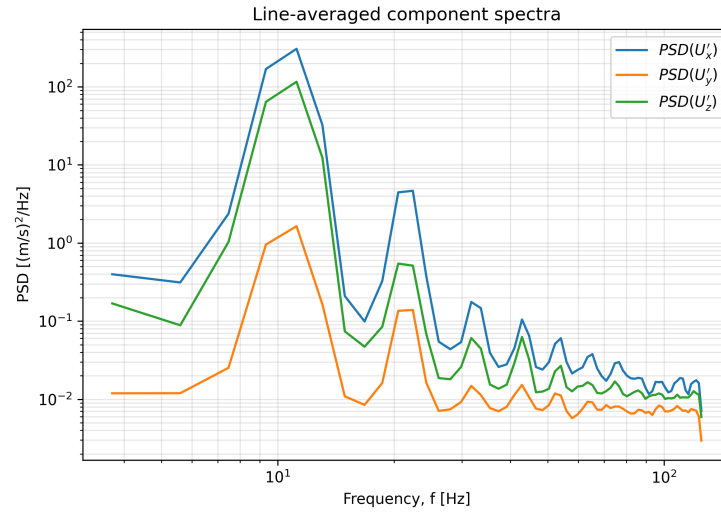


Figure 35: Energy spectrum split by velocity component. Based on the same data as the figure above.

## D.2 Project Timeline

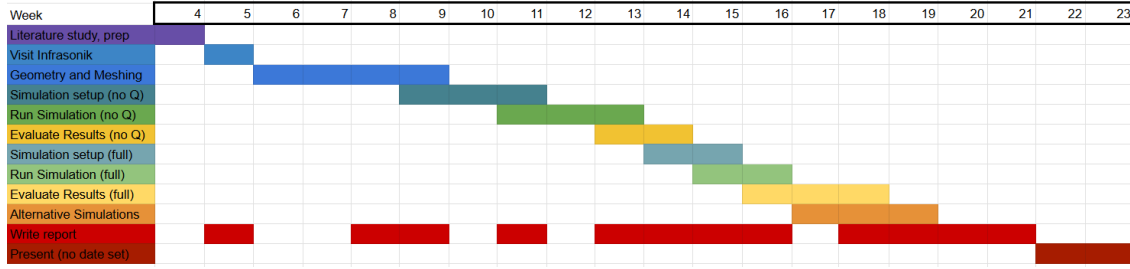


Figure 36: Gantt schedule of planned work over a 20 week timeline.

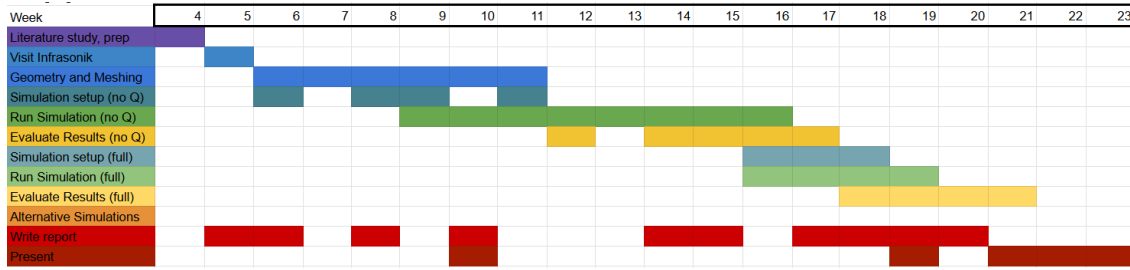


Figure 37: Gantt schedule of the actual work over a 20 week timeline.

## D.3 Division of work

This project has been carried out jointly by both authors, with all major tasks performed collaboratively. This includes the handling of the geometry, meshing, numerical setup, post-processing, analysis and writing of the report. Consequently, no strict division of responsibilities has been applied, as challenges, methodologies and results were discussed and approached cooperatively.