Homework 12: Natural Convection

ME 590: Applied CFD and Numerical Heat Transfer
Graham Wilson, AER

Problem Statement

Conduct a CFD analysis for a horizontal rectangular cavity in two configurations: heated on top and heated on bottom, using the same geometry, fluid properties, and wall temperatures as the validation case used in Example 12.1. Compare the heat flux in your models for each case to the values that would be predicted by the correlations in your heat transfer textbook.

Prepare a brief report that includes the following for each case:

- A comparison of the model heat flux to the heat flux predicted by the appropriate correlation
- A description of your mesh
- A plot of the temperature contours
- A plot of the velocity magnitude contours
- A plot of the velocity vectors
- A description of the flow field

Use the viscous-laminar model for each of the cases.

Solution

Constants and Given Values

$$g = 9.81\,\mathrm{m/s^2} \tag{1}$$
$$T_{\mathrm{hot}} = 325\,\mathrm{K} \quad \text{(Hot Wall Temperature)} \tag{2}$$
$$T_{\mathrm{cold}} = 275\,\mathrm{K} \quad \text{(Cold Wall Temperature)} \tag{3}$$
$$L = 0.05\,\mathrm{m} \quad \text{(Distance Between Plates)} \tag{4}$$
$$H = 0.5\,\mathrm{m} \quad \text{(Length of the Plates)} \tag{5}$$

Properties at Film Temperature

Film Temperature:

$$T_f = \frac{T_{\mathrm{hot}} + T_{\mathrm{cold}}}{2} = \frac{325 + 275}{2} = 300\,\mathrm{K} \tag{6}$$

Table 1: Relevant Fluid Properties at $T_f = 300\,\mathrm{K}$

Property	**Symbol**	**Value**
Density	ρ	$1.1614\,\mathrm{kg/m^3}$
Specific Heat	c_p	$1007\,\mathrm{J\,kg^{-1}\,K^{-1}}$
Thermal Expansion Coefficient	β	$\dfrac{1}{T_f} = 3.333 \times 10^{-3}\,\mathrm{K^{-1}}$
Dynamic Viscosity	μ	$1.846 \times 10^{-5}\,\mathrm{kg\,m^{-1}\,s^{-1}}$
Kinematic Viscosity	ν	$1.589 \times 10^{-5}\,\mathrm{m^2/s}$
Thermal Conductivity	k	$0.0263\,\mathrm{W\,m^{-1}\,K^{-1}}$
Thermal Diffusivity	α	$2.25 \times 10^{-5}\,\mathrm{m^2/s}$
Prandtl Number	Pr	0.707

Calculation of Rayleigh Number

The Rayleigh number is calculated using:

$$Ra_L = \frac{g\beta(T_{\text{hot}} - T_{\text{cold}})L^3}{\nu\alpha} \tag{7}$$

Substituting the values:

$$\begin{aligned}
Ra_L &= \frac{(9.81\,\mathrm{m/s^2})(3.333 \times 10^{-3}\,\mathrm{K^{-1}})(50\,\mathrm{K})(0.05\,\mathrm{m})^3}{(1.589 \times 10^{-5}\,\mathrm{m^2/s})(2.25 \times 10^{-5}\,\mathrm{m^2/s})} \\
&= \frac{(9.81)(3.333 \times 10^{-3})(50)(1.25 \times 10^{-4})}{(1.589 \times 10^{-5})(2.25 \times 10^{-5})} \\
&= \frac{0.000204375}{3.57525 \times 10^{-10}} \\
&= 571,440 \\
&\approx 5.714 \times 10^5
\end{aligned} \tag{8}$$

Case 1: Heated from Below ($\tau = 0^\circ$)

Calculation of Nusselt Number

For natural convection between horizontal plates heated from below, the correlation is:

$$\overline{Nu}_L = 0.069\, Ra_L^{1/3} Pr^{0.074}, \quad \text{for } 3 \times 10^5 < Ra_L < 7 \times 10^9 \tag{9}$$

Calculating $Ra_L^{1/3}$:

$$Ra_L^{1/3} = (5.714 \times 10^5)^{1/3} \approx 83.026 \tag{10}$$

Calculating $Pr^{0.074}$:

$$Pr^{0.074} = (0.707)^{0.074} \approx 0.9746 \tag{11}$$

Substituting back into the Nusselt number correlation:

$$\begin{aligned}
\overline{Nu}_L &= 0.069 \times 83.026 \times 0.9746 \\
&= 5.7288 \times 0.9746 \\
&\approx 5.583
\end{aligned} \tag{12}$$

Calculation of Heat Transfer Coefficient

$$
\begin{aligned}
\overline{h} = \frac{\overline{Nu}_L\, k}{L} &= \frac{5.583 \times 0.0263\,\mathrm{W\,m^{-1}\,K^{-1}}}{0.05\,\mathrm{m}} \\
&= \frac{0.1468\,\mathrm{W/m^2/K}}{0.05\,\mathrm{m}} \\
&= 2.936\,\mathrm{W/m^2/K} \qquad (13)
\end{aligned}
$$

Calculation of Heat Flux

The rate of convective heat transfer per unit length q' is:

$$
\begin{aligned}
q'_{\tau=0^\circ,\mathrm{th}} &= \overline{h} \times H \times (T_{\mathrm{hot}} - T_{\mathrm{cold}}) \\
&= 2.936\,\mathrm{W/m^2/K} \times 0.5\,\mathrm{m} \times 50\,\mathrm{K} \\
&= 2.936 \times 25 \\
&= 73.4\,\mathrm{W\,m^{-1}} \qquad (14)
\end{aligned}
$$

Comparison with CFD Model Heat Flux

From the CFD model, the average heat flux per unit length is $q'_{\tau=0^\circ,\mathrm{CFD}} = 91.527\,\mathrm{W\,m^{-1}}$ (obtained from the heat flux report in Figure 4).

The percent difference between the theoretical and CFD results is:

$$
\begin{aligned}
\text{Percent Difference} &= \left| \frac{q'_{\mathrm{CFD}} - q'_{\mathrm{th}}}{q'_{\mathrm{th}}} \right| \times 100\% \\
&= \left| \frac{91.527 - 73.4}{73.4} \right| \times 100\% \\
&= \left| \frac{18.127}{73.4} \right| \times 100\% \\
&\approx 24.7\% \qquad (15)
\end{aligned}
$$

In Case 1, the CFD-predicted heat flux is approximately **24.7%** higher than the theoretical value calculated using the empirical correlation for the Nusselt number. This discrepancy is relatively significant and can be attributed to several factors. The empirical correlation used for the Nusselt number is idealized and an approximation that may not fully capture the complex flow dynamics within the cavity, especially at Rayleigh numbers near the lower limit of the correlation's validity range ($3 \times 10^5 < Ra_L < 7 \times 10^9$). Also, natural convection flows are inherently unstable and can develop complex circulation patterns and boundary layer developments that complicate the heat transfer equations beyond what is predicted by the simplified correlations.

Description of the Mesh

The mesh used for the simulation consists of:

- Structured Cartesian grid with uniform spacing
- 2000 x-axis edge divisions and 200 y-axis edge divisions
- Total elements: $2000 \times 200 = 400,000$ cells
- Fine resolution to capture boundary layers and flow details

Temperature Contours

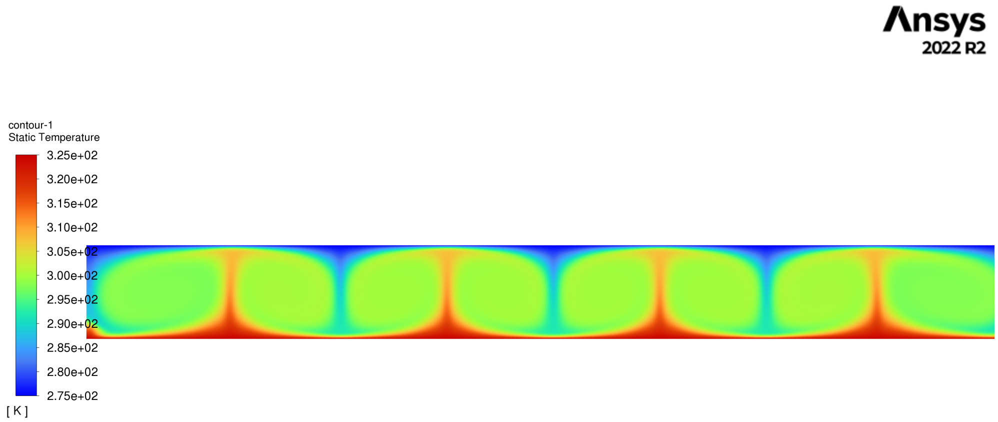

Figure 1: Temperature contours for Case 1: Heated from Below ($\tau = 0°$).

Velocity Magnitude Contours

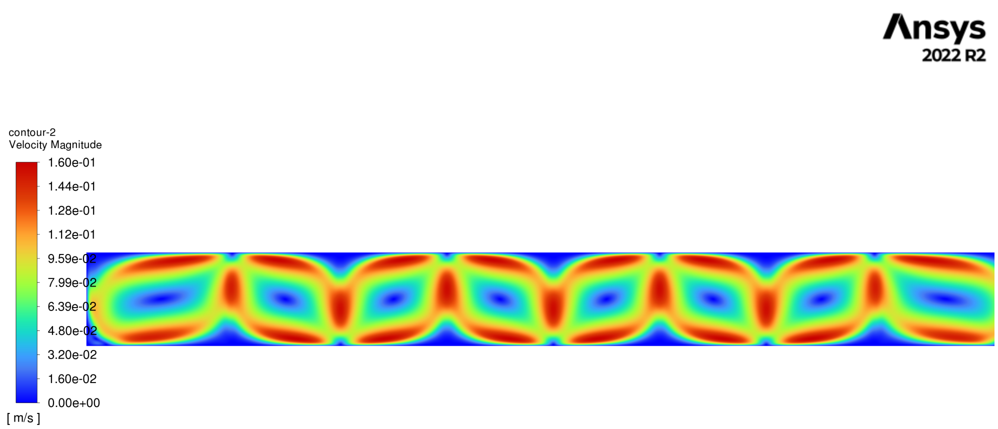

Figure 2: Velocity magnitude contours for Case 1: Heated from Below ($\tau = 0°$).

Velocity Vectors

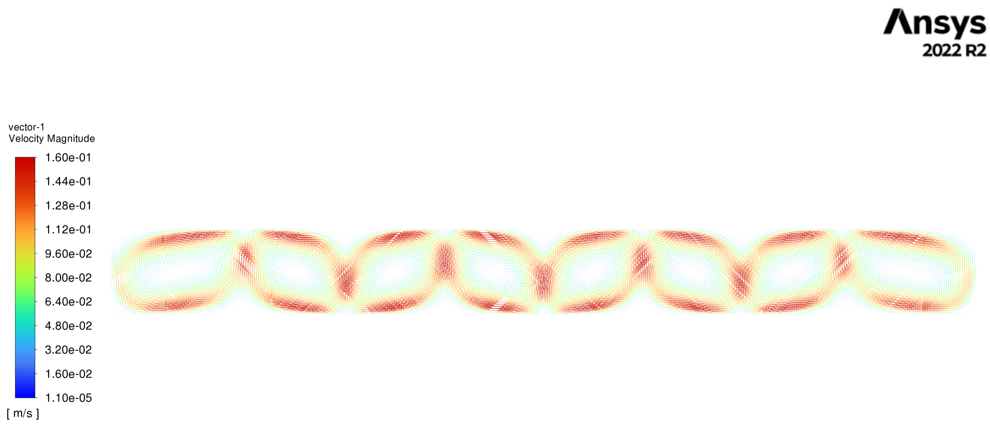

Figure 3: Velocity vectors for Case 1: Heated from Below ($\tau = 0°$).

Heat Flux Report

```
        Total Heat Transfer Rate                  [W]
-------------------------------- --------------------
                            xmax                   -0
                            xmin                   -0
                            ymax           -91.527054
                            ymin            91.527054
                ---------------- --------------------
                             Net       -1.831437e-07
```

Figure 4: Heat fluxes for Case 1: Heated from Below ($\tau = 0°$).

Description of the Flow Field

In Case 1, where the cavity is heated from below, natural convection causes the warmer, less dense air near the bottom to rise, while cooler, denser air near the top descends. This creates convection currents within the cavity, forming circulating cells. The temperature contours show hot air rising from the bottom wall, and the velocity vectors illustrate the circulating flow pattern characteristic of natural convection.

Case 2: Heated from Above ($\tau = 180°$)

Calculation of Nusselt Number

For natural convection between horizontal plates heated from above, convection is suppressed, and the Nusselt number approaches 1 (pure conduction):

$$\overline{Nu}_L = 1 \tag{16}$$

Calculation of Heat Transfer Coefficient

$$\overline{h} = \frac{\overline{Nu}_L\, k}{L} = \frac{1 \times 0.0263\,\mathrm{W\,m^{-1}\,K^{-1}}}{0.05\,\mathrm{m}} = 0.526\,\mathrm{W/m^2/K} \tag{17}$$

Calculation of Heat Flux

$$\begin{aligned} q'_{\tau=180^\circ,\mathrm{th}} &= \overline{h} \times H \times (T_{\mathrm{hot}} - T_{\mathrm{cold}}) \\ &= 0.526\,\mathrm{W/m^2/K} \times 0.5\,\mathrm{m} \times 50\,\mathrm{K} \\ &= 0.526 \times 25 \\ &= 13.15\,\mathrm{W\,m^{-1}} \end{aligned} \tag{18}$$

Comparison with CFD Model Heat Flux

From the CFD model, the average heat flux per unit length is $q'_{\tau=180^\circ,\mathrm{CFD}} = 12.8\,\mathrm{W\,m^{-1}}$ (obtained from the heat flux report in Figure 8).

The percent difference between the theoretical and CFD results is:

$$\begin{aligned} \text{Percent Difference} &= \left| \frac{q'_{\mathrm{CFD}} - q'_{\mathrm{th}}}{q'_{\mathrm{th}}} \right| \times 100\% \\ &= \left| \frac{12.8 - 13.15}{13.15} \right| \times 100\% \\ &= \left| \frac{-0.35}{13.15} \right| \times 100\% \\ &\approx 2.66\% \end{aligned} \tag{19}$$

Case 2 shows a much closer agreement between the CFD and theoretical heat fluxes, with a discrepancy of only **2.66%**. This small error is acceptable and indicates that both the CFD model and theoretical approach are accurately capturing the physics of the problem. When heated from above ($\tau = 180^\circ$), the cavity experiences stable stratification, suppressing convective currents. Heat transfer occurs mainly through conduction, which is well-characterized by the Nusselt number approaching 1. Both the CFD model and the theoretical calculation assume pure conduction, leading to similar results.

Description of the Mesh

The mesh used for $\tau = 180^\circ$ is the same as for $\tau = 0^\circ$.

Temperature Contours

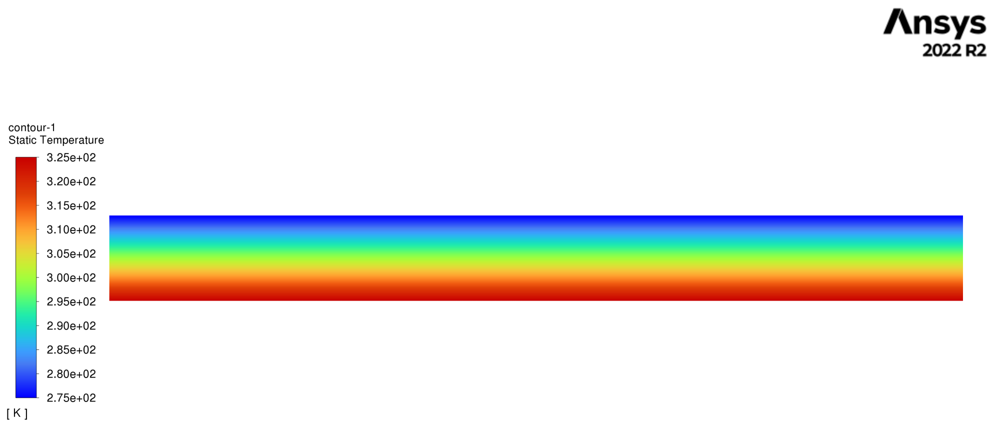

Figure 5: Temperature contours for Case 2: Heated from Above ($\tau = 180°$).

Velocity Magnitude Contours

Figure 6: Velocity magnitude contours for Case 2: Heated from Above ($\tau = 180°$).

Velocity Vectors

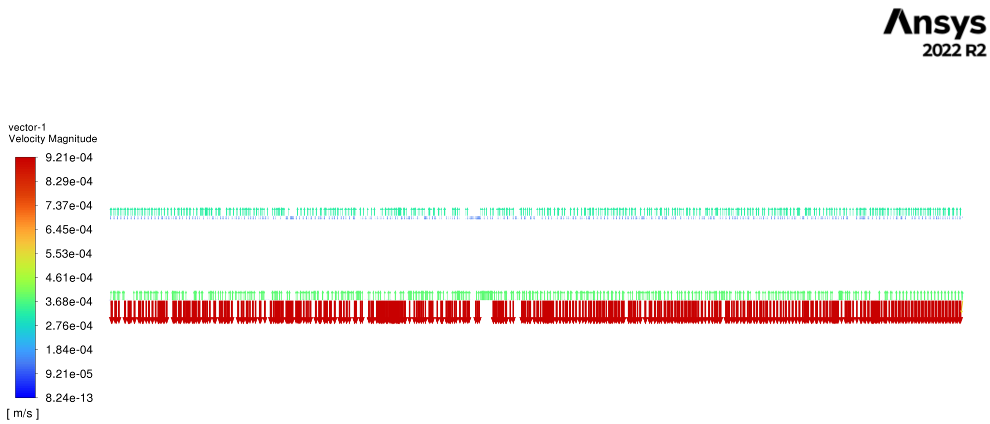

Figure 7: Velocity vectors for Case 2: Heated from Above ($\tau = 180°$).

Heat Flux Report

Total Heat Transfer Rate	[W]
xmax	-0
xmin	-0
ymax	-12.824848
ymin	12.825152
Net	0.00030418691

Figure 8: Heat fluxes for Case 2: Heated from Above ($\tau = 180°$).

Description of the Flow Field

In Case 2, where the cavity is heated from above, the system is stable, and natural convection currents are minimal. The warmer air at the top remains in place causing stable stratification, and heat transfer occurs primarily through conduction. The temperature contours are relatively horizontal, indicating negligible convective effects. The velocity magnitude contours and vectors show very low velocities throughout the cavity, confirming the absence of significant convective flow.