Brolin Daniel, 940901-5014, danbro-3@student.ltu.se

Kask Nathalie, 930722-3686, natkas-2@student.ltu.se

Last revision: March 7, 2018

Luleå Tekniska Universitet
F7024T Multifysik, simulering och beräkning

Assignment 4: Buoyant convection a multiphysics problem

With supervisor
Hans Åkerstedt

Abstract

This work contains the result and analysis of the fourth COMSOL-simulative exercise where the buoyant convection of water and air is simulated within a small squared surface where the left and right wall have different temperatures, creating a circulation of the water which speed varies with the temperature difference.
The first simulation to find the maximum temperature difference, with water as a medium, for when the system will no longer converge was 27.5 Kelvin, giving a Rayleigh number of $3.5 * 10^{10}$ and a Grashof number of $5.9 * 10^6$, confirming this is most likely due to the system leaving the laminar region.
Simulating the system with air instead of water resulted in the expected commensurable conclusion where the temperature needed to be vastly larger than that of water. Using the linearly increased calculation time as "proof" it is believed the system might not converge due to calculation being to resource heavy rather than the system reaching the transition to turbulence. The Rayleigh and Grashof numbers are $1.6 * 10^6$ respectively $.54 * 10^6$.
The time-dependent simulation of water needed about $200s$ to reach stability.

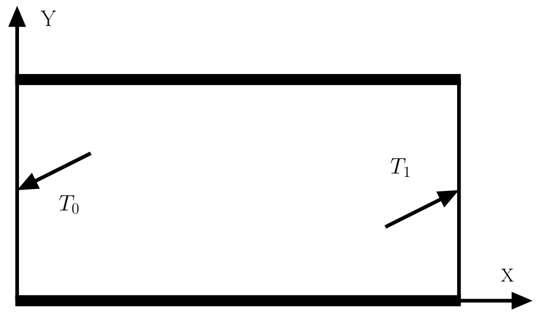


Figure 1: Observed system, where T_0 represents the lower temperature and T_1 the higher. The thick lines represents perfect thermal insulation and the box is filled with observed fluid or gas.

1 Introduction

COMSOL Multiphysics® is a general-purpose software platform, based on advanced numerical methods. It is a powerful tool useful to simulate flows; fields; forces and such in models provided either by files or built directly in COMSOL.

This report is a part of a written documentation of the COMSOL-laboratory exercises made in the course Multiphysics, Simulation and Computation at Luleå University of Technology. These exercises serve as practice in formulating mathematical models to describe physical and technical problems in a way that is suitable for implementation of the finite element method.

This work contains the result and analysis of the fourth COMSOL-laboratory exercise where the momentum of a fluid, in this case water and air, is studied when acted upon by two areas of different constant temperatures. A model of the simulated system can be seen in fig. 1.

The first study will be using water as the observed fluid. It will be modeled as a laminar compressible flow. The simulation will start with a temperature difference of zero, and then be increased until the solution no longer converges. What will be observed is the velocity field using an arrow surface overlay, we will also try to look at how the temperature will eventually spread out in the system. The solution will no longer converge when the difference is large enough that the natural convection layer becomes turbulent; which will be deemed a true hypothesis if we can calculate the approximate Rayleigh number required for turbulence as simulated results for the greatest possible temperature difference.

When the maximum temperature difference has been found a sweep, $\Delta T \epsilon [0, \delta T, T_{max}]$, will be run and some instances will be chosen to show the different flows in the stable systems. This should be at around $Ra \approx 10^9$[1]. An alternative way to calculate turbulence is the "Grashof number"[2][3]. Grashof indicates laminar conditions for $10^3 < Gr < 10^6$, transition to turbulent flow for natural convection in the range $10^8 < Gr < 10^9$ and undefined conditions in the range of $10^6 < Gr < 10^8$. The equation describing the Grashof number is rather similar to the Rayleigh number, see eq. (3). The hypothesis of turbulence being the sweep limit will be proved by calculating the Grashof number for the maximum temperature difference and check if it is outside the laminar range or not. In the assignment paper two more formulas were provided in the instructions[6]. These are the following eqs. (1) and (2) which can be used to calculate the Reynold number and thermal expansion, see eq. (4). Specific material values can be found in table 1.

$$Ra = \frac{\rho_0 g \alpha \Delta T h^3}{\nu \kappa} \tag{1}$$

$$\rho(T_1) = \rho_0(1 - \alpha(T_1 - T_0)) \tag{2}$$

$$Gr = \frac{g \alpha \Delta T h^3}{\nu^2} = \frac{\kappa}{\rho_0 \nu} Ra \tag{3}$$

which gives

$$\alpha = \frac{1 - \rho(T_1)/\rho_0}{T_1 - T_0} \tag{4}$$

Material	α	ΔT_{max}	ν	$\rho_0\|_{T=25^oC}$	κ
H_2O	$1.273 * 10^{-4}$	27.5^oC	$0.8539 * 10^{-6}$	997.05[4]	$0.143 * 10^{-6}$
air	$\approx 1/\Delta T$	$3E8^oC$	$47.54 * 10^{-6}$	1.1849[5]	$1.9 * 10^{-5}$

Table 1: Material constants

where α is the thermal expansion of the fluid, g is the gravitational constant, T is temperature, h is length, ν kinetic viscosity and κ thermal diffusivity[7].

The second study will be exactly the same as the first but simulating air instead of water. The maximum temperature difference before turbulence will be calculated for air as well, and since gases can move around easier than liquids it will be hypothesized the temperature difference, i.e. the fluid speed, will be vastly greater than that of water.

The third and last study will be to run this problem as a time dependent problem, exporting a .gif to illustrate the system and note the time scale as it moves towards stability.

2 Method

2.1 General

The exact method to calculate the coefficients and simulate the system is detailed and well explained in the instructions[6], but the general way to go about it is the same as most COMSOL projects.

1. Choose system type (fluids, laminar, $2D$).
2. Introduce global parameters.
3. Build geometry.
4. Set study specifications for your system type:
    - Set fluid parameters.
    - Set Boundary conditions.
    - Set initial conditions.
5. Build a mesh grid of your geometry.
6. Compute system.

Three global parameters were added. $T0$, $T1$ and $delT$ where $T0 = 293.15[K]$, $delT = 1[K]$ and $T1 = T0 + delT$. This way $delT$ could be easily sweeped and changed to control the difference in temperature. If any equation relating the speed of the flow to the difference in temperature could be found, this would have been very helpful in calculating the limiting speeds for convergence. Furthermore two studies were created, "Study h20" and "Study Air" to be run separately. Within these three sub-studies were added; one stationary step; one parametric sweep and one time dependent. This way Studies and sub-studies could easily be toggled on and off to produced any wanted result, and sweeps could easily be edited without interaction.

2.2 Stationary

When the largest possible temperature was found the Rayleigh number could be calculated using the provided eq. (1) within the global evaluations. table 1 with eqs. (1) and (2) gives some resulting Reynold numbers for water and air, see eqs. (5) and (6).

$$Ra_{water} = \frac{996 * 1.273 * 10^{-4} * 9.81 * 27.5 * .05^3}{.8539 * .143 * 10^{-12}} \approx 3.5 * 10^{10} \quad (5)$$

$$Ra_{air} = \frac{1.2 * 10^{-5} * 9.81 * .05^3}{47.54 * 1.9 * 10^{-11}} \approx 1.6 * 10^6 \quad (6)$$

To confirm the water simulation stops converging at $\approx 27.5^oC$ we also calculate the Grashof number according to eq. (3).

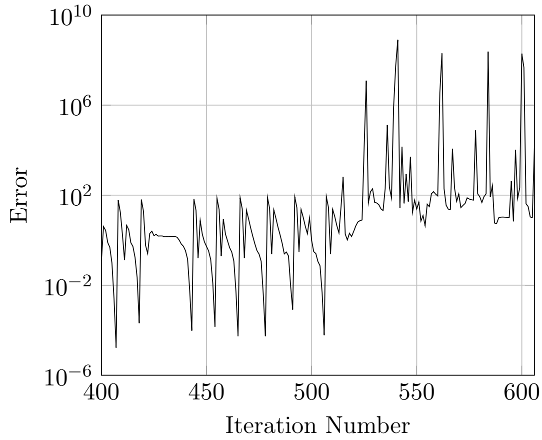
$$Gr_{water} = \frac{9.81 * 1.273 * 10^{-4} * 27.5 * .05^3}{(.8539 * 10{-}6)^2} \approx 5.9 * 10^6 \tag{7}$$

$$Gr_{air} = \frac{1.9 * 10^{-5}}{1.18 * 47.5 * 10^{-6}} 1.6 * 10^6 \approx .54 * 10^6 \tag{8}$$

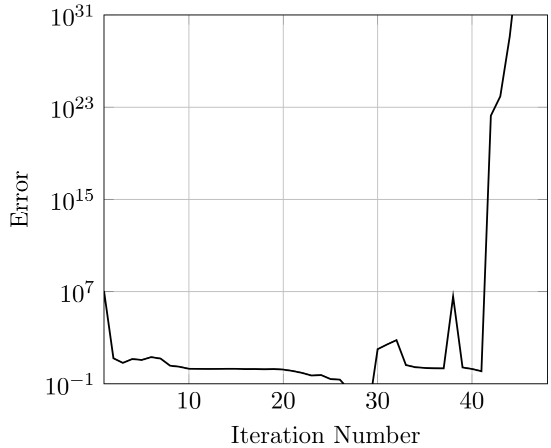
The sweeped intervals were incrementally increased until the simulation did not converge, either due to the solution being to complex or no longer laminar. In fig. 2 the last 10 or so ΔT before system didn't converge can be seen, and the last two tested numbers within the air-study.

2.3 Time Dependency

Instructions were followed also to generate a gif that will be included in this pdf if technology allows it. This was to visualize how long it took the system to assume a relatively steady state.



(a) water



(b) air

Figure 2: Convergence study, breakingpoints

3 Results and interpretation

3.1 Stationary solution, water

After running the simulations, the maximum working temperature difference was read at $\Delta T = 27.5^oC$. This gave a calculated Rayleigh number of $Ra_{water} \approx 3.5 * 10^{10}$ and Grashof number of $Gr_{water} \approx 5.9 * 10^6$. This is right outside the laminar region. It can be assumed then the hypothesis is correct and the solution no longer converges as the simulation settings specified laminar conditions.

Looking at the simulated velocity field, see fig. 3, there is a very obvious correlation between the velocity of the field and the temperature difference between the fields. An interesting further study would be to simulate how the speed correlates to the temperature in the turbulent area. This could possibly be useful for things like transportation in water, but plotting the fuel efficiency to speed and friction is a whole different project.

Looking at the simulated temperature field, see fig. 4, the effects of the velocity field is clear. The reason the velocity field is counterclockwise is because the heated fluid wants to move on the top, and the colder on bottom.

The larger the difference in temperature the more does the system crave entropy, straightening the heat spread out resulting in a larger velocity.

3.2 Stationary solution, air

Running the simulations, the maximum working temperature difference of air was read at $3*10^{8o}C$. This gives $Ra_{air} = 1.6*10^6$ and $Gr_{air} = .54 * 10^6$. This is most definitely in the laminar region, but the solution still does not converge. Presumably simulations larger than this requires much more computation time than standard settings allow. There is also a possibility the system will never become turbulent due to natural convection, since the density of air decreases almost logarithmically with temperature[8], with $\rho_{air,T=3E8} \approx 1.1 * 10^{-6}$.

Looking at the simulated velocity field, see fig. 5, it's clear that the lower viscosity of a gas allows it to flow more freely than the liquid, water. Already at 10^oC a clear circulation can be observed, and temperature differences $100 - 1000^oC$ shows that the unmoving layer between hot and cold flow we could observe in water is thin to non-existent with air as a medium.

Looking at the simulated temperature field, see fig. 6, we observe something completely different from the previous case with water. Instead of going from a rather diffuse spread at low temperatures to a clear separation at high temperatures, which was the case of water, we observe the opposite. The low temperatures allowed for a good circulation with a clearer separation of hot and cold, while the large difference in temperature shows an almost perfect gradient from hot to cold. As to why, it remains unclear; maybe this is the result of what in fluids would be a turbulent flow, or the thermal diffusion and thermal expansion somehow causes this.

3.3 Time domain

The time dependent solution reached steady state in approximately 200 s. The simulation was run for 400 s, with a frame step of 10 s, and animated with 10 f/s making the gif run for a total of 4 s.

As including moving images in a report is ill advised for compatibility issues the gif will be appended with the report.

References

[1] Thermopedia. Rayleigh number. http://www.thermopedia.com/content/1076/, 2018.

[2] Wikipedia. Natural convection. https://en.wikipedia.org/wiki/Natural_convection, 2018.

[3] Wikipedia. Grashof number. https://en.wikipedia.org/wiki/Grashof_number, 2018.

[4] The Engineering Toolbox. Density of water. https://www.engineeringtoolbox.com/water-density-specific-weight-d_595.html, 2018.

[5] Wikipedia. Density of air. https://en.wikipedia.org/wiki/Density_of_air, 2018.

[6] Multiphysics F7024T Hans Åkerstedt. Assignment #4, buoyant convection a multiphysics problem. Technical report, Department of Engineering Science and Mathematics, April 2017.

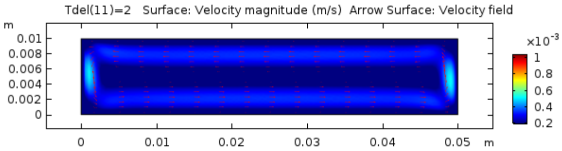
[7] Wikipedia. Thermal diffusivity. https://en.wikipedia.org/wiki/Thermal_diffusivity, 2018.

[8] Steve Gribble. Air density calculator. https://www.gribble.org/cycling/air_density.html, 2018.

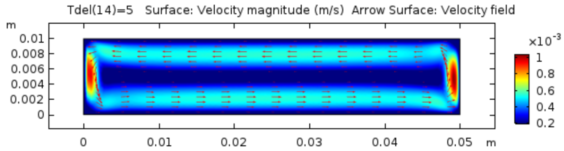
A APPENDIX

A.1 Resulting Images

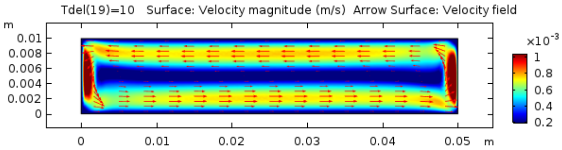
A.1.1 Static sweep, H_2O



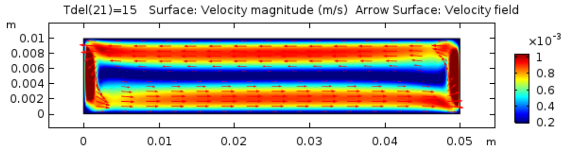
(a) $\Delta T = 2^oC$



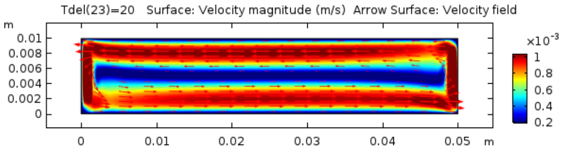
(b) $\Delta T = 5^oC$



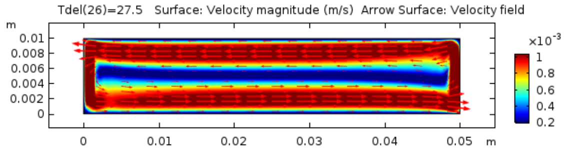
(c) $\Delta T = 10^oC$



(d) $\Delta T = 15^oC$



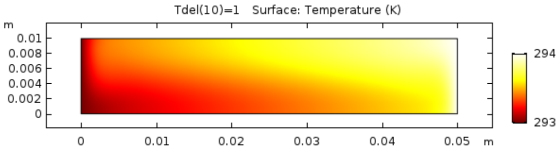
(e) $\Delta T = 20^oC$



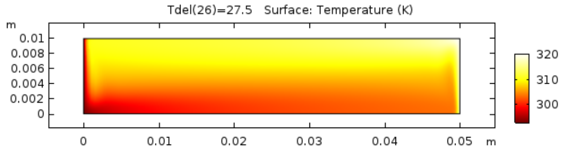
(f) $\Delta T = 27.5^oC$

Figure 3: Static velocity field, H_2O, for $\Delta T = [2, 27.5]^oC$

A.1.2 Temperature, H_2O



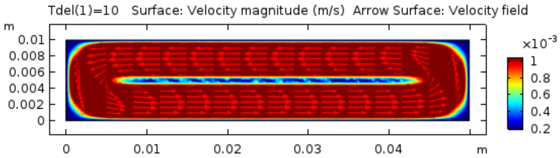
(a) $\Delta T = 2^oC$



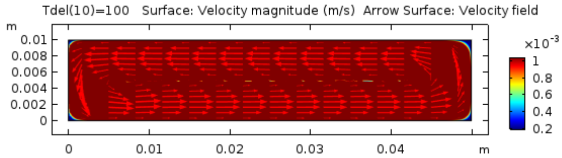
(b) $\Delta T = 27.5^oC$

Figure 4: Temperature over area at $\Delta T = 2, 27.5$

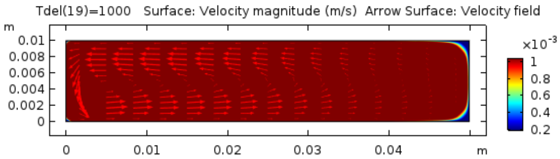
A.1.3 Static sweep, air



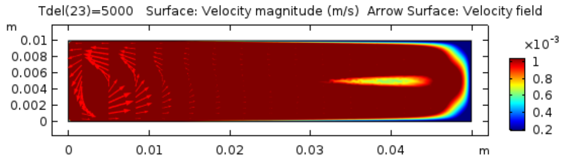
(a) $\Delta T = 10^oC$



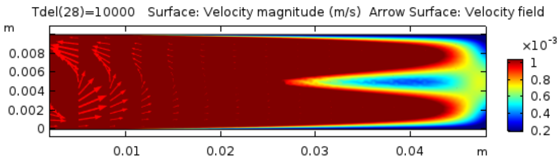
(b) $\Delta T = 100^oC$



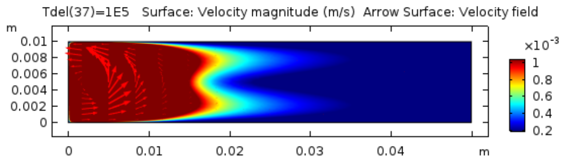
(c) $\Delta T = 1E3^oC$



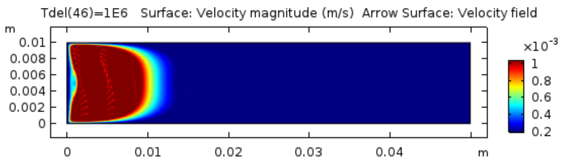
(d) $\Delta T = 5E3^oC$



(e) $\Delta T = 1E4^oC$



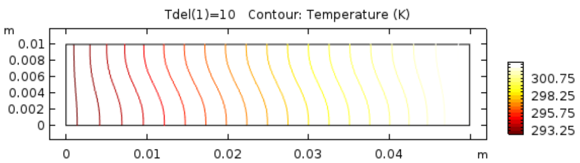
(f) $\Delta T = 1E5^oC$



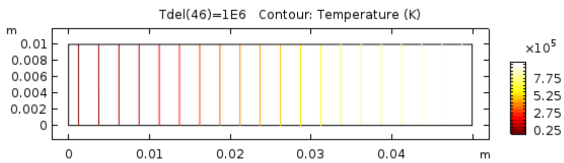
(g) $\Delta T = 1E6^oC$

Figure 5: Static velocity field, air, for $\Delta T = [10, 1E6]^oC$

A.1.4 Temperature, air



(a) $\Delta T = 10^oC$



(b) $\Delta T = 1E6^oC$

Figure 6: Temperature over area at $\Delta T = 10, 1E6$