Lab Exercice on Computational Fluid Dynamics

Natural convection from a cylinder in a square cavity

Instructors: Maximilien Danda maximilien.danda5@etu.univ-lorraine.fr,
Olivier Botella olivier.botella@univ-lorraine.fr

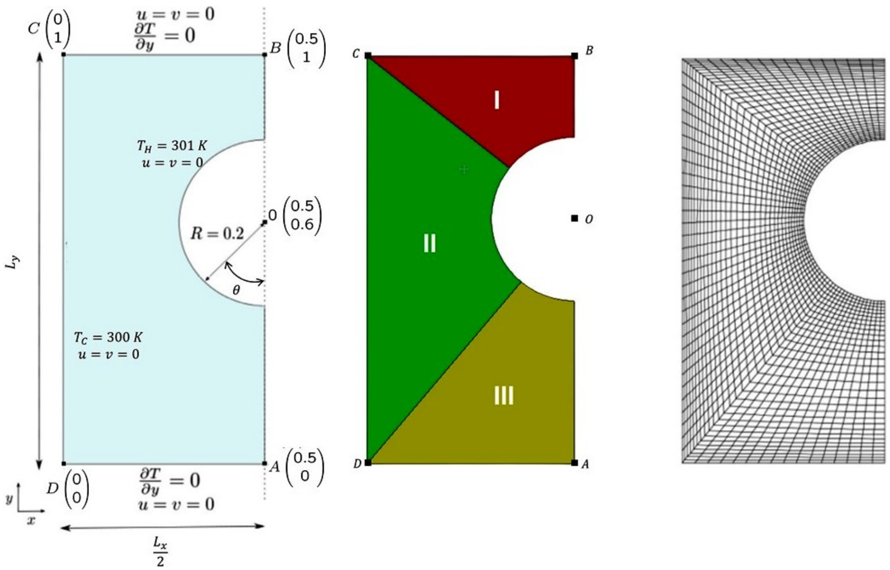

Figure 1: *Left* : Computational domain and boundary conditions. *Center*: Partition of the domain domain for mesh generation. *Right*: Curvilinear mesh used. Domain dimensions are given in meter (m) with $L_x = L_y = 1$ m.

1 Problem statement

This lab exercise concerns the computation of the natural (or free) convection flow generated by a heated cylinder placed in a square cavity, for which Demirdžić *et al.* [5] published highly accurate numerical solutions. In this problem, the heat equation is coupled with the Navier-Stokes equations in the Boussinesq approximation (see appendix). The computational domain and boundary conditions are shown in Fig. 1 (left). Since the flow displays a symmetry along the vertical axis AB, only half of the geometry is considered. The top and bottom walls of the domain are adiabatic and the temperatures of the circular cylinder and the right wall are respectively $T_H = 301$K and $T_C = 300$K. The gravity force $\rho \boldsymbol{g} = \rho g \boldsymbol{e}_y$ is in the downward direction, with $g = -1\ \mathrm{m \cdot s^{-2}}$. The fluid used in this exercise has the following custom thermophysical properties in SI units :

- Density $\rho_0 = 1\ \mathrm{kg \cdot m^{-3}}$,
- Specific heat $c_p = 1\ \mathrm{J \cdot K^{-1} \cdot kg^{-1}}$.
- Thermal conductivity $\kappa = 10^{-4}\ W \cdot \mathrm{m^{-1} \cdot K^{-1}}$,
- Dynamic viscosity $\mu = 10^{-3}\ \mathrm{kg \cdot m^{-1} \cdot s^{-1}}$,
- Thermal expansion coefficient $\beta = 0.1\ \mathrm{K^{-1}}$.

For comparing the results with [5], all reported quantities will be non-dimensionalized with the following reference values :

- $L_{\mathrm{ref}} = L_x = 1\,\mathrm{m}$,
- $T_{\mathrm{ref}} = T_C = 300\,\mathrm{K}$,
- $S_{\mathrm{ref}} = L_x \times 1 = 1\,\mathrm{m}^2$.

Note that the appendix gives a summary of all the notions on heat transfer in fluids used in this lab exercise, and their definitions in ANSYS Fluent. Read this appendix carefully while performing the exercise.

1.1 Objectives of the lab exercise

The configuration of Fig. 1 is a typical example of the mechanisms of room heating by natural convection (see Appendix A.2), where air motion is triggered by the buoyancy forces[1] generated by the heated cylinder. One of the main objectives of the exercise is to compute the total heat flux Q (measured in watt W) exchanged at the surface of the heated cylinder.

In ANSYS Fluent, the Total Surface Heat Flux[2] at a surface S is defined by :

$$Q = \int_S \boldsymbol{q} \cdot \boldsymbol{n}\, \mathrm{d}S, \tag{1.1}$$

where the (local) density of heat flux $\boldsymbol{q}$ (in $\mathrm{W \cdot m^{-2}}$) is given by Fourier's law :

$$\boldsymbol{q} = -\kappa \nabla T. \tag{1.2}$$

A surface heat flux is usually reported in dimensionless form as the Nusselt number Nu, which is the ratio of convective to conductive heat fluxes (see Appendix A.2.2). For the flow of Fig. 1, Demirdžić *et al.* have performed in [5] a series of highly accurate computations and have calculated with a 7-digit accuracy the Surface Nusselt Number (in m^2) at the surface S of the half-cylinder :

$$Nu\, S = 7.38376\,\mathrm{m}^2. \tag{1.3}$$

The aim of this lab exercise is to perform this numerical simulation with ANSYS Fluent and compare your results with the benchmark results of [5].

Question 1. *Calculate the Prandlt and Rayleigh numbers of the flow (see appendix). Deduce the principal features of the flow, and verify that the Boussinesq approximation of the flow equations is valid.*

[1] En français : forces de flottabilité ou poussée d'Archimède.
[2] ANSYS Fluent expressions are written in sans serif font.

2 Geometry creation with DesignModeler

The computational domain of Fig. 1 (center) will be decomposed in 3 parts (or blocks) as shown in Fig. 2, in order to build a high quality curvilinear mesh in each part. To do so, each part needs to be created in a separate sketch (use the New Sketch tool).

The contour of Part 1 is created by first drawing triangle OBC and the circular arc of center O, then removing the excess lines. Here are the steps for sketching Part 1 :

Step 0) Go to the Settings toolbox to apply the appropriate settings for this sketch : Major Grid Spacing : 0.1 m, Minor-Steps per Major : 5. Show the grid and toggle the Snap option for aligning the drawings to the grid lines.

Step 1) Sketching of triangle OBC : Draw segments BC and BO and set their dimension. Draw segment OC.

Step 2) Draw a circular arc between OC and OB using the Arc by center tool and set the radius value.

Step 3) Remove excess lines using the Trim tool.

Once Part 1 is drawn, apply the same strategy for the other parts in distinct sketches. Here are some tips for the geometry creation :

- It is important that each sketch forms a closed surface.
- Set the correct dimensions of the triangle and the circular arc before using the Trim tool.
- For creating the surface sketches, use the Add Frozen operation to keep the 3 surfaces separated.

Question 2. *To check that the generated geometry is correct, verify that its area A_{geom} (in m^2) is equal to its analytic value :*

$$A_{geom} = \frac{(L_{ref}^2 - \pi R^2)}{2}.$$

Display the geometry in your Word report.

3 Mesh generation with Ansys Meshing

The purpose of this section is to generate the curvilinear mesh shown in Fig. 1 (right), which is refined near the boundaries of the domain, in particular near the left wall and the cylinder surface to account for the thin thermal boundary layers.

A curvilinear grid is a structured grid for complex domains where the grid lines are curved to fit the boundaries of the physical domain. Such grids are built by coordinate transform of simple Cartesian domains, as shown in Fig. 2. To ease the coordinate transform performed by the meshing software, the physical domain is decomposed in 3 blocks, such as each is block is meshed with a "bottom up" strategy starting with the edge meshing.

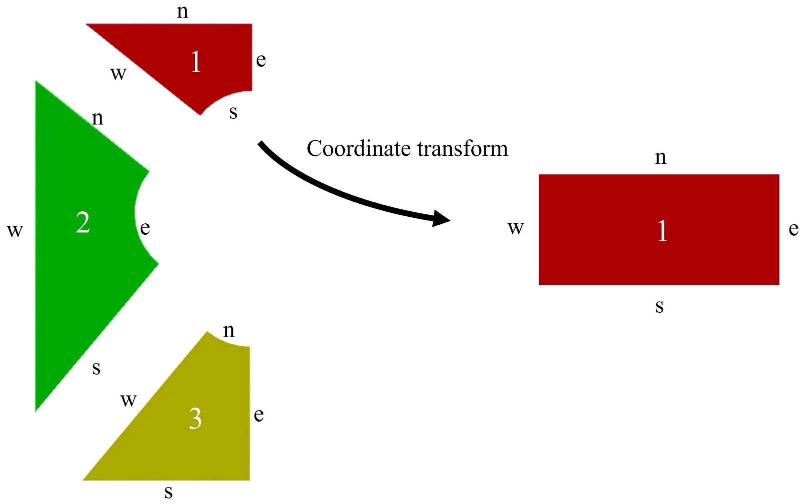

Figure 2: Edge labelling of the 3 blocks of the curvilinear grid.

		♯ of divisions	Bias type	Bias factor	Name
I	n	64	X	No bias	wall_top
	e	128	- --- ----- --- -	3	symmetry
	s	64	X	No bias	wall_h
	w	128	- --- ----- --- -	3	X
II	e	128	X	No bias	wall_h
	s	128	- --- ----- --- -	3	X
	w	128	X	No bias	wall_c
III	n	64	X	No bias	wall_h
	s	64	X	No bias	wall_bottom
	e	128	- --- ----- --- -	3	symmetry

Table 1: Sizing details for the edges shown in Fig. 2. Number of Divisions corresponds to the number of edges vertices N. Bias Type creates a graded mesh near the edge boundaries, where the Bias Factor is the ratio of the largest element to the smallest element.

Guidelines for mesh creation :

- The sizing details of the edges are given in Table 1.
- Use Behavior : Hard for all edges[3].
- Use quadrangular cells for the face meshing.
- For the ease of the solution setup in Fluent , give meaningful names to the external edges of the domain with the Named Selection tool.

> **Question 3.** *Display the mesh in your* Word *report. Calculate the expected number of cells of each block and verify that the total is correct in the* Mesh Statistics *of* Ansys Meshing .

4 CFD analysis with ANSYS Fluent

This part is dedicated to the computation and the analys of the results on the natural convection flow. Follow the guidelines of the previous lab exercises. Only the new steps will be detailed below.

4.1 Step 1 : Solution setup

In this preliminary stage, you will define the flow physics (flow equations, working fluid, cell zone and boundary conditions, operating conditions, reference values, ...), select the iterative solver and the discretization schemes.

4.1.1 Boussinesq setup

The setup of the Boussinesq model of Appendix A.2.1 amounts to set the values of the various parameters in the momentum equation (A.6) and the energy equation (A.7).

- The gravity acceleration $\boldsymbol{g} = g\boldsymbol{e}_y$ is defined in the General task page. It can also be set in the Operating Conditions panel of the Physics menu bar.
- The thermophysical properties of the custom Boussinesq fluid of Section 1 are set by creating a new material in the Materials task page. In the Create/Edit Materials panel, use the boussinesq property for the density with the value of ρ_0.
- The average temperature T_0 corresponds to the Operating Temperature (k) in the Operating Conditions panel. Set its value according to the guidelines of Appendix A.2.1.

[3] In contrary to Soft, the Hard behavior overrides the Max face Size and other sizing properties used elsewhere in the mesh.

4.1.2 Guidelines for the solution setup

- General task page : Use the steady pressure-based solver and set the gravity acceleration.
- Models task page : Use the Viscous (laminar) model and activate the energy equation.
- Materials task page : Define the custom Boussinesq fluid and use it in the Cell Zone Conditions.
- Boundary Conditions task page : Set the boundary conditions of Fig. 1 (left) for both momentum and energy equations. Note that a Neumann boundary condition for temperature amounts to prescribing the heat flux density $\boldsymbol{q}$ (1.2).
- Reference Values task page : Set the reference values given in Section 1. They will be used used for reporting the dimensionless quantities of Appendix A.3.

4.2 Step 2 : Flow computation

Here are some advice for the solution methods to use :

- Use the SIMPLE scheme for pressure-velocity coupling.
- Use the most accurate gradient interpolation available : for ANSYS V2020 and later versions, Least Squares Cell Based is recommended.
- For high-Rayleigh number natural convection, Body Force Weighted is the recommended pressure interpolation scheme.
- Use a second-order method for convection such as the QUICK scheme for all fluid flow and thermal equations.

Note that the choice of the discretization methods and the version of ANSYS may have an impact on the last digits of the alphanumeric data reported in Question 4 to 7, but will not change the outcome of the exercise. In your Word report, report your results with a 7-digit accuracy as in (1.3).

4.2.1 Initializing the natural convection flow simulation

Actual Problem	Initial Condition
Heat Transfer	Isothermal
Natural convection	Low Rayleigh number
Combustion / reacting flow	Cold flow (no combustion)
Turbulence	Inviscid (Euler) solution

Table 2: Best practices for initializing coupled flow simulations.

The simulation of fluid flows coupled with other transport phenomena like heat or mass transfer can be plagued with numerical convergence difficulties. To mitigate these issues, it is customary to start the computation with a “simplified” version of the flow, for example by using the isothermal flow as the initial condition of a coupled heat and flow problem. The best practices for solving coupled problems are summarized in Table 2. However, in this lab exercise the convective flow motion is induced by the temperature differences in the domain : the heat equation should be solved first.

> **Question 4.** *Solve the heat transfer problem by conduction only (Tip : set the gravity acceleration to $g = 0$ in order to decouple the energy equation* (A.7) *to the momentum equation* (A.6)*). Verify there is no flow motion and display the isothermal lines in the computational domain (Tip : Use* Colormap Size=10 *in the* Colormap Options ... *panel to display sharper contour lines). Export this plot to your* Word *report and comment the results.*

This solution will be used as the initial condition of the natural convection problem considered next.

4.2.2 Solving the natural convection flow with default convergence settings

In this part, you will perform the natural flow computation (with $g = -1$) with the basic procedure to monitor the convergence of the numerical solution, which is to check that the residuals of the discretized equations fall below a specific tolerance ϵ. By default, the tolerance thresholds of Fluent are :

- $\epsilon_u = 10^{-3}$ for flow equations,
- $\epsilon_T = 10^{-6}$ for thermal equations.

It is recommended that the thermal tolerance ϵ_T be 3 orders of magnitude below the fluid flow tolerance ϵ_u, for reasons that will be explained next.

One of the objectives of this simulation is to calculate the Surface Nusselt Number $Nu\,S$ at the surface S of the half-cylinder and compare with its benchmark value (1.3). For this purpose, set up the calculation of $Nu\,S$ on the Report Definitions task page. Appendix A.3 gives information on the ANSYS Fluent calculation of $Nu\,S$ and the other heat transfer quantities used in this lab exercise.

> **Question 5.** *Compute the flow with the default tolerance thresholds. In your* Word *report, report (and plot) the monitoring history of the residuals and $Nu\,S$ (Tip : Use* Reporting Interval=10 *in the* Run Calculation *task page to avoid printing the residuals at each iteration). When the solution is converged, report the value of $Nu\,S$ and its normalized error with respect to its benchmark value.*

4.2.3 Checking overall flux conservation

The basic procedure for assessing convergence may not be sufficient to obtain a physically realistic computations. It is thus recommended to lower the threshold ϵ_u and ϵ_T, but it is also important to check the convergence of relevant physical quantities such as forces or heat fluxes at solid walls.

For example, the equation of energy (A.7) states that the total heat flux rate (1.1) is conserved at the external boundaries eb of the domain when the steady state is reached :

$$\sum_{eb} Q = 0. \tag{4.1}$$

A physically realistic computation should verify this balance of fluxes.

> **Question 6.** *For the simulation with $\epsilon_u = 10^{-3}$ and $\epsilon_T = 10^{-6}$ computed in the previous question, report the (normalized) value of the heat flux imbalance* (4.1)*. Give your conclusions on the convergence of this simulation.*

4.2.4 Solving the natural convection flow with improved convergence settings

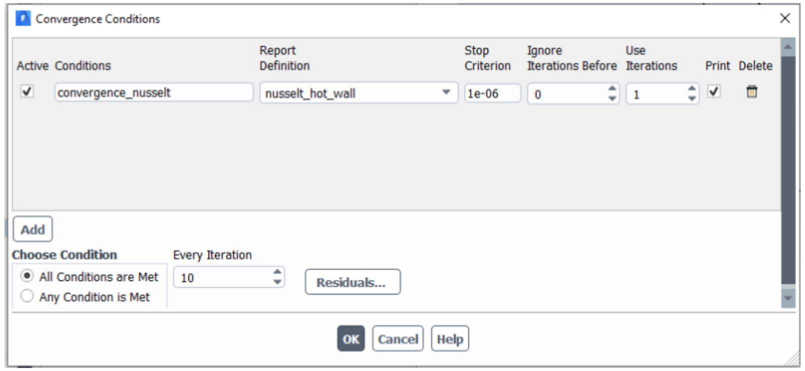

Figure 3: Convergence Conditions panel for $Nu\,S$. This panel defines the condition $|Nu^k - Nu^{k-p}|S < \epsilon$ where $p =$ Use Iterations and $\epsilon =$ Stop Criterion.

In addition to the residual history, Fluent allows to monitor the convergence of global physical quantities during the iterative solution procedure. Of particular importance in this lab exercise is the monitoring of the Surface Nusselt Number $Nu\,S$ set up previously. Use the Convergence Conditions panel (see Fig. 3) to activate this monitoring.

> **Question 7.** *Perform a simulation with $\epsilon_u = 10^{-6}$, $\epsilon_T = 10^{-9}$ and $\epsilon_{Nu} = 10^{-6}$ for the surface Nusselt number. In your* Word *report, report (and plot) the same quantities as the previous two questions. Give your conclusions.*

4.3 Step 3 : Post-processing

This part is devoted to the analysis of the results obtained for the natural convection problem.

> **Question 8.** *Display the contour plot of the temperature in the domain. Export this graph to your* Word *report. Compare with the plot obtained in Question 4 for conduction only.*

> **Question 9.** *Create a* Custom Field Function *for the Boussinesq density $\rho(T)$ given by* (A.3)*. Display the contour plot of the density in your* Word *report. Comment your results.*

> **Question 10.** *Display the velocity vectors in the domain (Tip : for better vizualisation of the flow circulation, color the vectors by temperature, and adjust the values of the* Scale *and* Skip *options). Export this graph to your* Word *report. Use the plots obtained in Questions 8-10 to explain the phenomenon of natural convection in a closed domain.*

The final question is devoted to the plotting of the local Surface Nusselt number along the surface S of the hot cylinder. Demirdžić has given in [5] benchmark values for this quantity at selected locations on S. The plot of these benchmark values is performed in the Matlab file `plot_local_surf_nusselt.m` available in the course website[4].

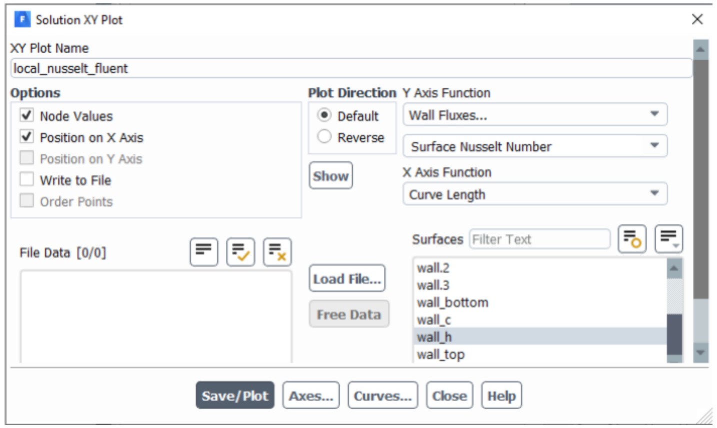

Figure 4: Panel for plotting the local Nusselt number on a selected surface. Solution X Axis Function : Curve length corresponds to the the curvilinear abscissa of the surface[5]. The option Write to File allows you to write the results in a text file ready to be uploaded in Matlab .

[4]Course Website on ARCHE : `https://arche.univ-lorraine.fr/course/view.php?id=9396`.

> **Question 11.** *Use the* Solution XY Plot *panel (see Fig. 4) to plot the local Nusselt number along the cylinder surface. Export your data in* Matlab *and compare your results with the benchmark values in the same* Matlab *plot.*

A Appendix : Basics on convective heat transfer

This part briefly addresses the basic notions of heat transfer in fluids that are useful for performing this lab exercise. More information is available in standard textbooks, *e.g.* [3, 8, 2]. In addition, the last section gives the ANSYS Fluent definition of heat transfer quantities.

A.1 Basic regimes of heat transfer in fluids

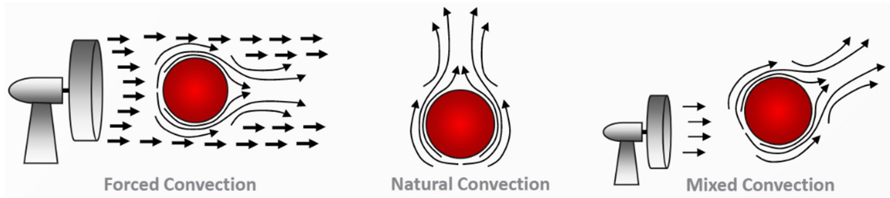

Figure 5: Basic regimes of convective heat transfer from a hot cylinder.

Convective heat transfer refers to the transport of energy in fluid flows, which is coupled to the transfer of momentum (the Navier-Stokes equations). The 3 main types of heat transfer in fluid are depicted in Fig. 5. In *forced convection*, fluid motion is driven an external force (pumps, fans, . . .), enabling energy transfer due to temperature differences. In *natural (or free) convection*, fluid motion is caused by buoyancy forces due to the differences between hot and cold zones in the fluid. Finally, *mixed convection* refers to the regime where natural and forced convection are both present, with similar intensities.

A.2 Natural convection

As shown by the equation of state for a perfect gas :

$$p = \rho r T, \tag{A.1}$$

at constant pressure the gas density decreases when the temperature increases, allowing the gas to expand. This effect plays a central role in natural convection, where fluid motion is generated by the spatial variations of density (*i.e.* , stratification) in conjunction with the gravitational field : the so-called buoyancy forces. In the configurations of Fig. 6, the fluid heated from below expands and rises up spontaneously due to the buoyancy forces generated by the temperature differences. In the closed domain of Fig. 6 (right), the fluid that reaches the top cools down and then sinks, enabling flow circulation and mixing in the domain.

[5]Issues with using Curve length may be related to the fact that Fluent is running in parallel mode. To enter the serial mode, close the Fluent application. Then, deselect the option Run Parallel Version in the properties of the Setup cell of your WorkBench project.

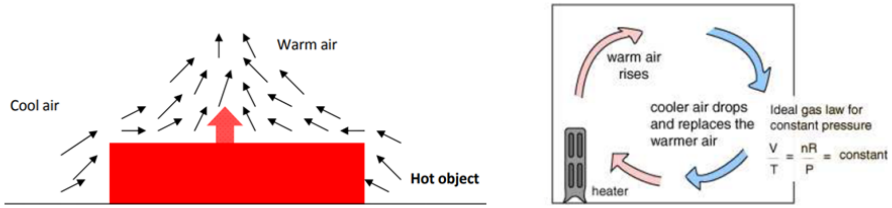

Figure 6: *At left* : Natural convection from a hot body in an open domain. *At right* : Heating in a closed room through natural convection.

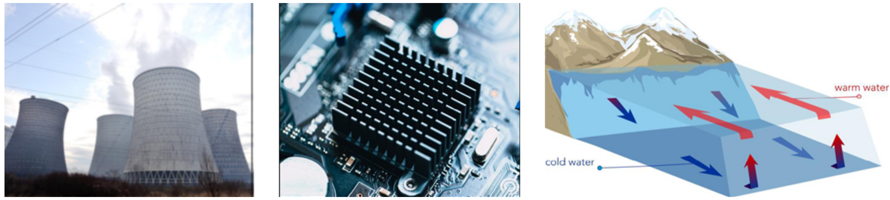

Figure 7: Examples of natural convection flows in industrial and natural processes.

In the absence of flow motion, heat transfer would occur by conduction only at a much lower rate. Hence, natural convection plays an important role in industrial applications such as power generation, food processing, HVAC[6] or electronic cooling (Fig. 7 and Ref. [1]). It is also the driving mechanism of many natural processes such as oceanic and atmospheric currents.

A.2.1 Boussinesq approximation of the Navier-Stokes equations

The Boussinesq approximation (1903) is the most usual approach for modelling natural convection flows. This approximation is valid when the temperature variations are "sufficiently" small around an average temperature T_0 to induce small density fluctuations around $\rho_0 = \rho(T_0)$.

The Boussinesq approximation assumes that all thermophysics coefficients of the Navier-Stokes and energy equations are constant, except in the body force term $\rho \boldsymbol{g}$ of the momentum equation which accounts for gravity. In order to express the buoyancy force of natural convection, the density is assumed to depend on temperature only : $\rho = \rho(T)$. For constant pressure, the Taylor expansion of $\rho(T)$ at $T = T_0$ is :

$$\rho(T) = \rho(T_0) + \left.\frac{\partial \rho}{\partial T}\right|_p (T - T_0) + \mathcal{O}(T - T_0)^2, \tag{A.2}$$

which yields the following linear approximation of the density :

$$\rho(T) = \rho_0 - \beta \rho_0 (T - T_0), \tag{A.3}$$

where the thermal expansion coefficient is :

$$\beta = -\frac{1}{\rho_0} \left.\frac{\partial \rho}{\partial T}\right|_p . \tag{A.4}$$

[6]HVAC: Heating, ventilation and air-conditioning.

In practical applications, the Boussinesq approximation is considered valid when the temperature variations δT do not cause large density changes, *i.e.* according to (A.3) :

$$\beta \delta T \ll 1. \tag{A.5}$$

Inserting the Boussinesq approximation (A.3) into the momentum equation gives :

$$\rho_0 \Big(\frac{\partial \boldsymbol{v}}{\partial t} + \nabla \cdot (\boldsymbol{v} \otimes \boldsymbol{v}) \Big) + \nabla \big(p + \rho_0 \boldsymbol{g} \cdot \boldsymbol{x}\big) - \mu \nabla^2 \boldsymbol{v} = -\beta \rho_0 (T - T_0) \boldsymbol{g}, \tag{A.6}$$

where $\boldsymbol{x} = (x, y, z)$. The energy equation is simply :

$$\rho_0 c_{\mathrm{p}} \Big(\frac{\partial T}{\partial t} + \nabla \cdot (\boldsymbol{v} T) \Big) - \kappa \nabla^2 T = 0. \tag{A.7}$$

The RHS in the momentum equation (A.6) is responsible of the buoyancy effects and depends on the temperature T, which strongly couples the Navier-Stokes equations with the energy equation. The average values T_0 and ρ_0 in (A.6) are problem-dependent and have to be set by the user. Their "exact" values does not impact the solution to the Boussinesq equations, since $\rho_0 \boldsymbol{g} \cdot \boldsymbol{x}$ in the LHS of (A.6) accounts for the hydrostatic pressure. However, incorrect values of T_0 and ρ_0 may lead to round-off errors and impact the convergence of iterative solvers. For closed domains, it is advised to set T_0 and ρ_0 as their averaged values in the domain (using information at the boundaries, for example). For open domains, it is advised to use the external ambient density and temperature.

A.2.2 Heat transfer coefficient and dimensionless numbers for natural convection

For calculating the heat flux rate Q/S across a surface S between the fluid and the solid, one of the key step is to evaluate the heat transfer coefficient h (in $\mathrm{W} \cdot \mathrm{m}^{-2} \cdot \mathrm{K}^{-1}$) which is defined by Newton's law of cooling $Q/S = h \Delta T$. Analytic calculations of h can be done for canonical configurations only. For complex configurations, h needs to be numerically computed (as in this lab exercise), or estimated through empirical correlations that involves dimensionless numbers representative of the flow under consideration, such as the Nusselt number $Nu = hL/\kappa$:

$$Nu = F(Pr,\, Gr), \quad \text{or simply} \quad Nu = F(Ra).$$

In the context of this lab exercise on ANSYS Fluent, the precise definition of the Nusselt number Nu, the Prandtl number Pr, the Grashof number Gr and the Rayleigh number $Ra = Pr\, Gr$ are given next.

Prandtl number

The Prandtl number Pr of a fluid is the ratio of its momentum diffusivity to its thermal diffusivity :

$$\begin{aligned} Pr &= \frac{\text{Momentum diffusivity}}{\text{Thermal diffusivity}} = \frac{\mu/\rho}{\kappa/(c_{\mathrm{p}}\rho)} \\ &= \frac{c_{\mathrm{p}}\mu}{\kappa}. \end{aligned} \tag{A.8}$$

Since Pr does not depend on scale variables such as L_{ref} (or U_{ref}), the Prandtl number is representative of the material properties of the fluid only. For large values of Pr, momentum diffusion dominates and heat is exchanged at a lower rate. In consequence, the thermal boundary layer of laminar flows on a flat plate is thiner than the hydrodynamic boundary layer (see Fig. 8).

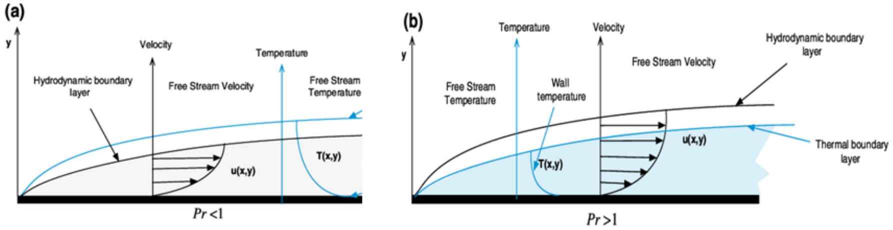

Figure 8: Thermal and momentum boundary layers for (a) $Pr < 1$ and (b) $Pr > 1$ (taken from [4].

Grashof and Rayleigh numbers

The Grashof number Gr is defined as the ratio of the buoyancy forces (which drive the flow) to viscous forces (which resist flow motion). With the notations of this lab exercise, it is given by:

$$Gr = \frac{\text{Buoyancy forces}}{\text{Viscous forces}} = \frac{g L_{\text{ref}}^3 \beta \, (T_{\text{H}} - T_{\text{C}}) \, \rho_0^2}{\mu^2}. \tag{A.9}$$

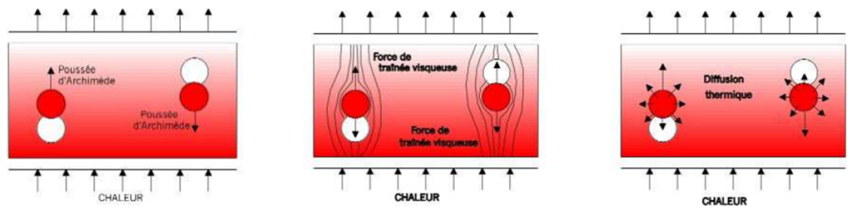

Figure 9: The Rayleigh number (A.10) expresses the competition between driving forces (buoyancy, at left) and resistive forces (viscous resistance, middle ; and thermal diffusion that increases the heat exchanged by the displaced fluid parcel, at right.

Many empirical correlations for the Nusselt number involves the Rayleigh number Ra given by :

$$\begin{aligned} Ra &= \frac{\text{Buoyancy forces}}{\text{Viscous forces}} \frac{\text{Momentum diffusivity}}{\text{Thermal diffusivity}} \\ &= Gr \; Pr = \frac{g L_{\text{ref}}^3 \beta \, (T_{\text{H}} - T_{\text{C}}) \, \rho_0}{\mu \kappa}, \end{aligned} \tag{A.10}$$

which expresses the ratio of the driving buoyancy forces to the stabilizing forces of viscosity and thermal diffusivity (see Fig. 9).

In natural convection the Reynolds number no longer characterize the flow, since a reference value U_{ref} cannot be defined. Instead, the Grashof or Rayleigh numbers are used. For example, in the configuration of Fig. 10, the fluid is at rest for low values of Ra and heat transfer occurs by conduction only. Above a critical value of $Ra_{\text{c}} \approx 1700$, gravity forces become more prominent and induce fluid circulation, forming the so-called Bénard convection cells [7]. At higher values of Ra, the flow becomes turbulent. In engineering applications, the flow is considered turbulent when $Ra \geq 10^9$ [3, 2, 1].

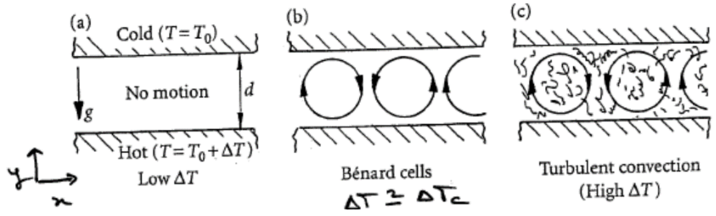

Figure 10: Example of Rayleigh-Bénard convection between 2 horizontal plates with $L_x \gg L_y$. The fluid is heated from below with a temperature difference of ΔT between the 2 plates. The various flow regimes are sketched for increasing value of the Rayleigh number $Ra = f(\Delta T)$.

Nusselt number and heat transfer coefficient

The Nusselt number Nu is a dimensionless parameter that measures the amplification of heat transfer across a surface S due to the convective fluid motion. It is expressed as the ratio of the heat flux in the presence of convection to the heat flux given by conduction only [6, 3] :

$$Nu = \frac{Q_{\text{convection}}}{Q_{\text{conduction}}}. \tag{A.11}$$

In the case of the configuration of Fig. 1, $Q_{\text{convection}}$ is given by (1.1). The conduction heat flux can be approximated as $Q_{\text{conduction}} \simeq \kappa \nabla T_{\text{conduction}} S$, where $\nabla T_{\text{conduction}}$ represents the temperature difference between the hot cylinder and the cold left wall :

$$\nabla T_{\text{conduction}} \simeq \frac{T_{\text{H}} - T_{\text{C}}}{L_{\text{x}}}, \tag{A.12}$$

resulting in the following expression of the Nusselt number (A.11) :

$$Nu = \frac{Q}{\kappa \frac{T_{\text{H}} - T_{\text{C}}}{L_{\text{x}}} S}. \tag{A.13}$$

The Nusselt number is the dimensionless form of the heat transfer coefficient h given by Newton's law of cooling :

$$\frac{Q}{S} = h(T_{\text{H}} - T_{\text{C}}). \tag{A.14}$$

By combining this expression with (A.13), one retrieves the well-known relation :

$$Nu = \frac{h L_x}{\kappa}. \tag{A.15}$$

For practical flow configurations, the expression of the Nusselt number and heat transfer coefficient used by ANSYS Fluent will be given next.

A.3 Postprocessing heat transfer with ANSYS Fluent

The calculation of heat transfer quantities is available in the Results → Reports task page (see Fig. 11). The following definitions refer to calculations performed *on a given surface S of temperature T*. For calculating quantities such as the heat transfer coefficient or the Nusselt number, the value of L_{ref} and T_{ref} needs to be set in the Reference Values task page.

Figure 11: Reports panel from the Results menu.

Panel Reports → Fluxes...

The Total Heat Transfer Rate (in W) corresponds to the heat flux Q given by (1.1).

Panel Reports → Surface Integrals...

- The report type Area gives the value of S in m^2.
- The report type Integral corresponds to quantities integrated on surface S. For the field variable Wall Fluxes..., the following quantities are available :
  - Total Surface Heat Flux (in W): It is the same as the Total Heat Transfer Rate Q defined above (up to a minus sign).
  - Surface Heat Transfer Coef. (in $\mathrm{W \cdot K^{-1}}$): In analogy with (A.14), it corresponds to the quantity hS calculated by ANSYS Fluent as :

$$hS = \frac{Q}{T - T_{\mathrm{ref}}}. \tag{A.16}$$

  - Surface Nusselt Number (in m^2): In analogy with (A.13), it corresponds to the quantity $Nu\,S$ calculated by ANSYS Fluent as :

$$Nu_{,}S = \frac{Q}{\kappa \dfrac{T - T_{\mathrm{ref}}}{L_{\mathrm{ref}}}}. \tag{A.17}$$

- The report type Area-Weighted Average gives the above quantities divided by the surface area S.

References

[1] Natural Convection - ANSYS Innovation Courses. `https://courses.ansys.com/index.php/courses/natural-convection/`.

[2] R. Bird, W. Stewart, and E. Lightfoot. *Transport Phenomena*. Wiley, 2007.

[3] P. Boulet and D. Lacroix. Notes de cours M1 Energie - UE Transferts Thermiques. Université de Lorraine.

[4] M. Darwish, L. Mangani, and F. Moukalled. *The Finite volume method in computational fluid dynamics: an advanced introduction with OpenFOAM® and Matlab®*. Springer, 2016.

[5] I. Demirdžić, Ž. Lilek, and M. Perić. Fluid flow and heat transfer test problems for non-orthogonal grids: Bench-mark solutions. *International Journal for Numerical Methods in Fluids*, 15(3):329–354, 1992.

[6] M. Griebel, T. Dornseifer, and T. Neunhoeffer. *Numerical Simulation in Fluid Dynamics: A Practical Introduction*. SIAM, 1998.

[7] E. Guyon, J.-P. Hulin, L. Petit, and C. D. Mitescu. *Physical Hydrodynamics*. Oxford University Press, 2015.

[8] Y. Jannot, C. Moyne, and A. Degiovanni. *Transfert de Chaleur 1 : Conduction et Convection*. ISTE Editions, 2022.