FCTUC FACULDADE DE CIÊNCIAS E TECNOLOGIA
UNIVERSIDADE DE COIMBRA

Energy Systems Modelling

Students:

Felipe Tassari Aveiro (2020218786)
Henrique Leal Honrado (2019236069)

Educators:

Nuno Cláudio Ferreira Rosa

Coimbra, March 2023

Index

1. Introduction

This report was written in the scope of the Energy Systems Modeling discipline aiming to simulate, in the Ansys software, some fluid flow situations, namely air, and analyze the results obtained. In this sense, three situations are studied: the airflow around a room, the airflow inside an open-loop earth-air heat exchanger tube (EAHE), and the airflow inside a room equipped with heat radiators in the context of both heating and cooling. Therefore, the procedures performed for the simulation of each of these situations will be described, and the results obtained will be demonstrated.

2. Airflow around a room

2.1 Objective

Create the simulation of airflow around a room and visualize the flow. In this regard, the turbulence option selected was Intensity and Length Scale, with a value of 0.05, corresponding to 5% turbulence, a medium intensity level, and a swirl length scale value of 0.1 m. Additionally, the flow around the body was modeled using Shear Stress Transport for turbulence and an incompressible flow. Finally, the adopted wind speed was 16 km/h (approximately 4.44 m/s).

2.2 Geometry

The geometry required for this analysis was created in the AutoCAD Autodesk Inventor Professional 2023 software – which will be used for all future sections – consisting of a parallelepiped representing the air, with a cavity on one of its faces representing the room. From this perspective, since both the airflow and the geometry are symmetrical relative to a vertical plane, only half of the geometry was modeled to obtain the solution, as the results obtained from the analysis of the symmetry plane are the same if the entire room were considered. Thus, the parallelepiped has dimensions of 130x50x50 [m] - corresponding to length, width, and height, respectively - and the cavity has dimensions of 10x5x10 [m]. Finally, in order to improve the approximation of the results obtained, a copy of the geometry was also modeled to cut it into sections to allow for mesh refinement around the room.

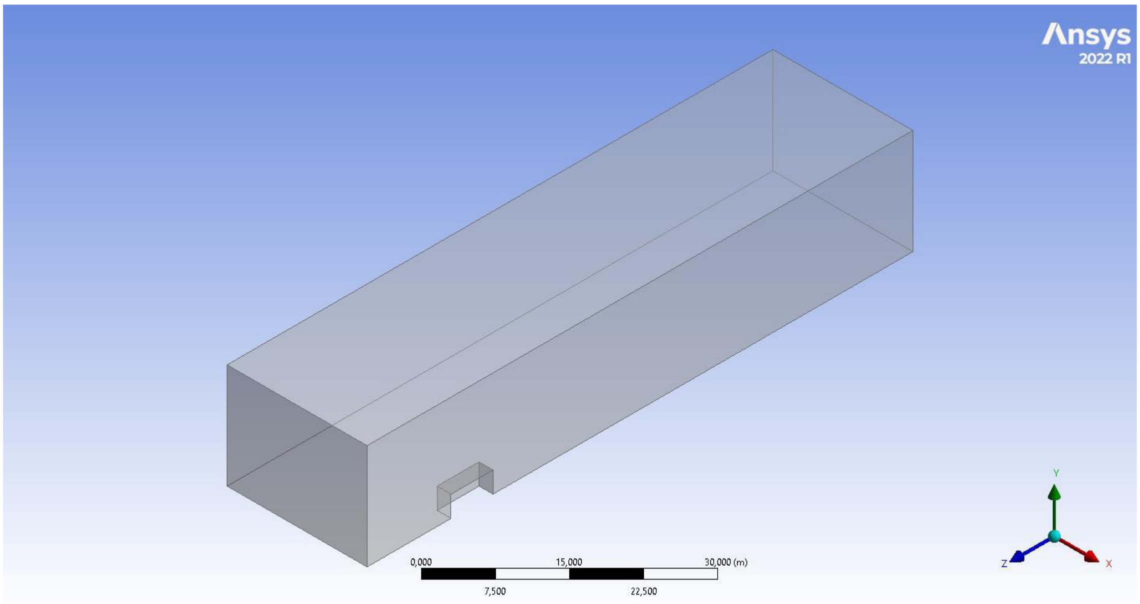

Figure 1: Uncut geometry of the airflow around a room

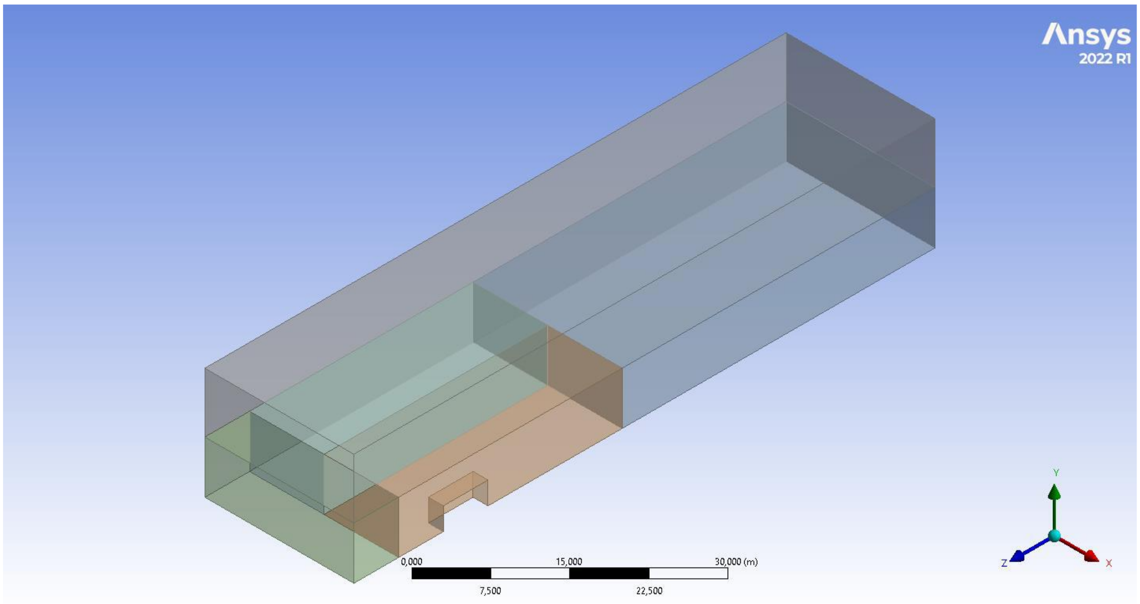

Figure 2: Cut geometry of the airflow around a room

2.3 Mesh

After modeling the geometries, a mesh was generated with an element size of 1 m for both cases. However, for mesh refinement and consequent improvement of result approximation, an Inflation option was applied in the region around the room. In this regard, the default configurations were adopted for this option at the time of implementation in the mesh.

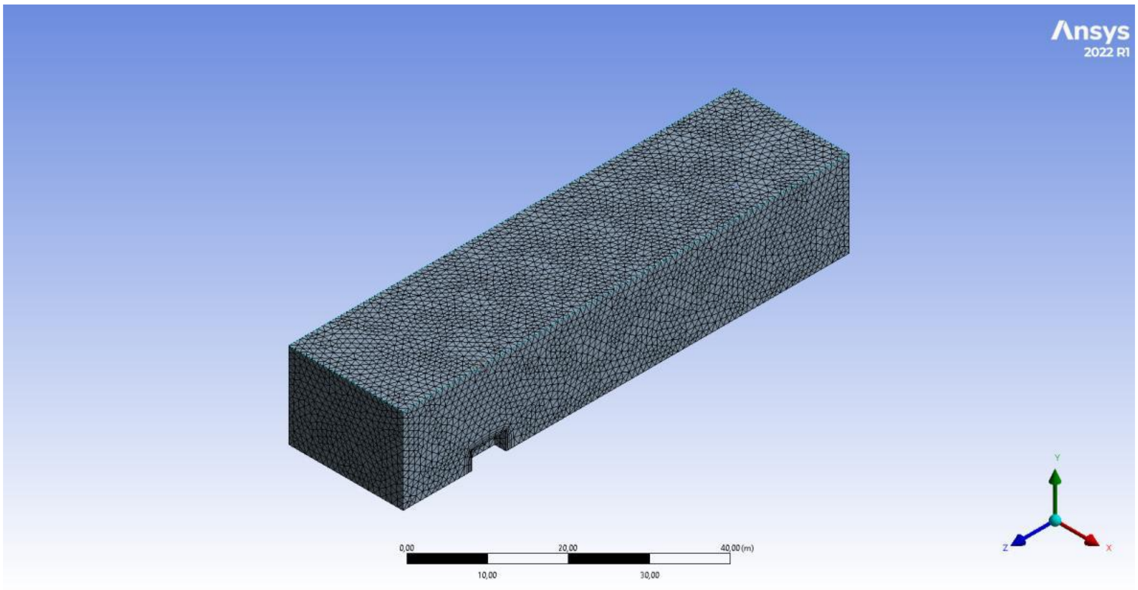

Figure 3: Unrefined mesh of the airflow around a room

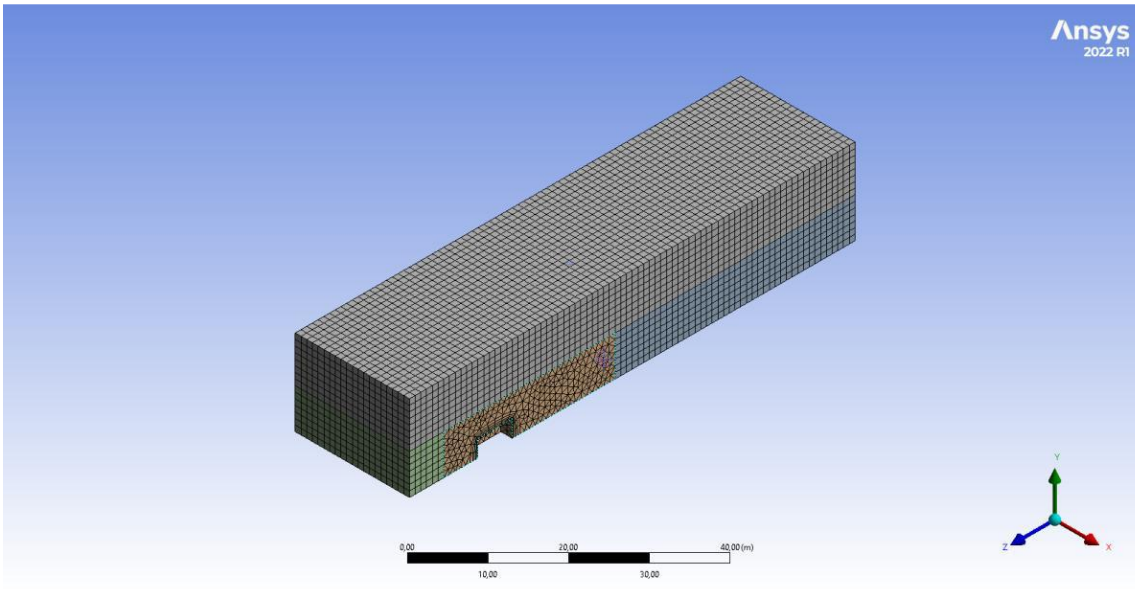

Figure 4: Unrefined mesh of the airflow around a room

2.4 Setup

In this step, the following options were defined:

- The faces of the air domain, except for the face with the cavity, the inlet face, and the outlet face, are set as Free Slip Walls, as there is no drag on the boundaries of the parallelepiped since the study focuses on the room in an open-air flow;

- The face with the cavity has the Symmetry boundary condition, which was chosen because the program defines symmetry for the conditions observed on both sides of the plane/face, i.e., inside and outside the fluid model created;
- The faces inside the cavity are set as No Slip Walls because there is drag on these faces;
- The air inlet face is set as an Inlet boundary condition with parameters for subsonic flow, a velocity of 16 km/h (approximately 4.44 m/s), and turbulence defined by the Intensity and Length Scale option. The turbulence has a value of 0.05, corresponding to 5% turbulence, with a medium intensity level and a swirl length scale value of 0.1 m;
- The air outlet face is set as an Outlet boundary condition with parameters for subsonic flow and a static pressure equal to the relative pressure of 0 Pa, as it represents atmospheric pressure;
- The domain around the cavity is a fluid domain consisting of a continuous fluid, in this case, air at 25°C, with a reference pressure of 1 atm. The turbulence model is Shear Stress Transport and buoyancy is disregarded;
- The simulation is performed in steady-state with an advection scheme set to High Resolution, a maximum number of iterations equal to 200, a convergence criterion defined by RMS equal to $1.0*10^{-5}$, and conservation target is active with a value of 0.01.

To monitor the air velocity, a point is defined at coordinates (15;2;45) [m] from the origin, which would correspond to the point (5;2;33) [m] if the origin were actually the vertex formed by the face of the cavity, the inlet face, and the bottom face of the parallelepiped, and the positive direction of the z-axis were inverted.

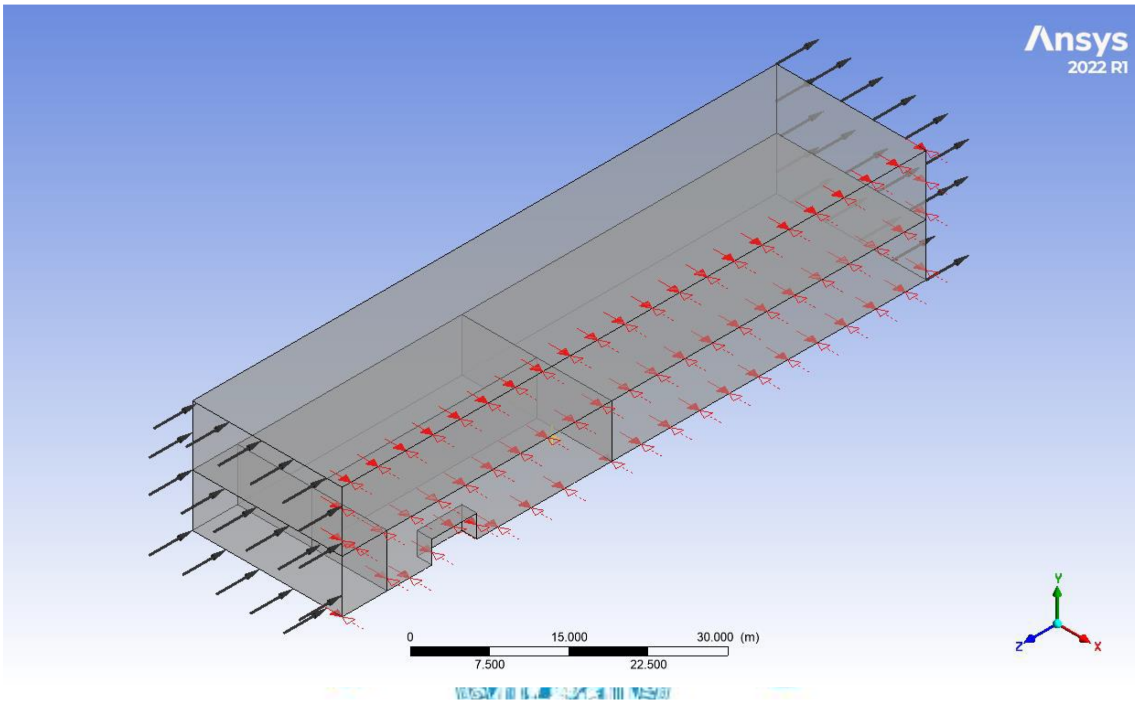

Figure 5: Setup of the unrefined iteration for the airflow around a room

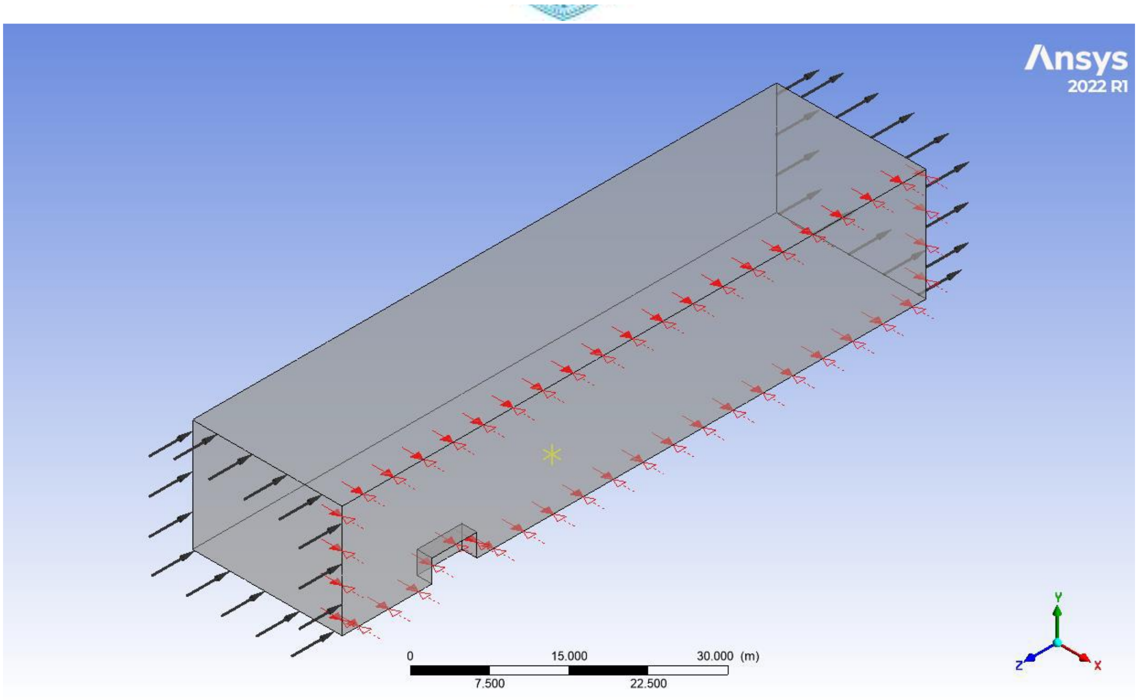

Figure 6: Setup of the refined iteration for the airflow around a room

2.5 Solution

After setting the program to compute using 4 logical processors and considering the initial conditions, it is observed that the solution converges.

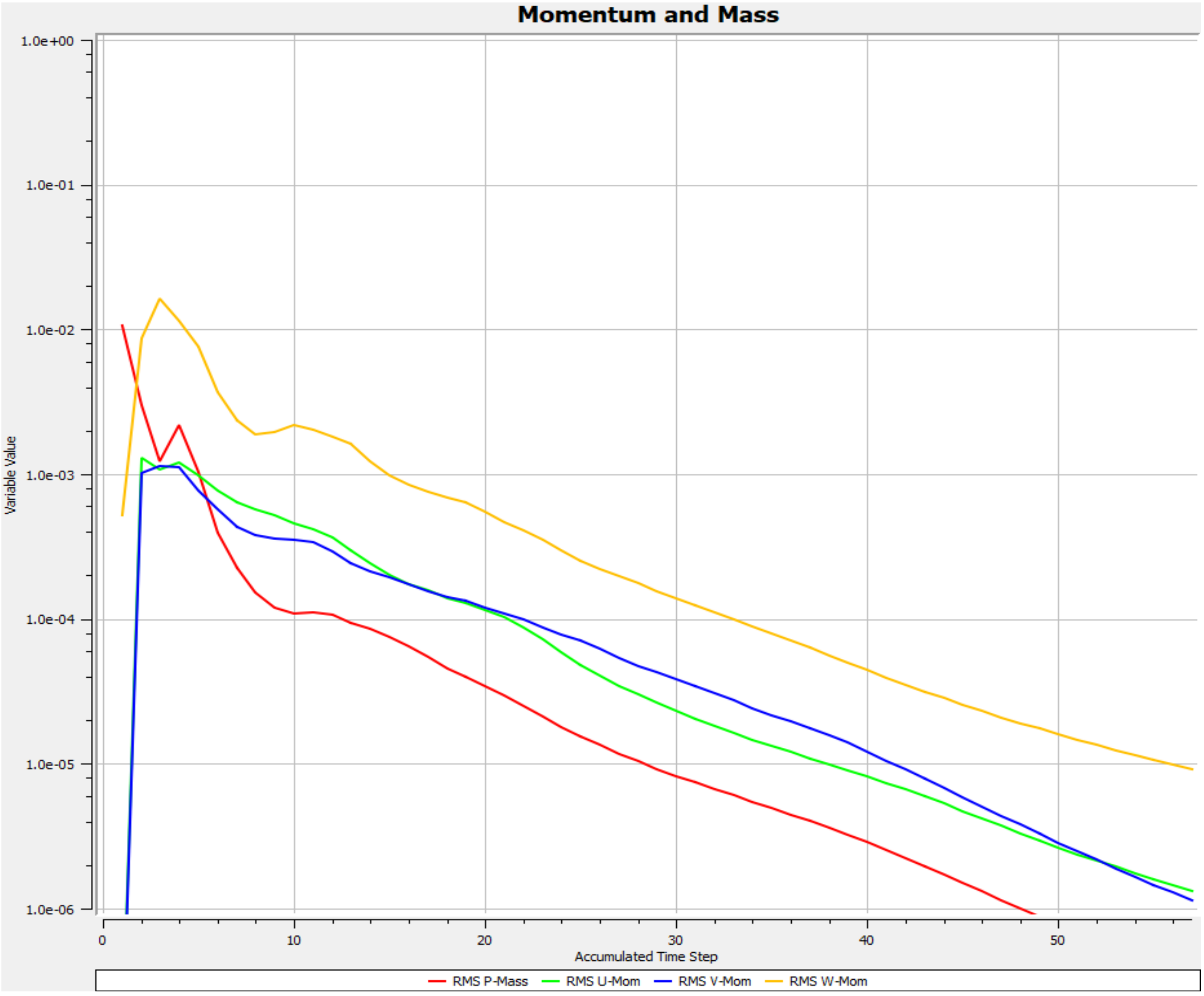

Figure 7: Convergence of momentum and mass for the unrefined iteration of airflow around a room

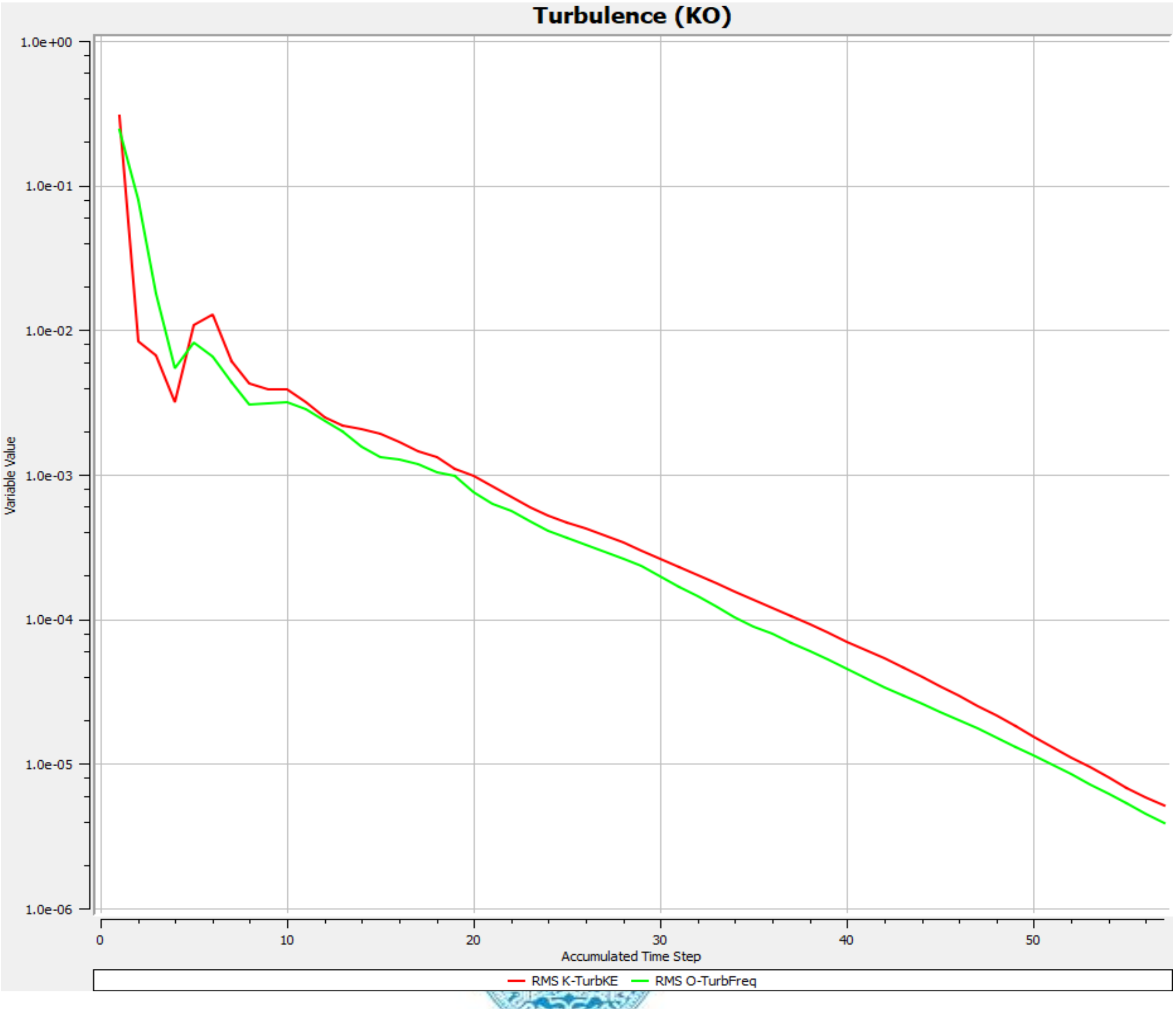

Figure 8: Convergence of turbulence for the unrefined iteration of airflow around a room

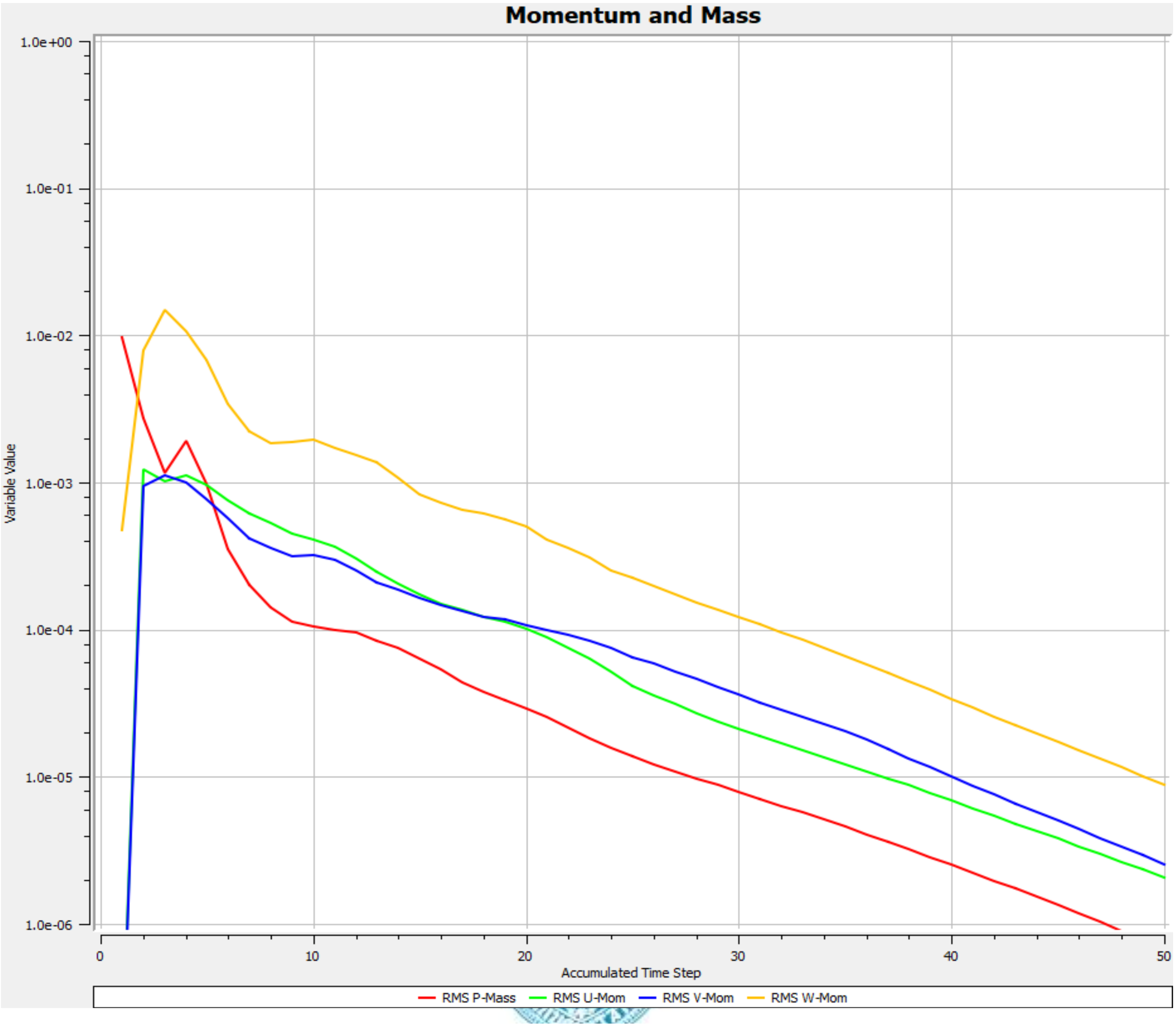

Figure 9: Convergence of momentum and mass for the refined iteration of airflow around a room

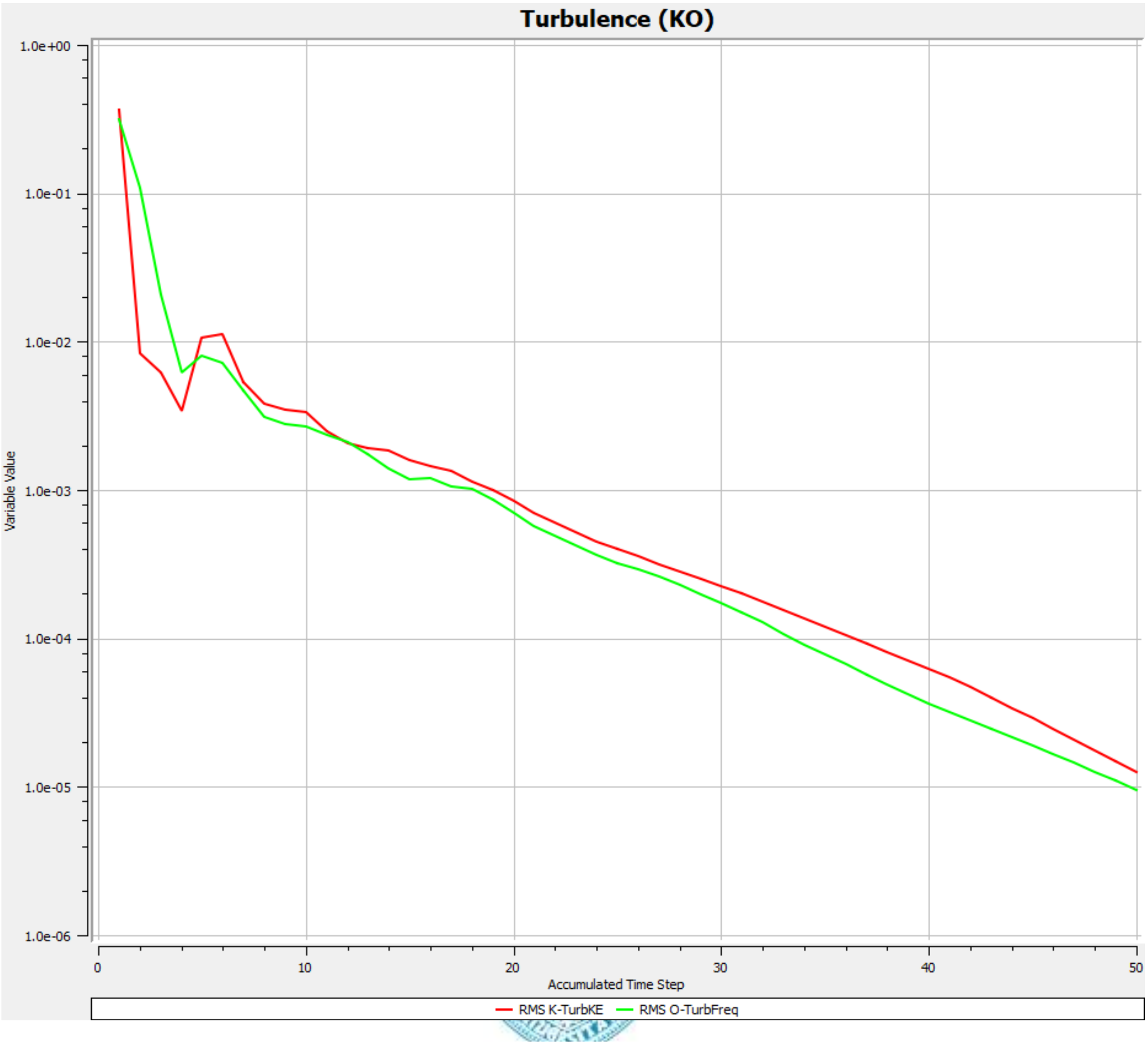

Figure 10: Convergence of turbulence for the refined iteration of airflow around a room

2.6 Results

The results obtained demonstrate that the airflow velocity decreases in the area around the room, especially behind the room. Additionally, in this low-pressure zone, turbulence increases, noticeable by the presence of vortices.

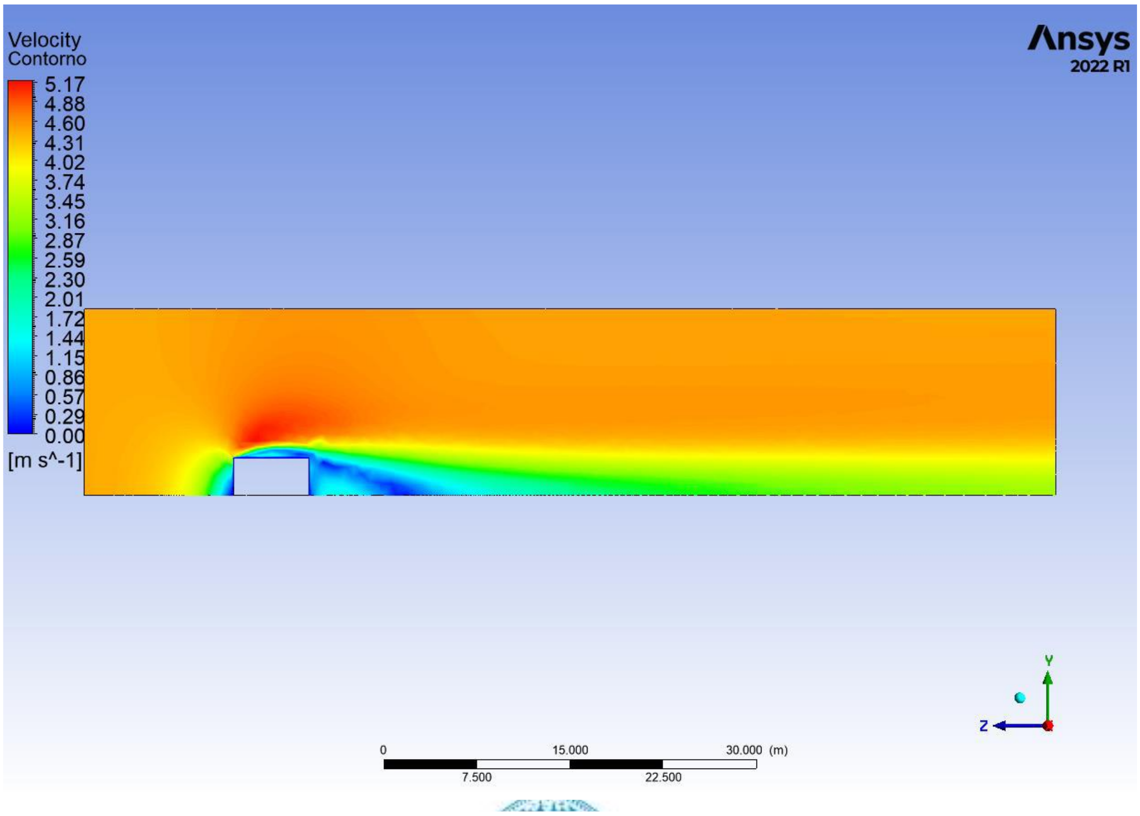

Figure 11: Contour of the velocity distribution for the unrefined iteration of airflow around a room

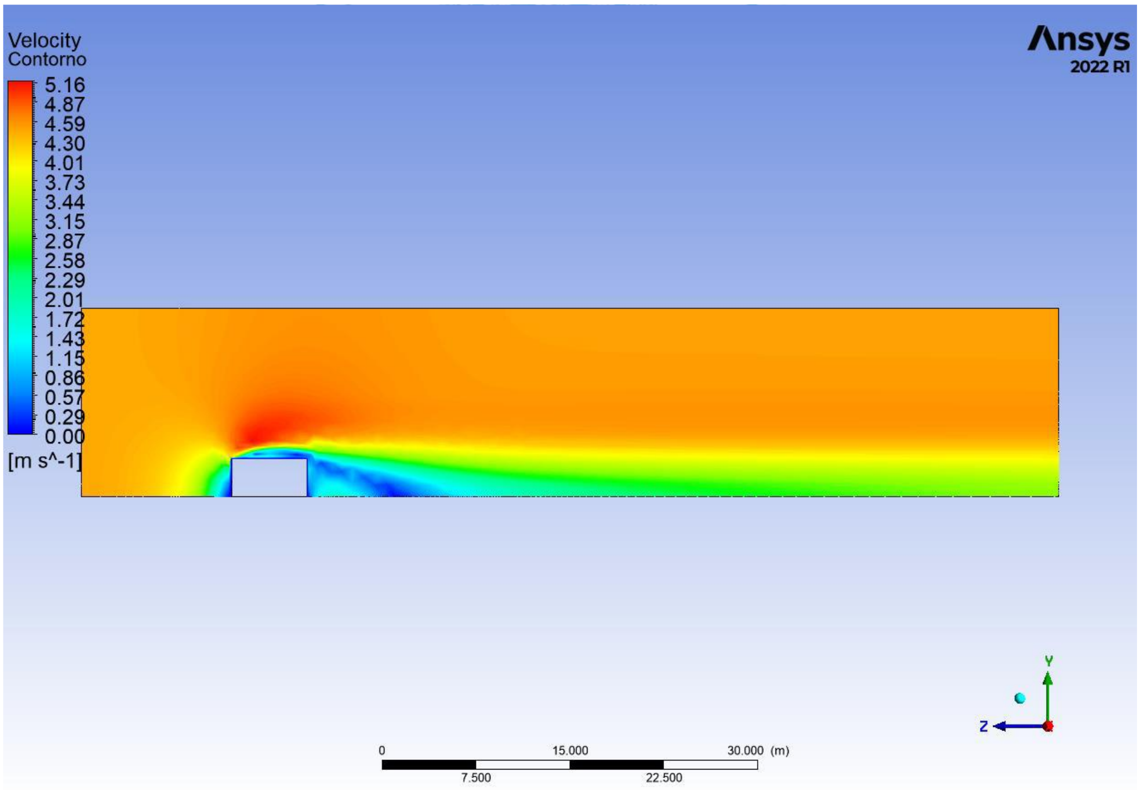

Figure 12: Contour of the velocity distribution for the refined iteration of airflow around a room

Finally, the graph showing the evolution of airflow velocity at the mentioned point is presented below:

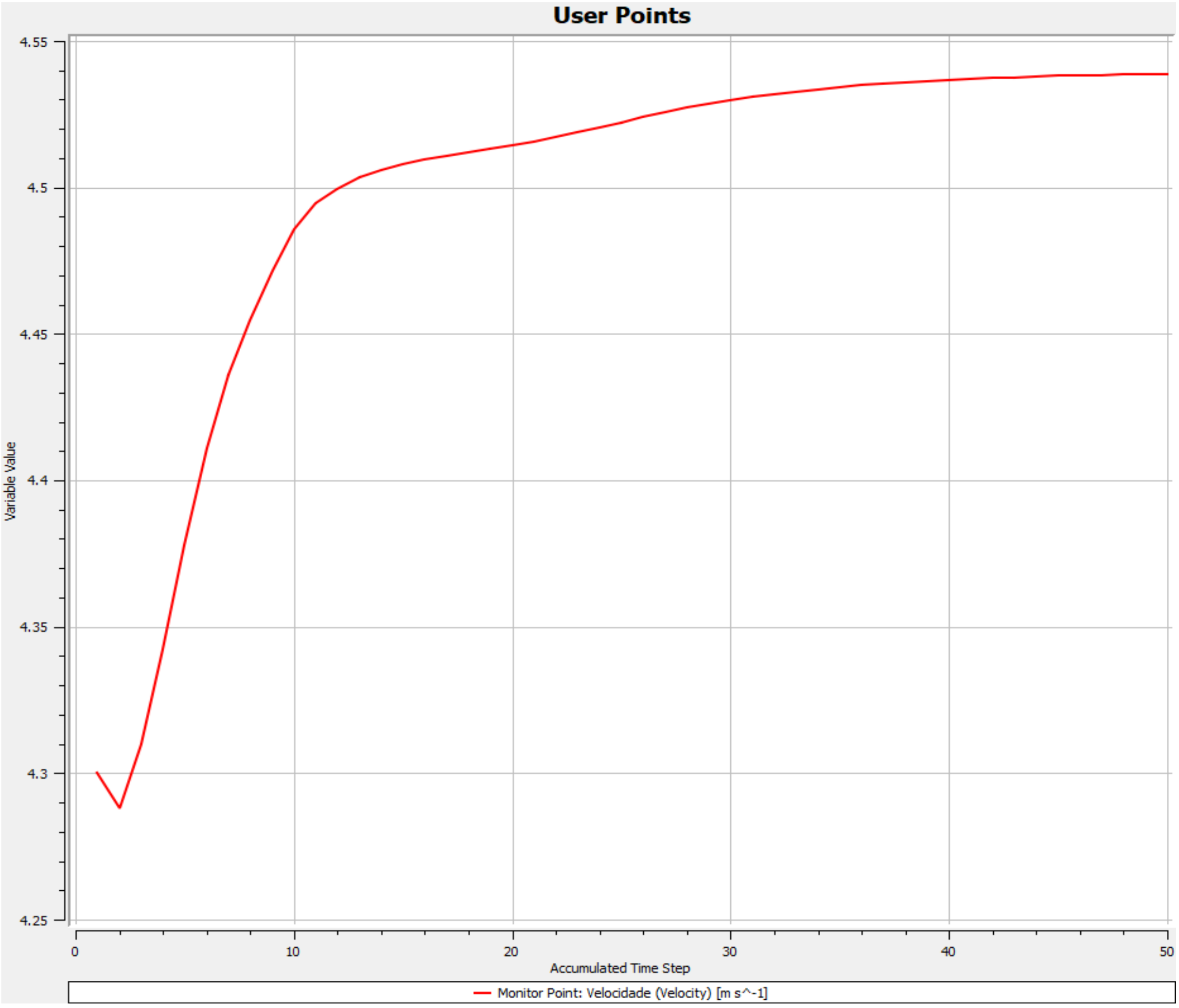

Figure 13: Evolution of airflow velocity for the unrefined iteration of airflow around a room

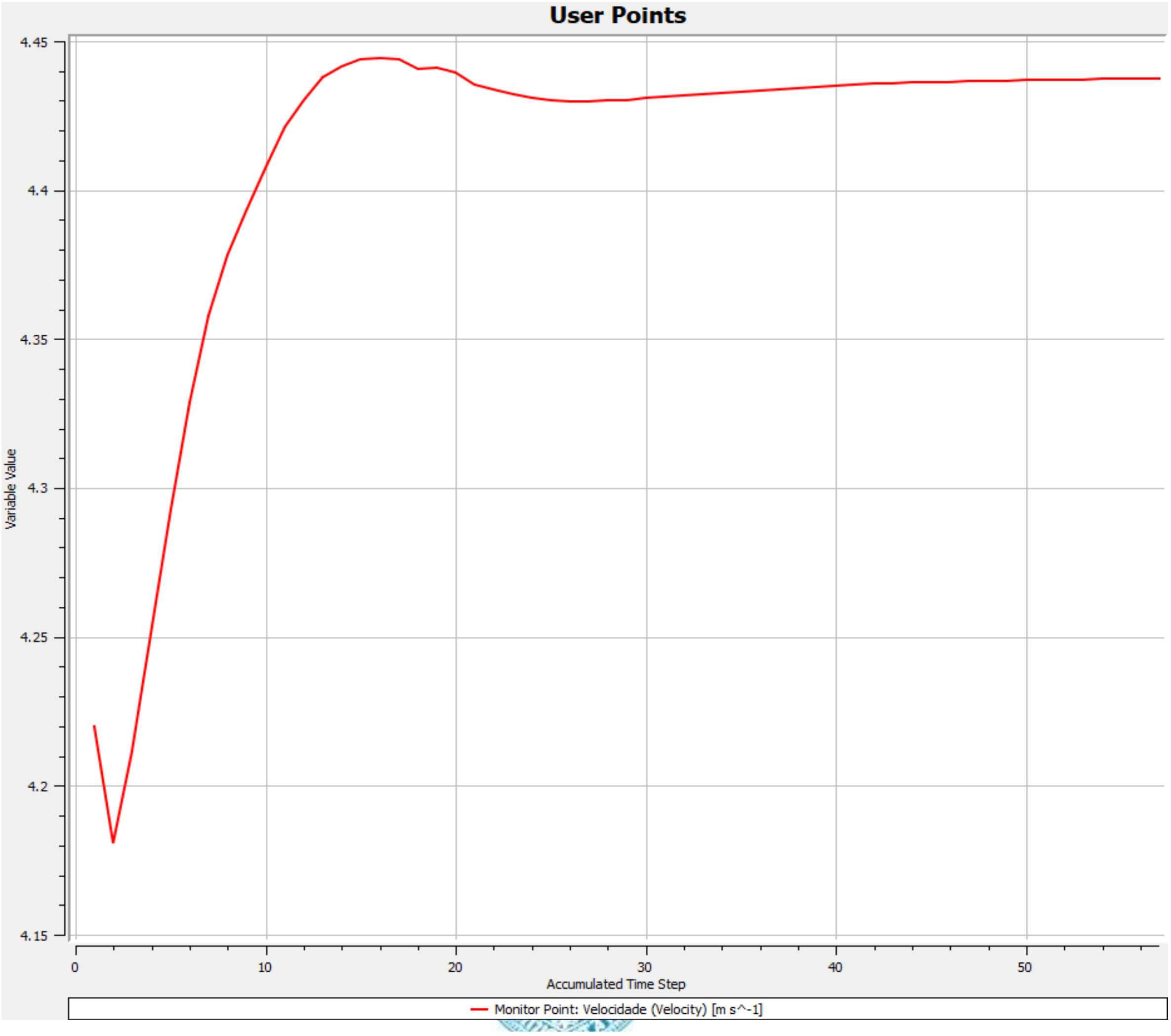

Figure 14: Evolution of airflow velocity for the refined iteration of airflow around a room

3. Airflow inside an open-loop earth-air heat exchanger tube

3.1 Objective

Simulate airflow in a ground-air heat exchanger for two summer days using the inlet temperatures provided by the educator. Also, present the heat distribution in the tube and the ground during maximum and minimum temperatures. Figure 14: Evolution of airflow velocity from the refined iteration of airflow around an inlet room into the tube, and temperatures at 3 monitoring points in the ground at different depths.

3.2 Geometry

The geometry was similarly created in AutoCAD Autodesk Inventor Professional 2023 and consists of 3 parts.

First, the tube, which has an internal nominal diameter of 180 mm and a thickness of 2 mm, is constructed with a length of 6 m buried 1.9 m below the ground. The air occupies the internal space of the tube. Lastly, we have the ground, which resembles a parallelepiped with dimensions of 1 m width by 8.4 m length by 3 m depth. The tube is symmetrically inserted into the ground with the air inlet and outlet at the surface.

Due to the project's symmetry, only half of the project was built, and the "symmetry" function of CFX was used later to obtain the results and thus save simulation time.

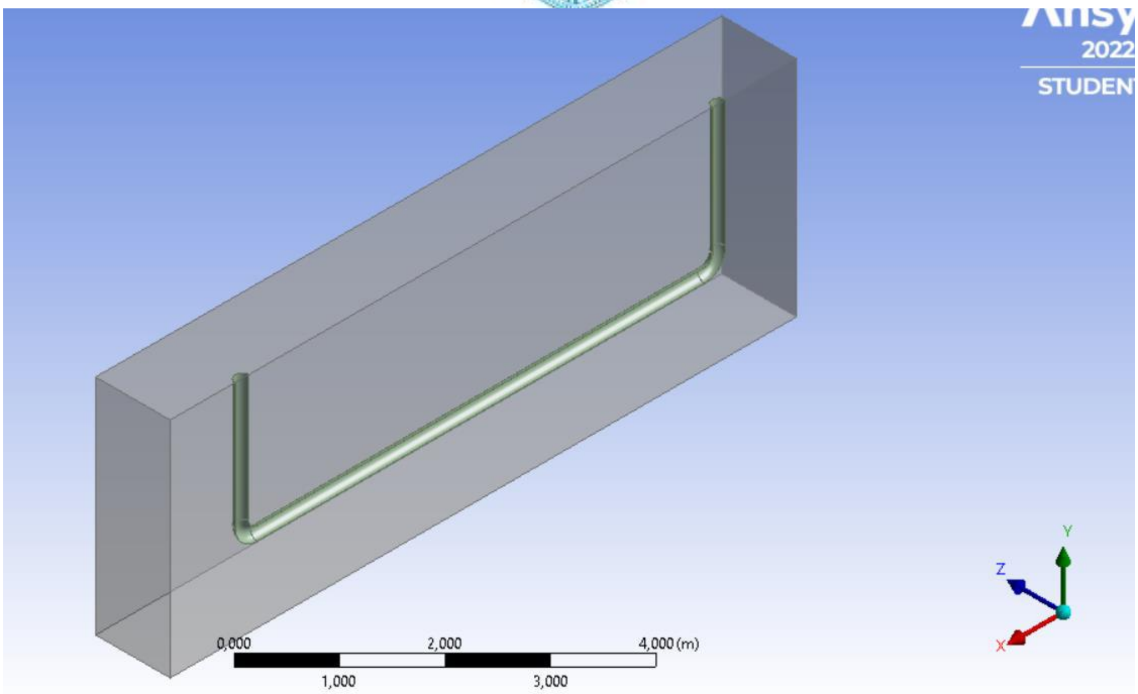

Figure 15: Ground-tube-air geometry

3.3 Mesh

The first simulation was conducted using an automatically generated mesh.

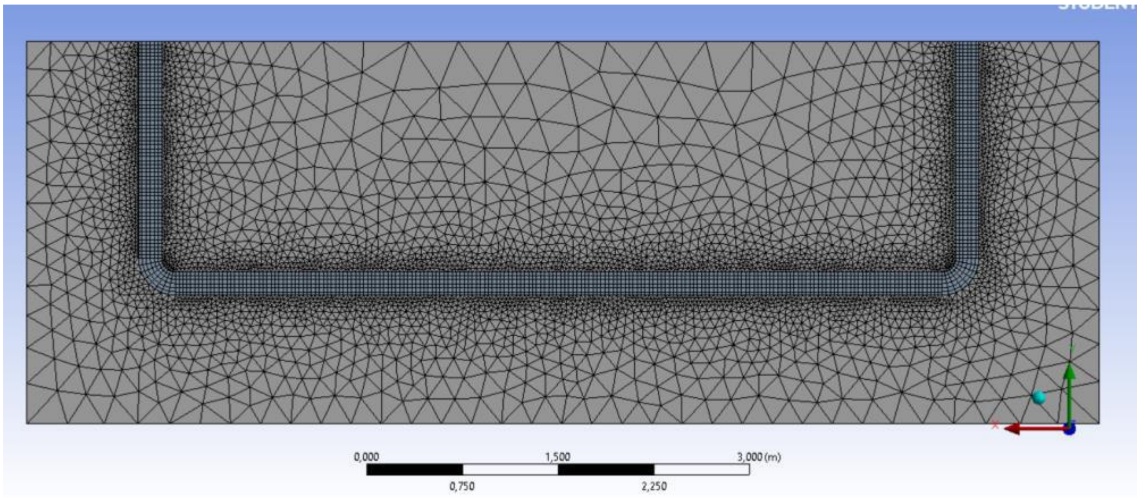

Figure 16: Unrefined mesh

To refine the mesh, inflation was introduced into the air mesh, focusing more on areas closer to the tube, which consequently experience more friction and disturbances due to surface roughness. To calculate the thickness of the first layer, the provided spreadsheet in the course materials was used, where the internal diameter of the tube, mass flow rate, and air characteristics were inputted, resulting in a thickness of 0.0003 m. Additionally, a mesh with an element size of 0.005 m was created, but due to license limitations, we were only able to simulate the unrefined mesh.

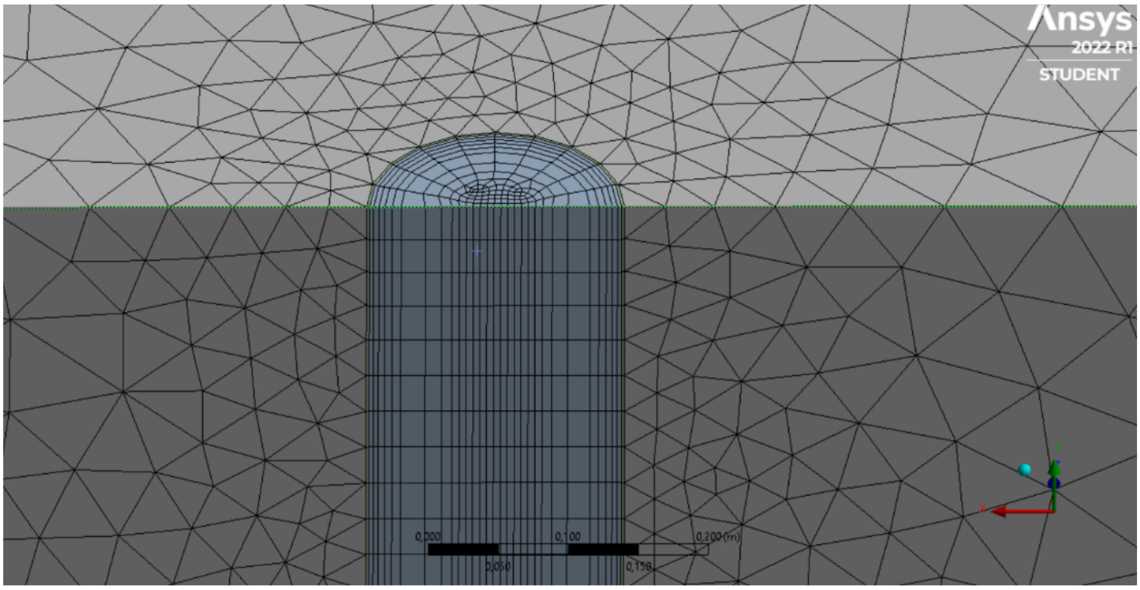

Figure 17: Inflation detail in the mesh

3.4 Setup

For the setup of this simulation, the following conditions were imposed:

- A transient simulation with a total simulation time of 2 days (172800 seconds) and a time step of 360 seconds;
- The air starting the simulation inside the tube is at 17°C and has no initial movement;
- The air enters the inlet with a mass flow rate of 0.08444 kg/s, which is half of the value given in the statement due to simulating only half of the tube;
- The inlet air temperature is obtained using a .txt file that provides the air temperature throughout the day. A function *InletT* is created for its application, which is then transformed into a time-dependent function Inlet T and set as the static temperature in the inlet details;
- The outlet has a relative pressure of 0 Pa;

- The tube was simulated as being made of steel with an initial temperature of 17°C;
- A new material was created in the program's database for the soil with the characteristics requested in the statement;
- The soil is also initialized at 17°C.

To simulate only half of the project, it was necessary to create boundaries for each domain at the symmetry point, which were then defined as symmetry boundaries.

In the solver control, we have an RMS of 0.00001 and a conservation target of 0.01.

For the output control, we monitor the temperature at the inlet, the temperature at the outlet, and the temperature at three points halfway through the tube but at different depths in the soil (0.5 m, 0.9 m, 1.5 m).

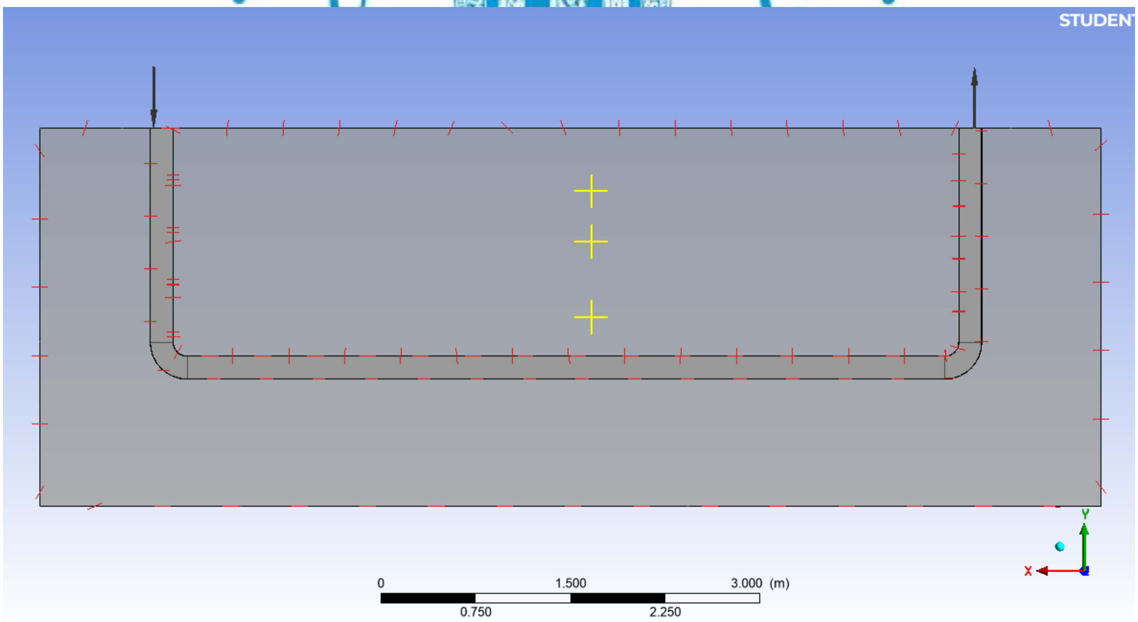

Figure 18: Control points

3.5 Solution

Confirming that the simulation converges, the obtained results are as follows:

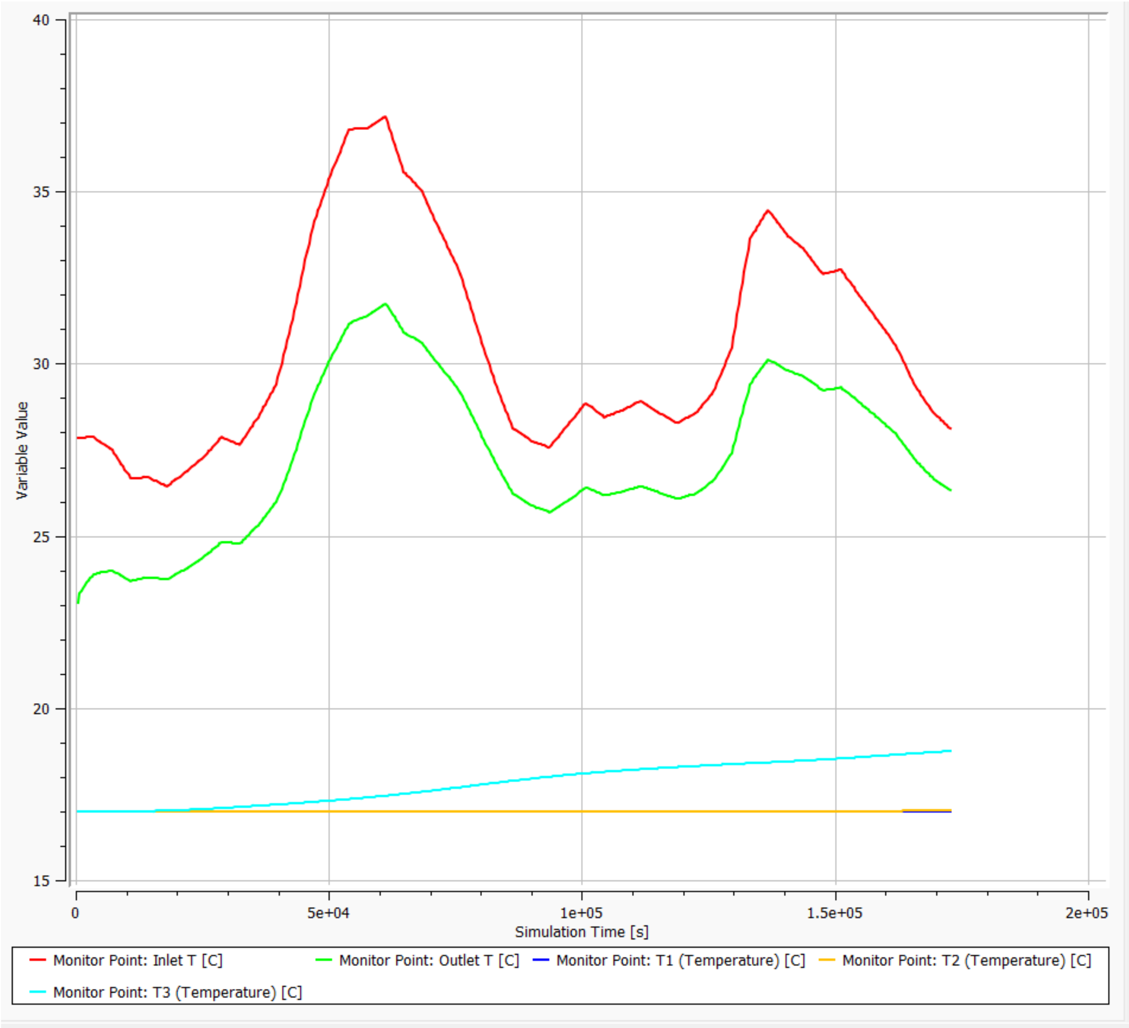

Figure 19: Mesh control points for the unrefined mesh

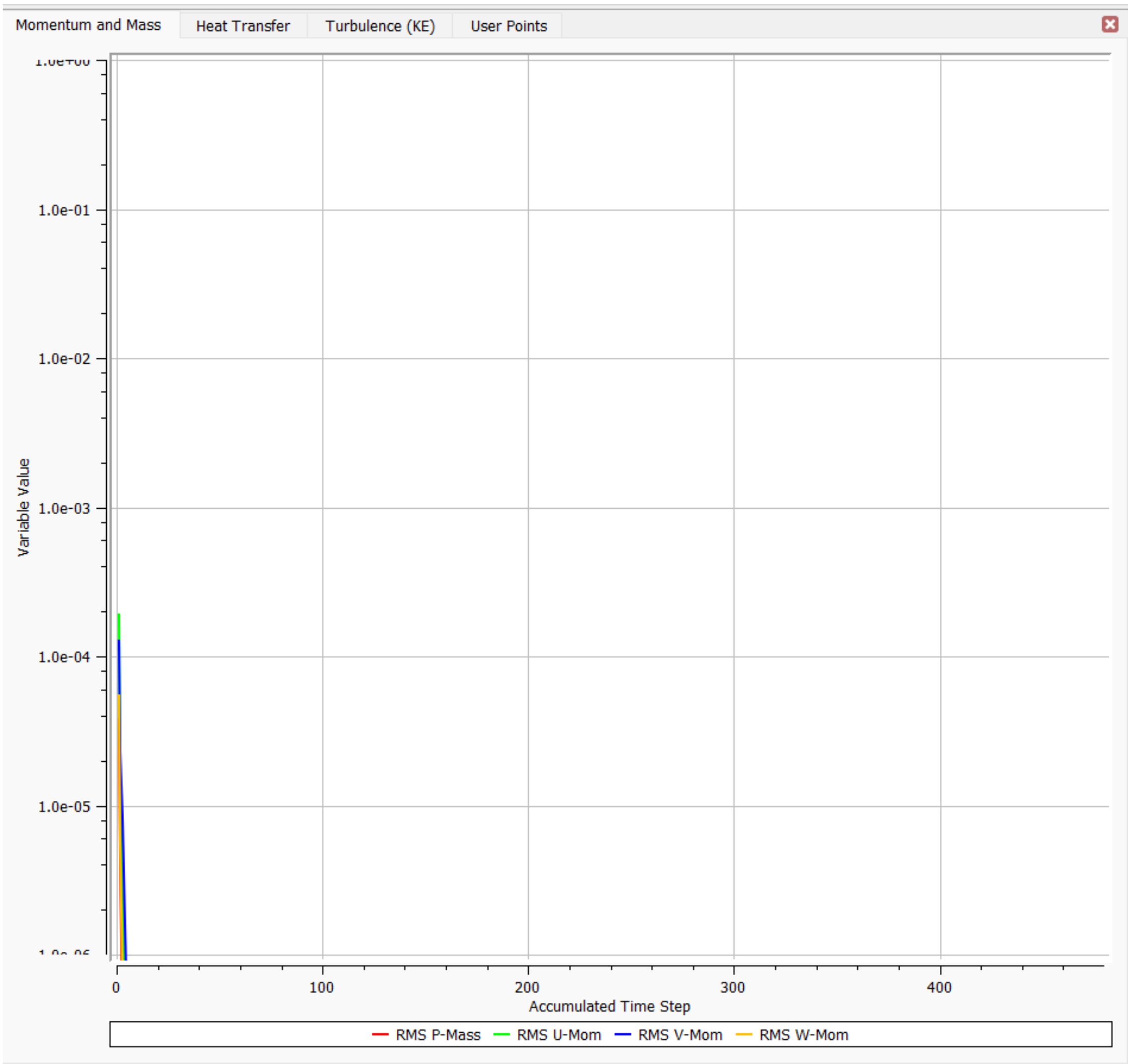

Figure 20: Convergence of momentum and mass

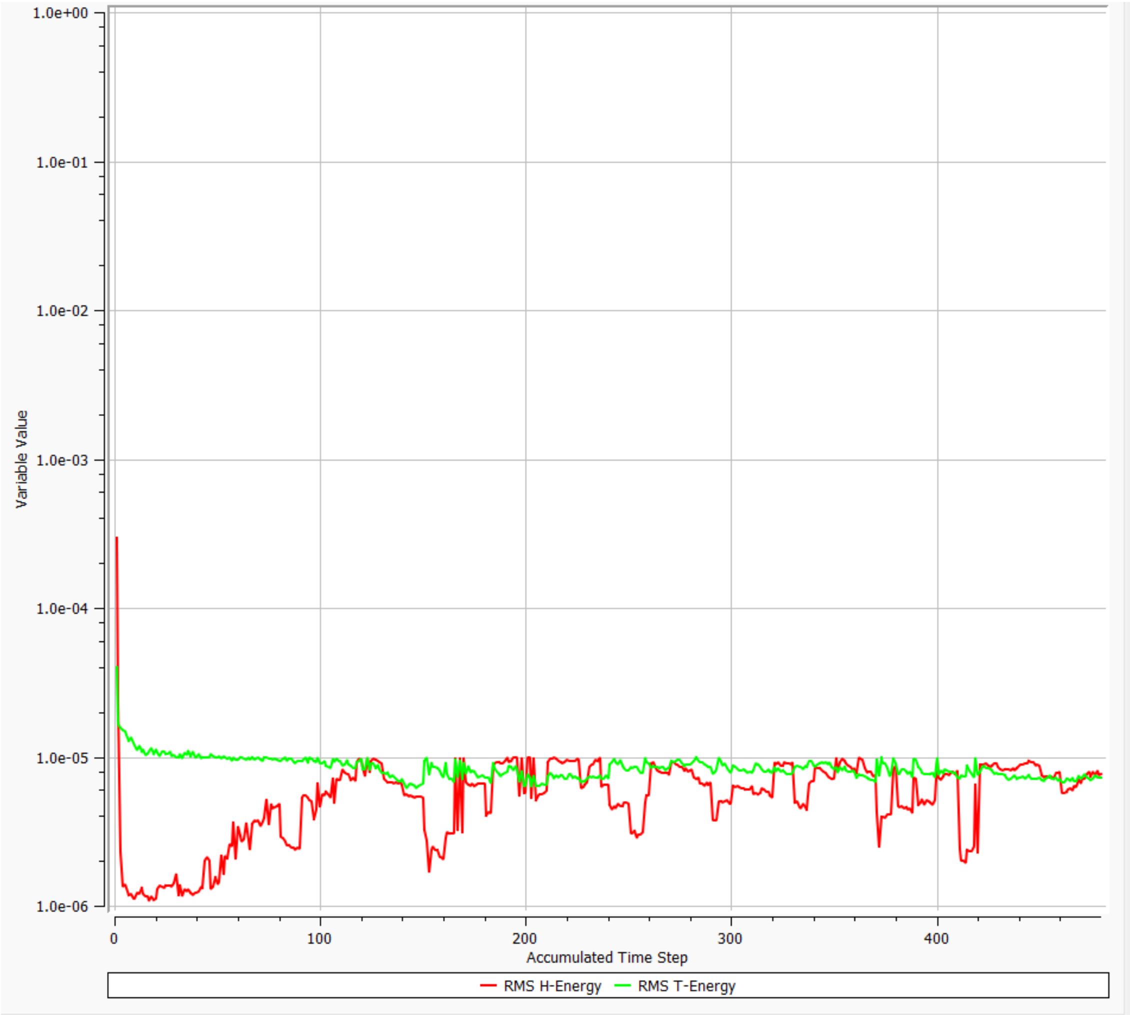

Figure 21: Convergence of heat transfer

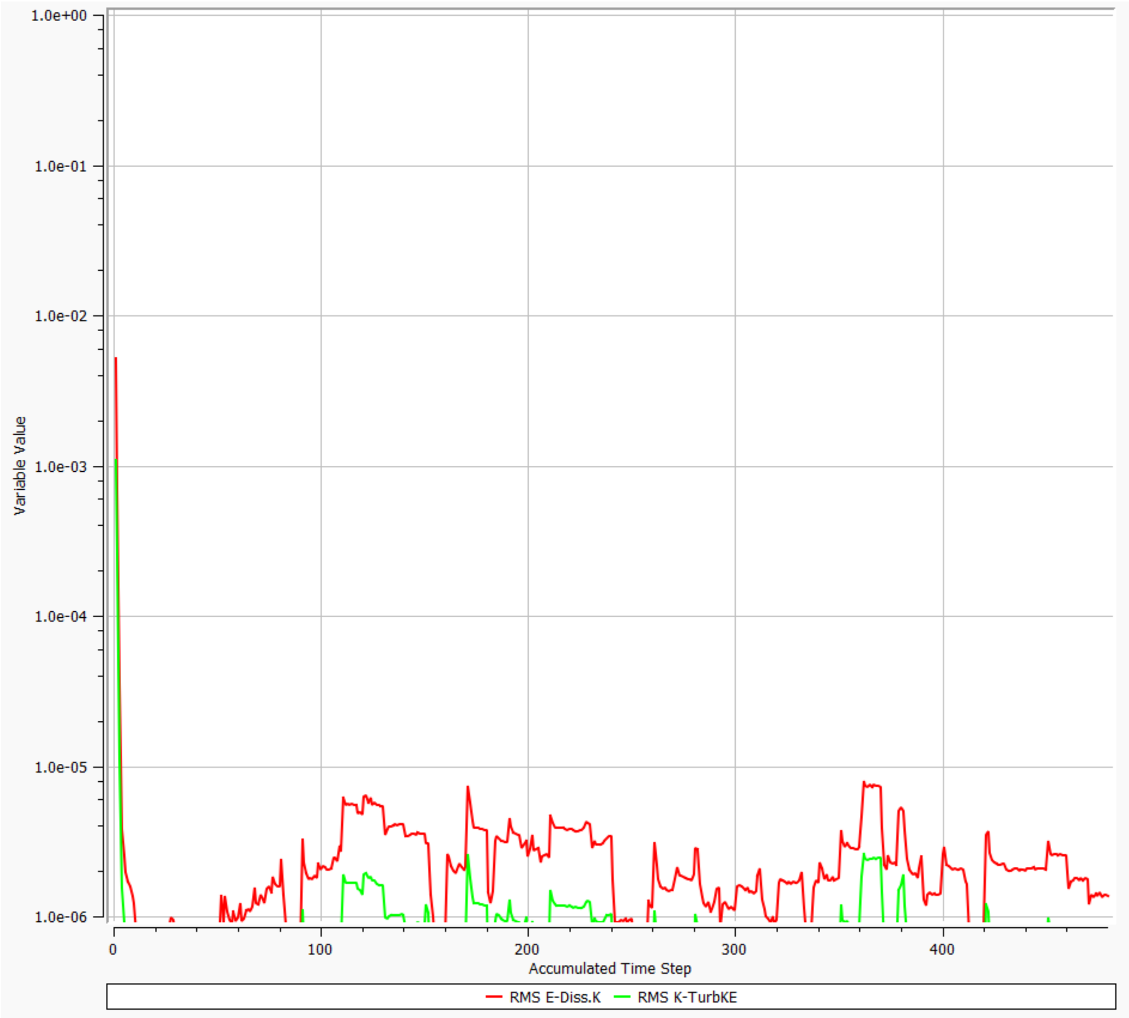

Figure 22: Convergence of turbulence

3.6 Results

The results confirm that the air cools upon contact with the cooler ground and conversely, the ground gradually heats up. This can be observed in the differences between Figure 22 and 23, representing the first timestep and the final timestep, respectively.

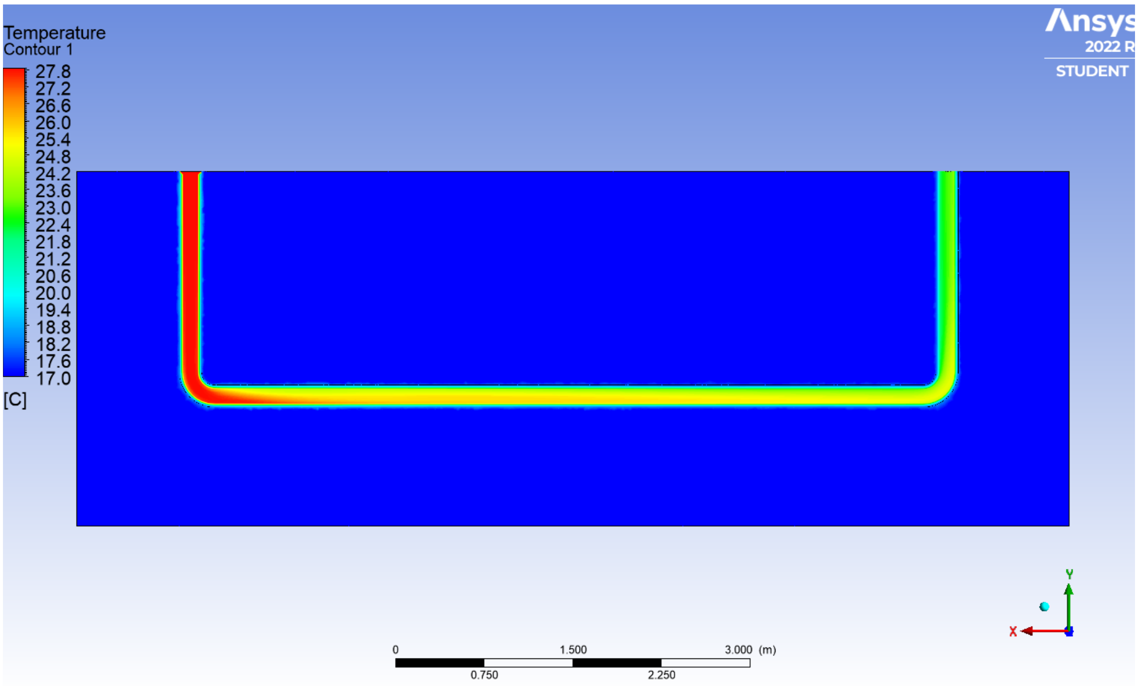

Figure 23: Temperature at the first timestep

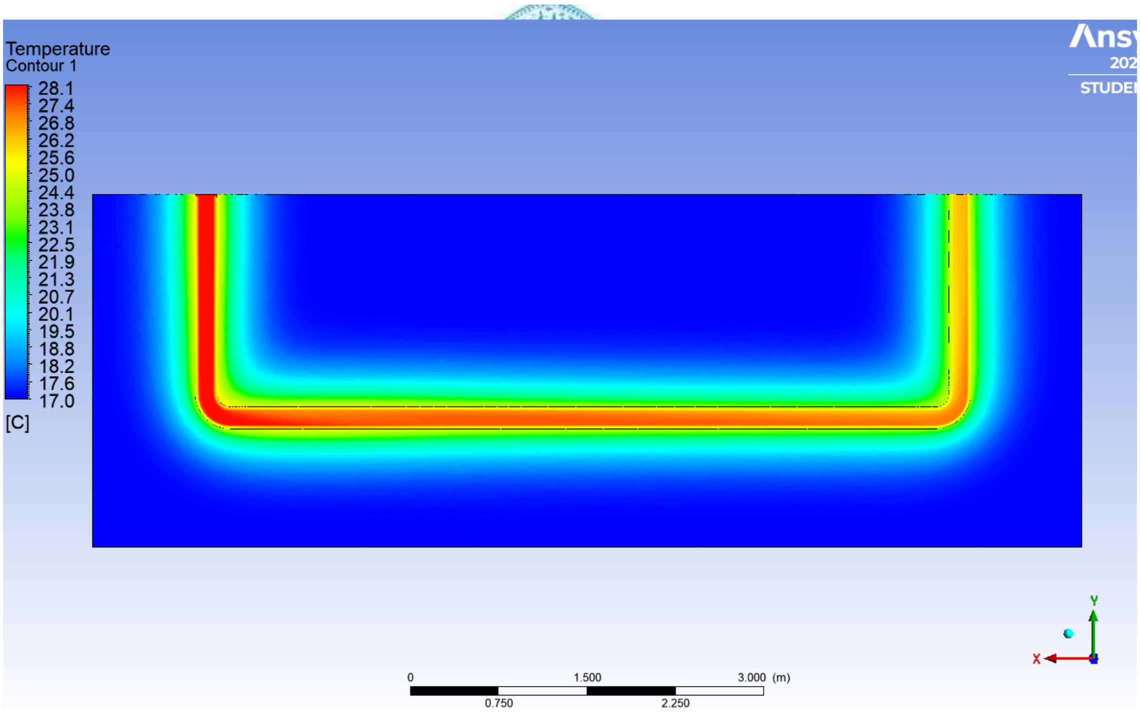

Figure 24: Temperature at the final timestep

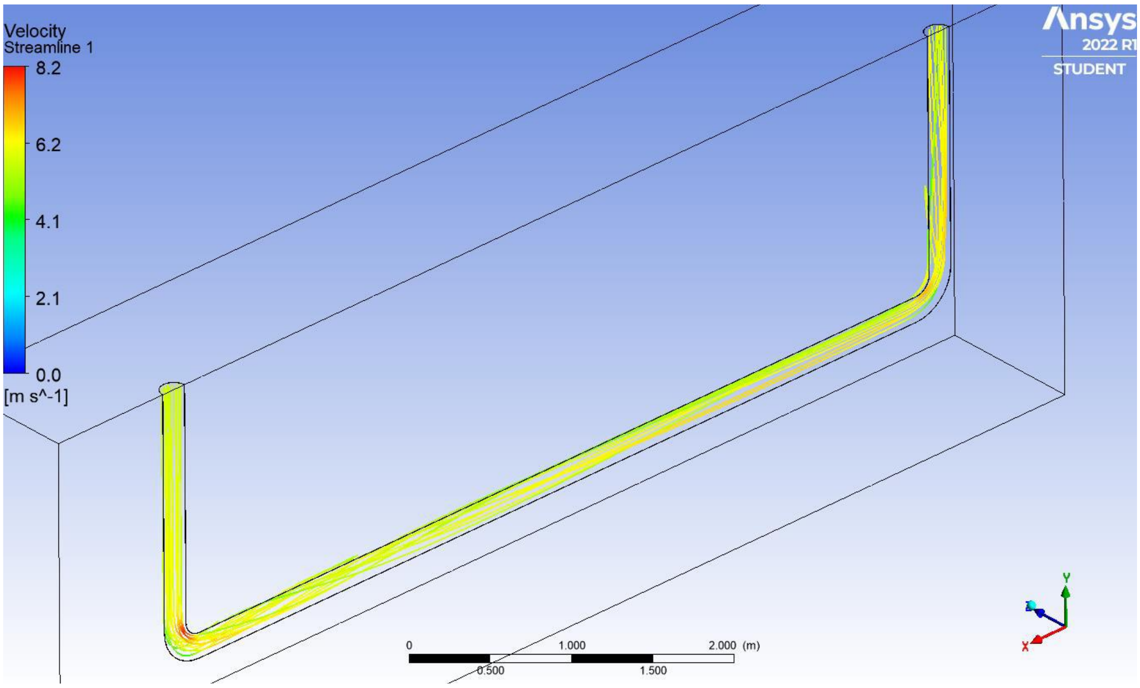

Figure 25: Airflow inside the tube

4. Airflow in a room equipped with radiators

4.1 Objective

This section combines two different requests. Firstly, it asks for the study of airflow in a room equipped with radiators and air renewal grills for a winter day in a 60-second simulation. For this purpose, a temperature monitoring point is requested, as well as the visualization of temperature flow and air stratification in the room.

The second request involves studying airflow in the same room, but now for two summer days, using the temperature output from the earth-air heat exchanger simulation in the previous section as the input for the inlet grills. The same monitoring point used in the winter day study is used for result visualization.

4.2 Geometry

The construction of this project begins with the drawing of a parallelepiped with dimensions of 6x4x3 m (LxWxH). Next, both air inlets are created with dimensions of 100x250 mm at the top left of our parallelepiped, using the "split" function of the program to obtain a new face separated from the air parallelepiped.

Using the same function, the windows (700x1200 mm), the door (2000x800 mm), and the outlet grill (400x250 mm) are also drawn. The distances and placements of these elements can be observed in the following figures.

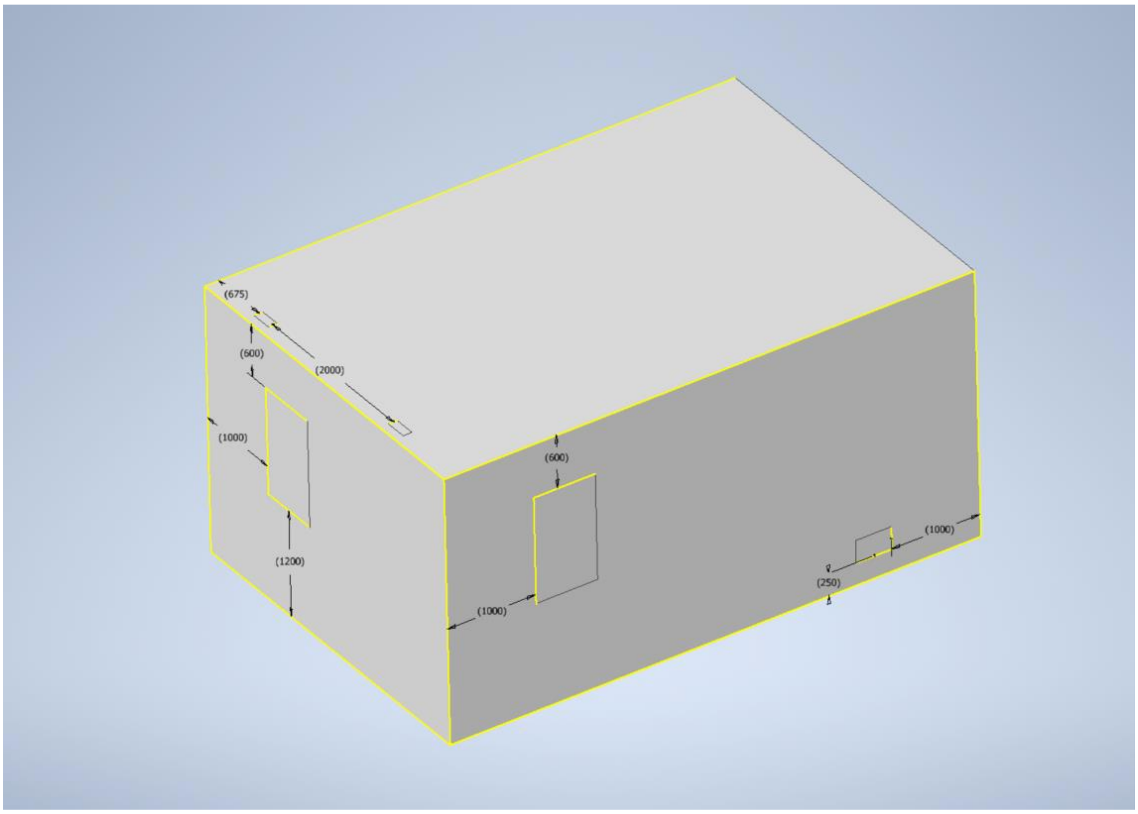

Figure 26: Geometry details

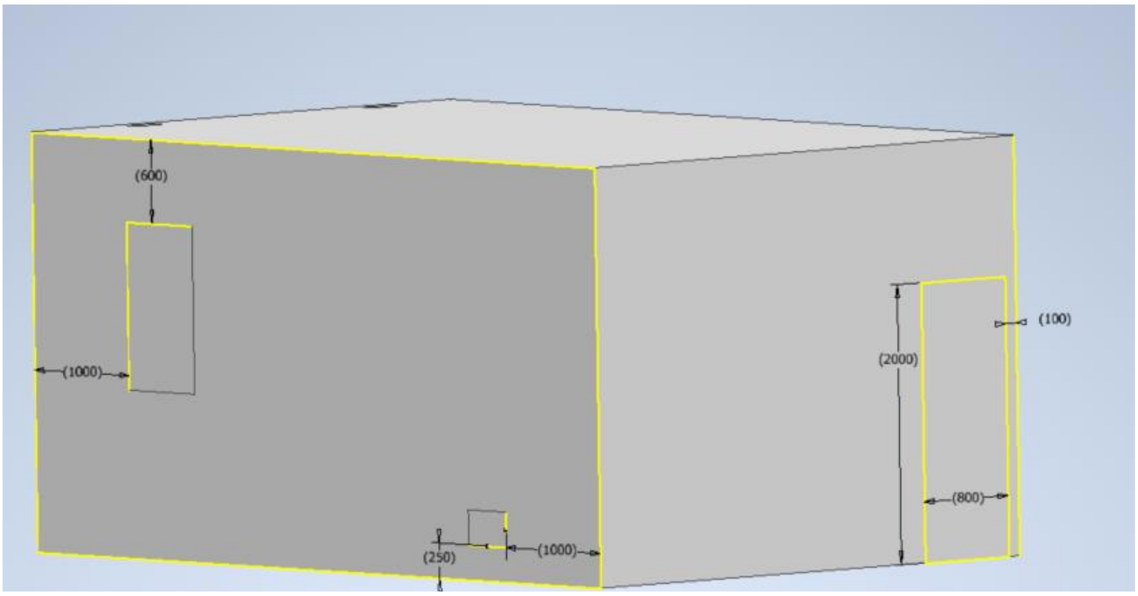

Figure 27: Geometry details

The radiators, being a new body, were created by cutting two volumes of 1200x100x700 mm from the parallelepiped using the "extrude cut" function. Then, using the same sketch, the "extrude" function was used again, adding the volume to create the two radiators as separate bodies from the air. The left radiator is located 1.4 m from the wall and 1.5 m from the other radiator.

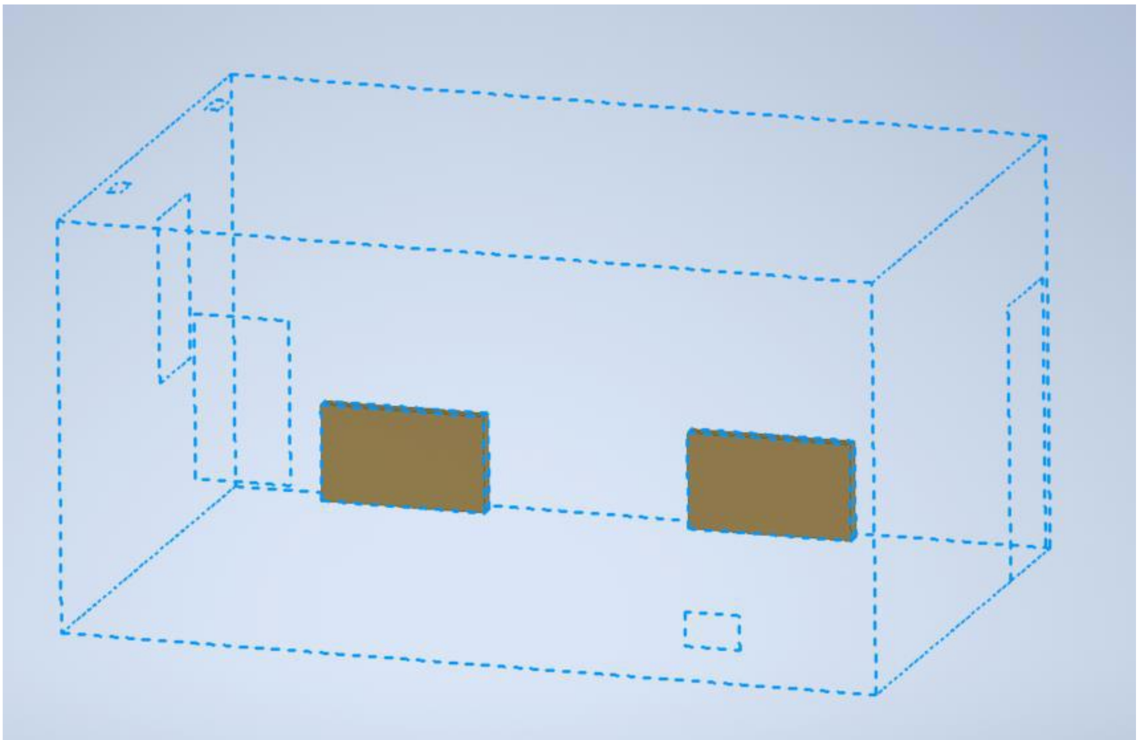

Figure 28: Positioning of the radiators

4.3 Mesh

For mesh construction, we started from the inside out, beginning with the radiators. In this regard, we used the "Insert-Sizing" function to define a mesh with element sizes of 0.04 m. Next, we did the same for the air, using elements with sizes of 0.15 m. For the inlets and outlet, we used 0.05 m, and finally, for the windows and door, we used 0.07 m.

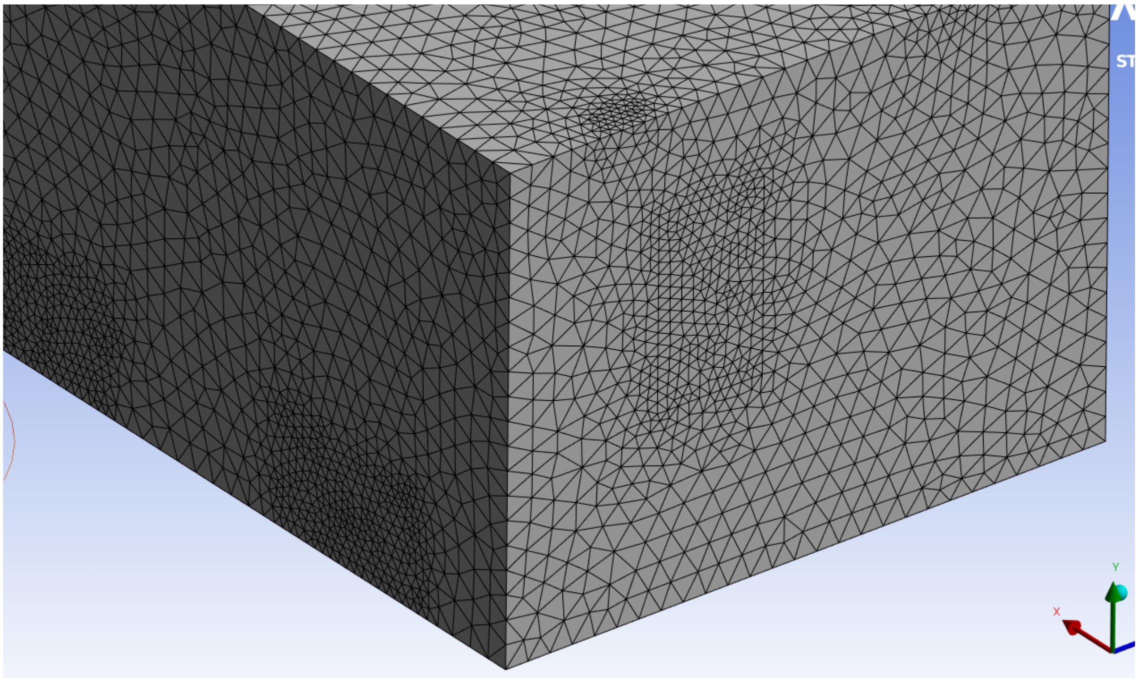

Figure 29: Details of the exterior mesh

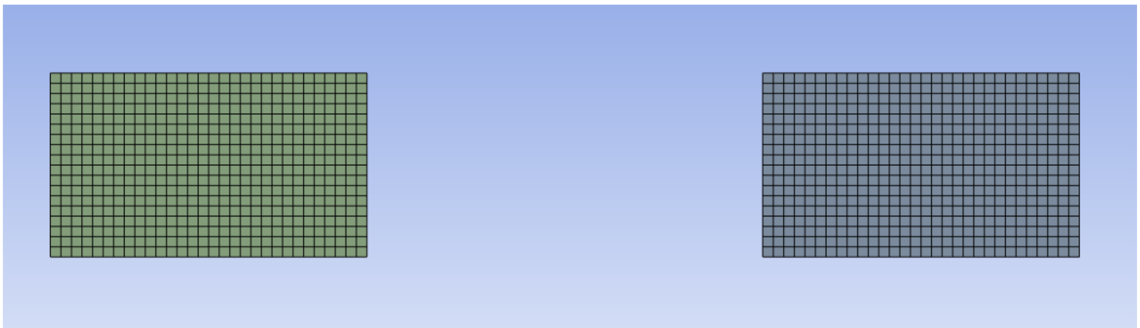

Figure 30: Mesh on the radiators

4.4 Setup for the winter day

To correctly perform the simulation, the following was defined in the setup:

- The simulation is transient and is conducted for 60 seconds with a timestep of 60 seconds;
- The parallelepiped is made of air at 25°C;
- The inlets have a mass flow rate of 0.09 kg/s each with a constant temperature of 10°C;
- The outlet functions as an opening with a relative pressure of 0 Pa and a temperature of 20°C;

- Both the walls and the door were defined as "no slip walls" with a temperature of 20°C and opaque with emissivity and fractional diffusion of 1;
- The windows are "no slip walls" with a temperature of 20°C and opaque with an emissivity of 0.94 and a diffusion of 0.2;
- The windows also exhibit radiation effects, and to simulate this, a directional radiative flux of 600 W/m² is placed in the source tab, equally divided among the 3 directional components, with negative in the vertical direction and positive in the remaining directions;
- The radiators are simulated as being made of steel;
- The simulation of radiator radiation is done using the "thermal radiation" function with the Monte Carlo option, a transfer mode of participating media, and a number of histories of 10000;
- The radiators are initialized with a temperature of 80°C;
- For the solver control, we have an RMS of 0.00001 and a conservation target of 0.01;
- The control point is created in Output Control, Monitor, and it will measure temperature at coordinates (5.8 m, 2 m, 1 m), positioned as seen in figure 29.

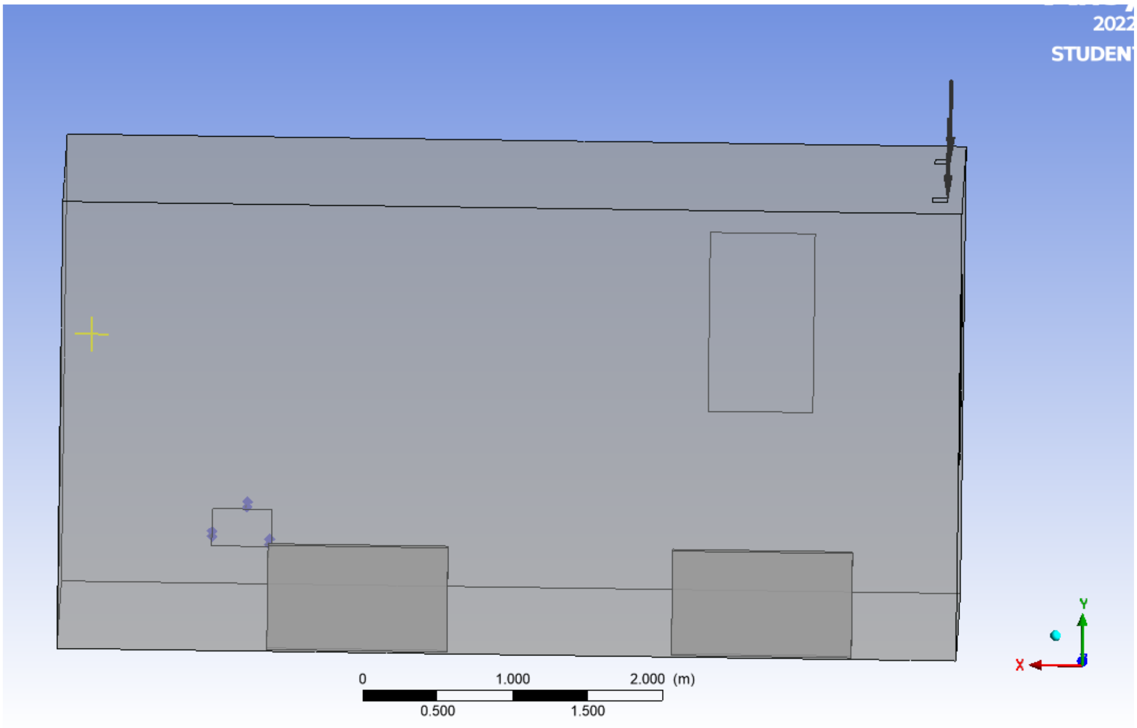

Figure 31: Positioning of the control point (thermostat)

4.5 Solution for the winter day

After completing the simulation, the following results are obtained:

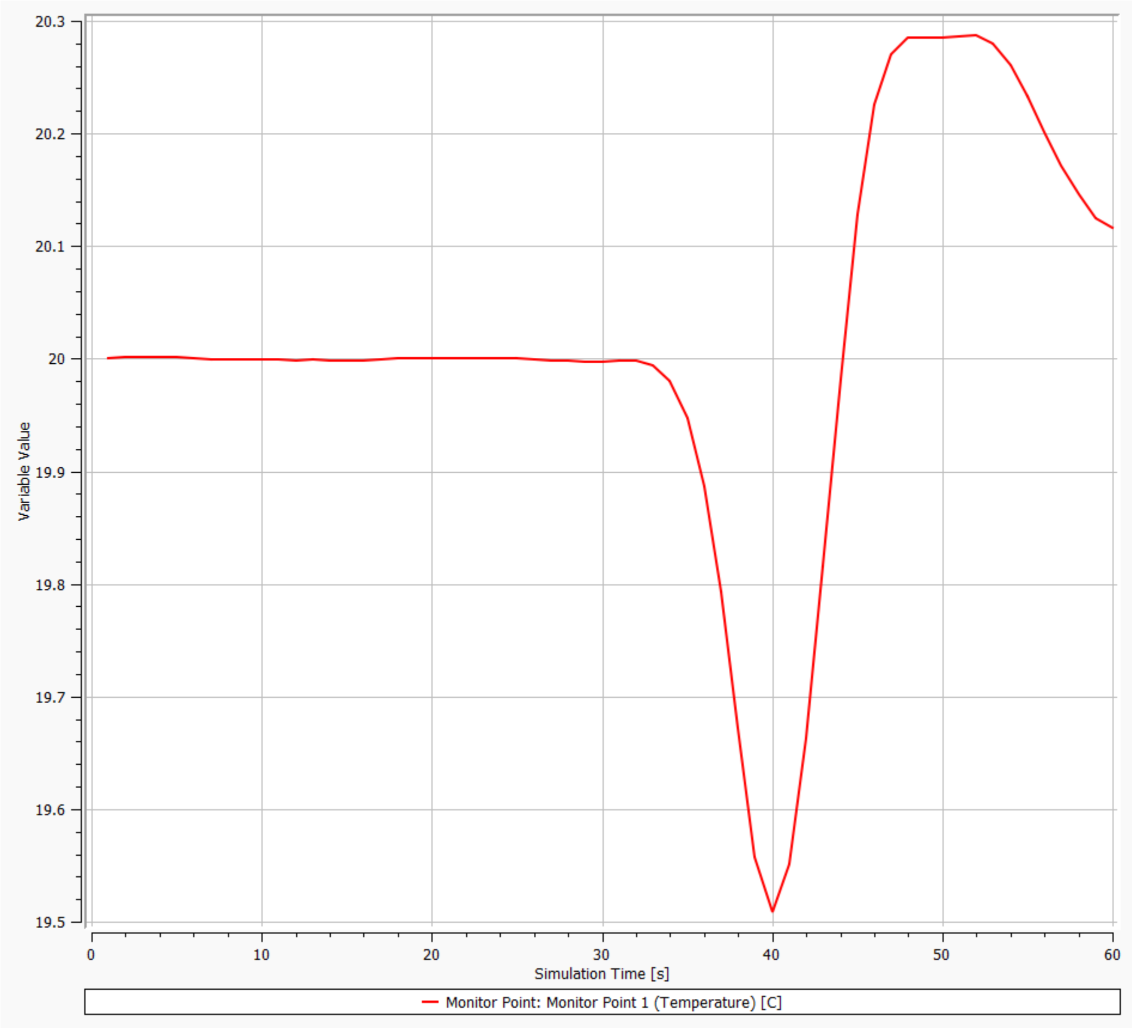

Figure 32: Temperature at the control point

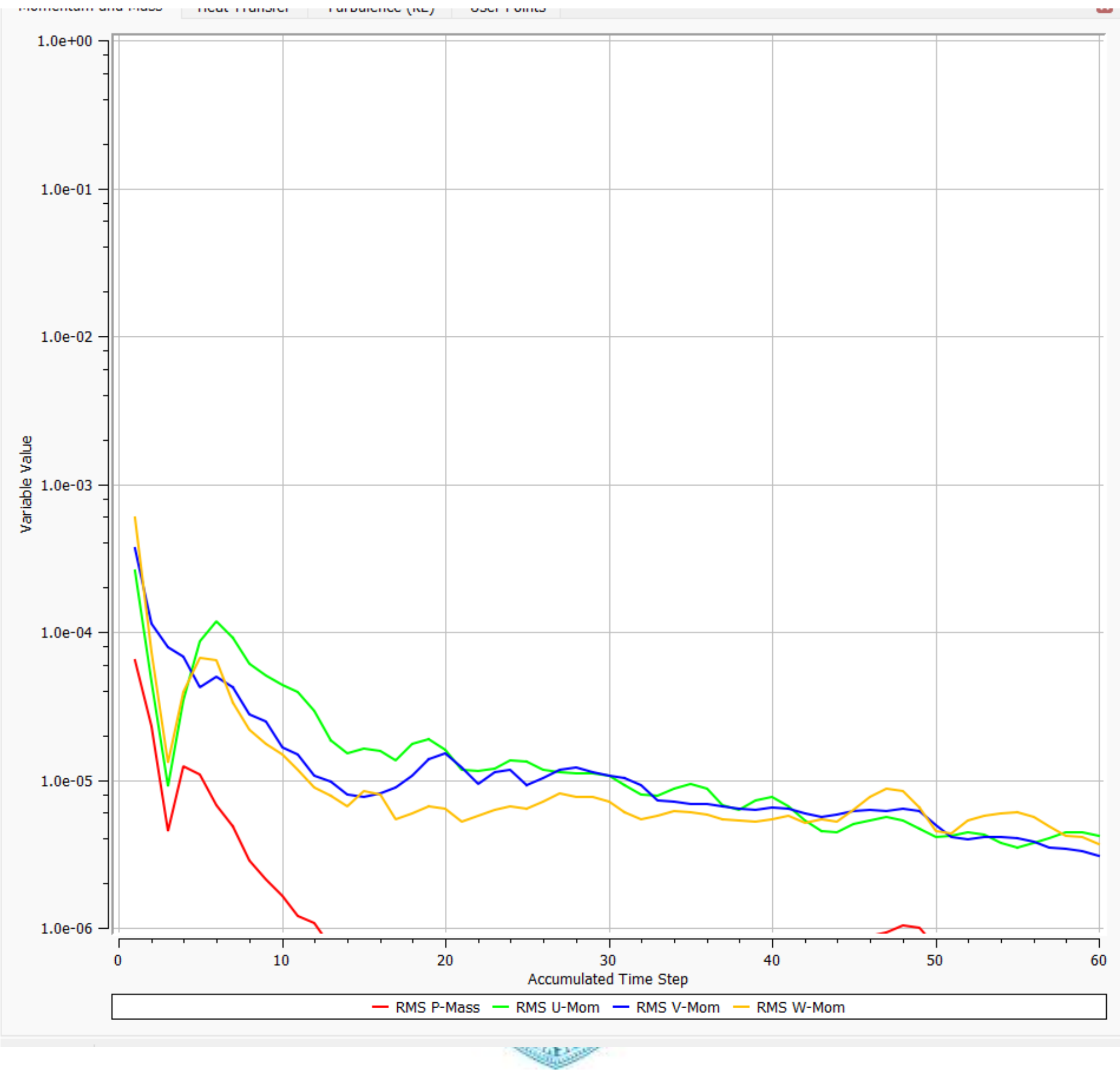

Figure 33: Convergence of momentum and mass

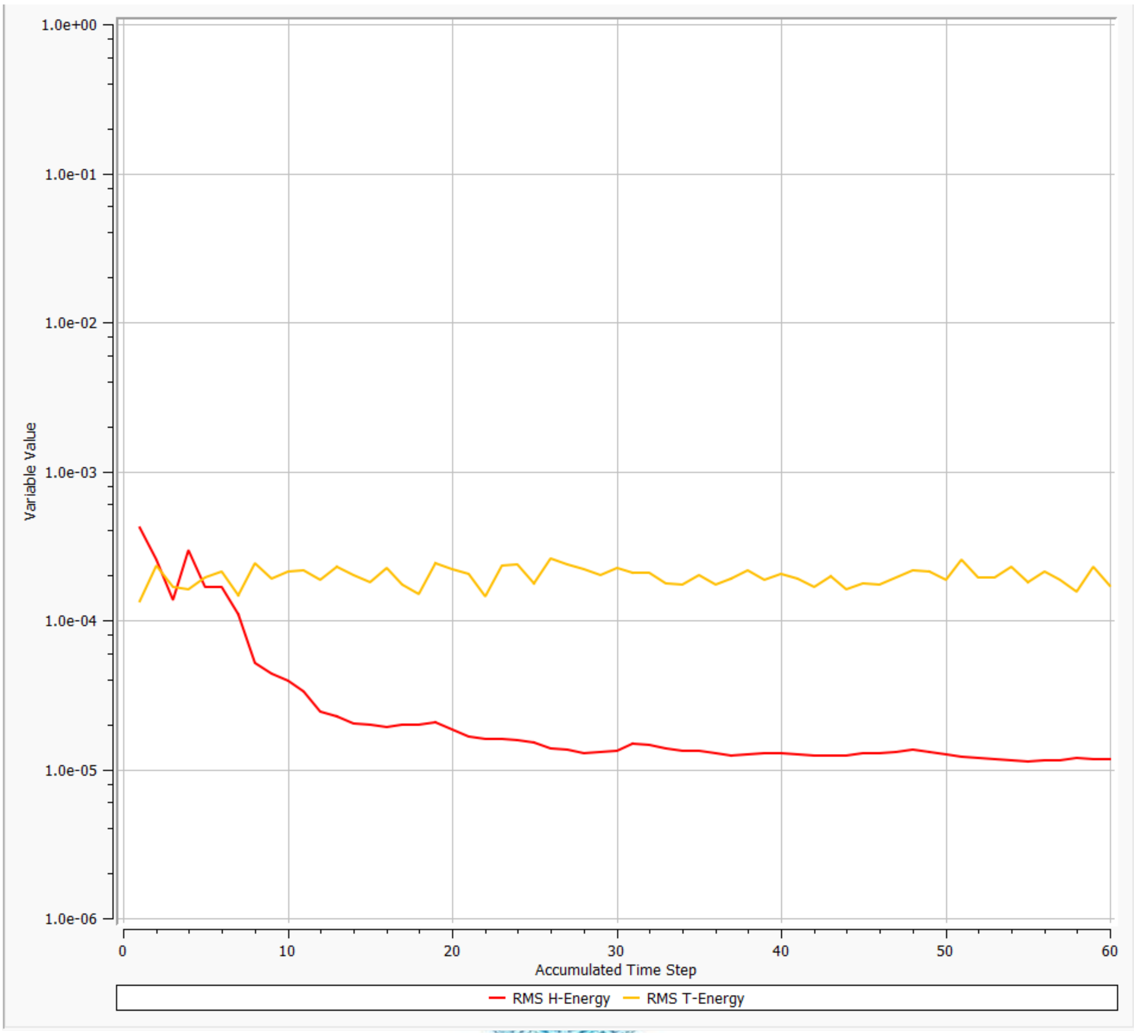

Figure 34: Failed convergence of heat transfer

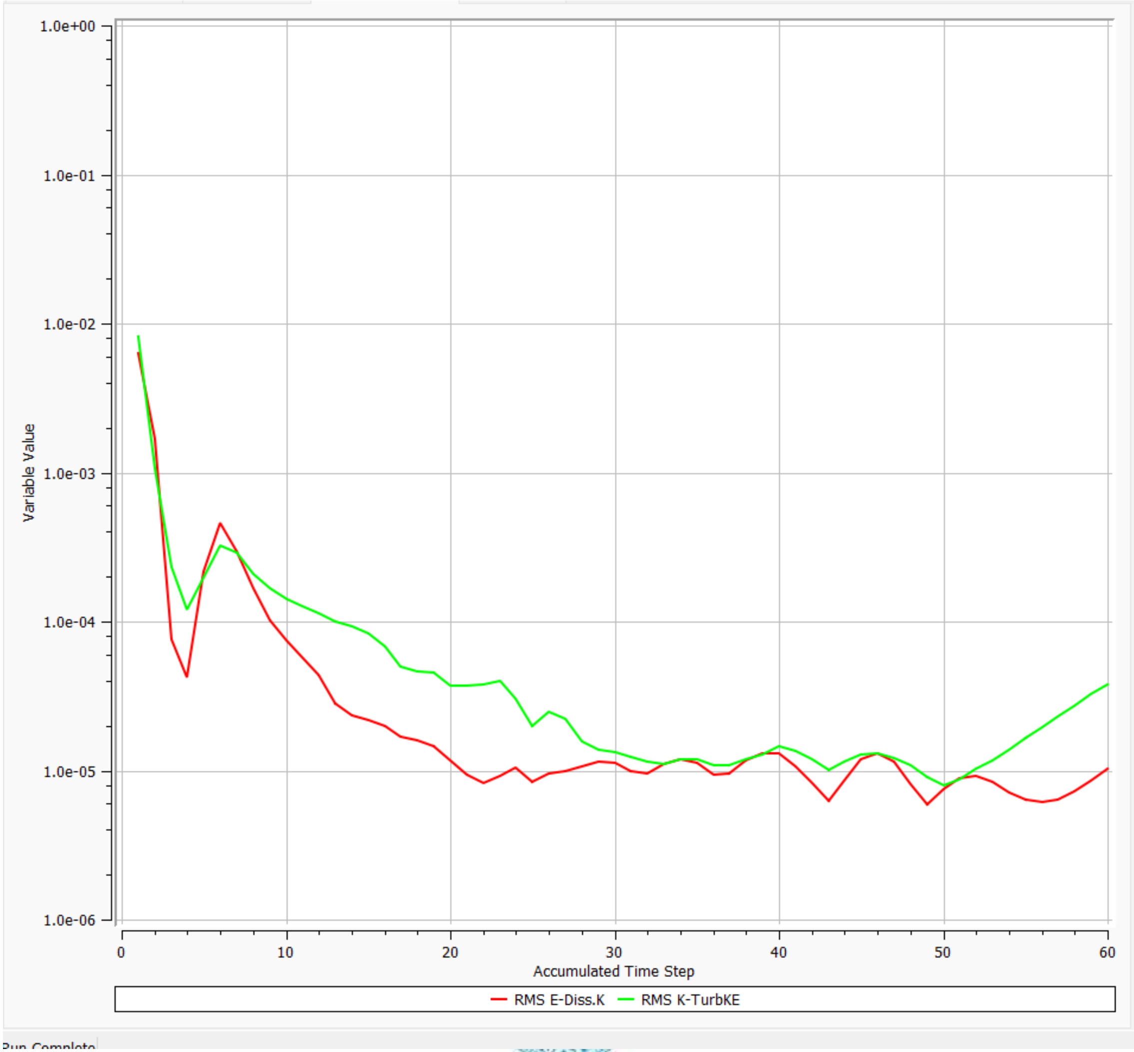

Figure 35: Failed convergence of turbulence

4.6 Results for the winter day

In figures 36 and 37, one can observe the heat flow radiated by the radiator and also the stratification of the air, where the colder zones are predominantly near the floor and the zone closer to the ceiling has a predominance of warmer air.

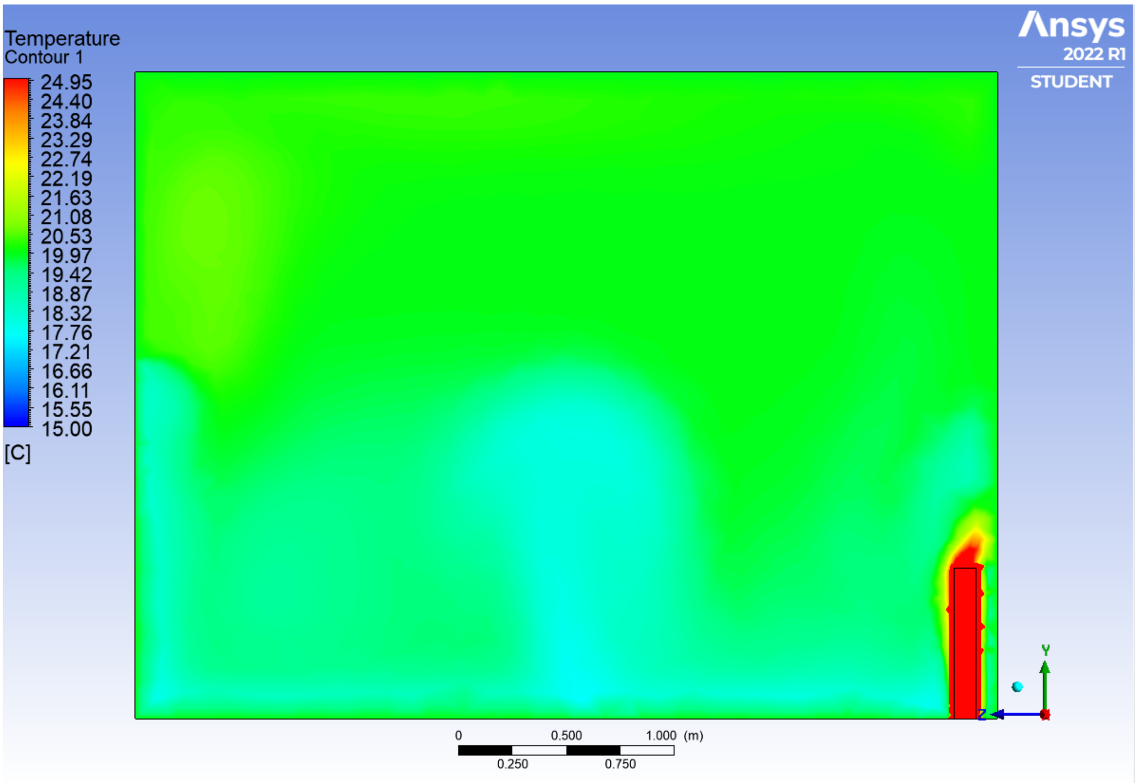

Figure 36: Stratification of the air

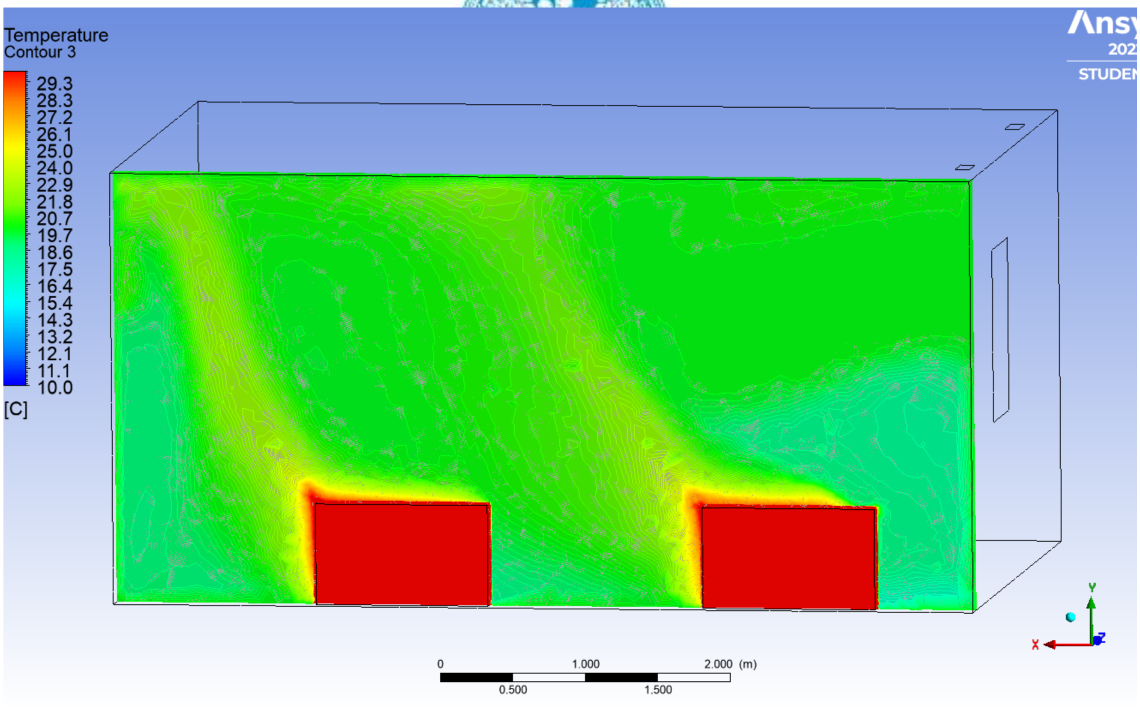

Figure 37: Radiation from the radiators

In the next figure, the airflow in the room is observed, along with the velocities it reaches.

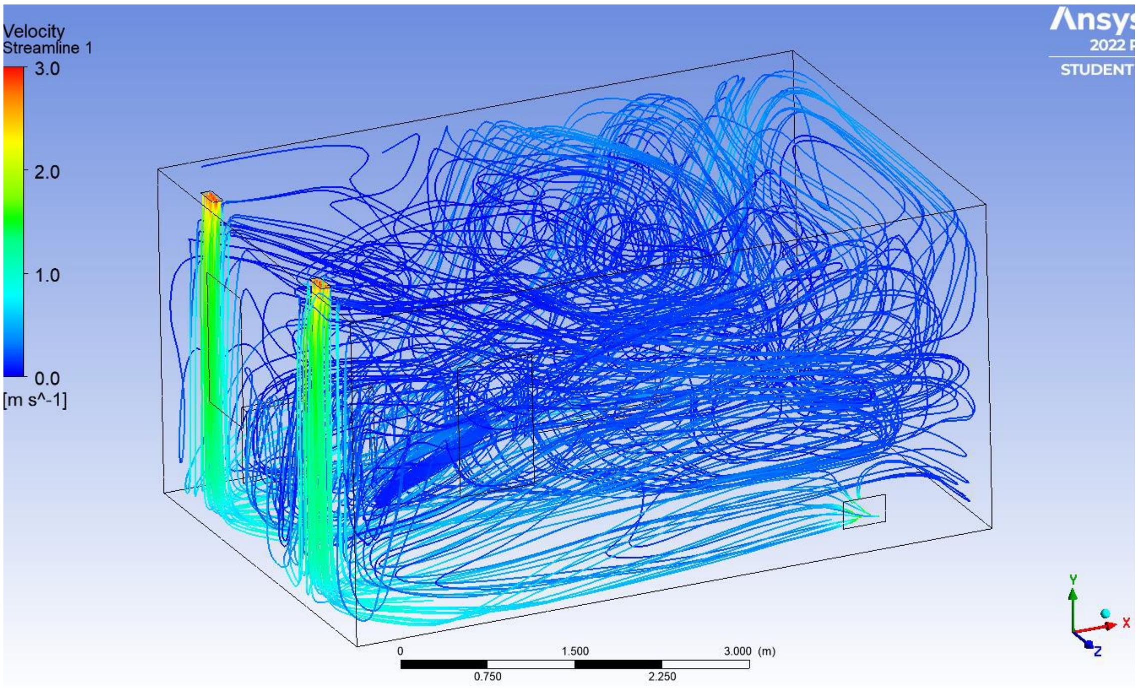

Figure 38: Airflow in the room

4.7 Setup for the summer day

The setup is quite similar to that performed on the winter day, with the main differences being:

- The simulation lasts for 2 days (172800 seconds) and has a timestep of 30 minutes (1800 seconds);
- The walls have a heat transfer coefficient of 25 W/m²K, with an exterior temperature of 25°C;
- The inlet air temperature is obtained using the temperatures obtained at the outlet of the earth-air heat exchanger from section b), which are extracted from a .txt file created from the project in section b). These temperatures are then inserted into the current project using the "user functions" function. Thus, the expression TempInlet is created, which is placed as the static temperature in the definitions of each inlet;
- The temperature of the windows is changed to 25°C;

- The temperature of the radiators is set to 25°C.

4.8 Solution for the summer day

Using the aforementioned conditions, this graph was obtained for the temperature at the thermostat (control point) for the two days in which the simulation was performed.

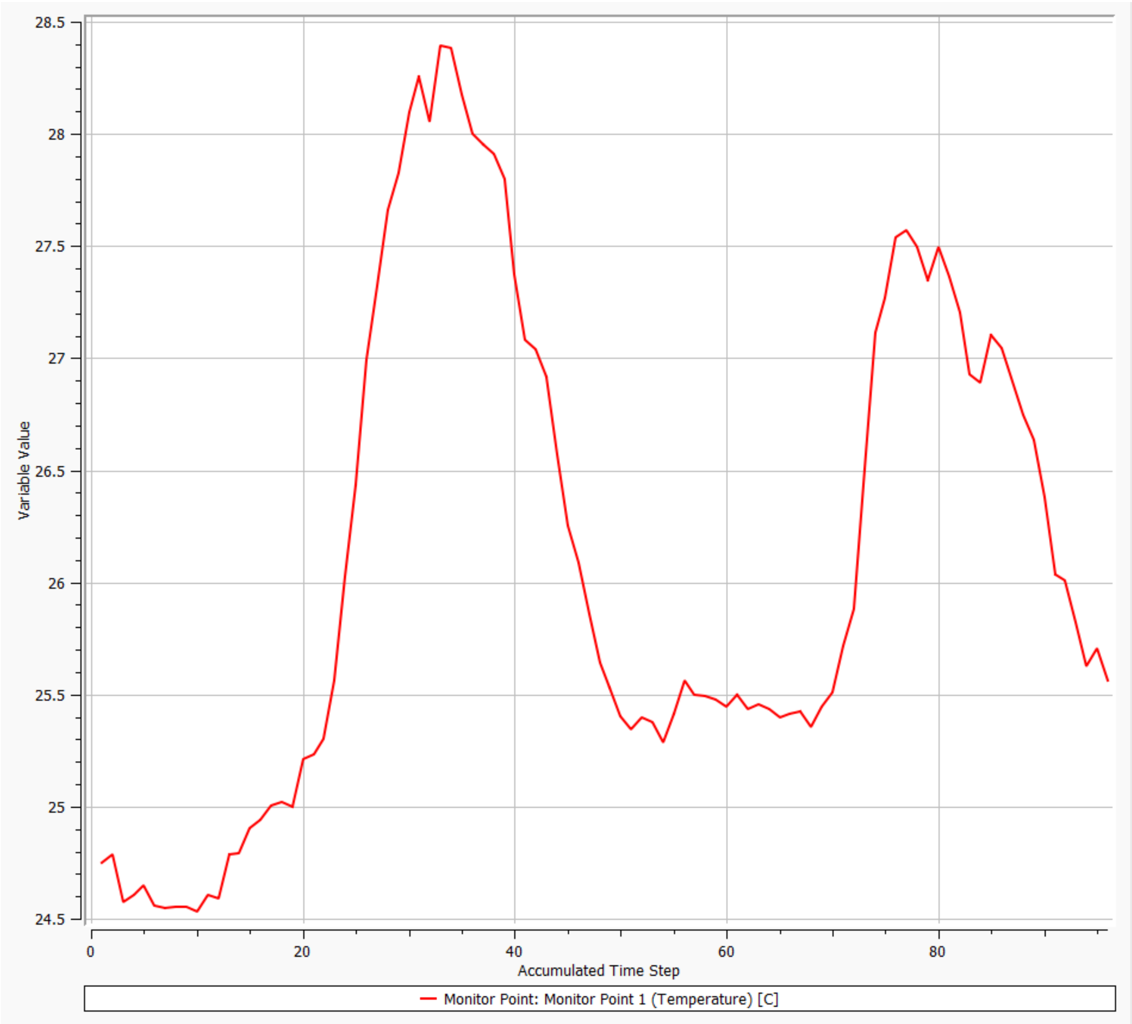

Figure 39: Thermostat (control point) temperature

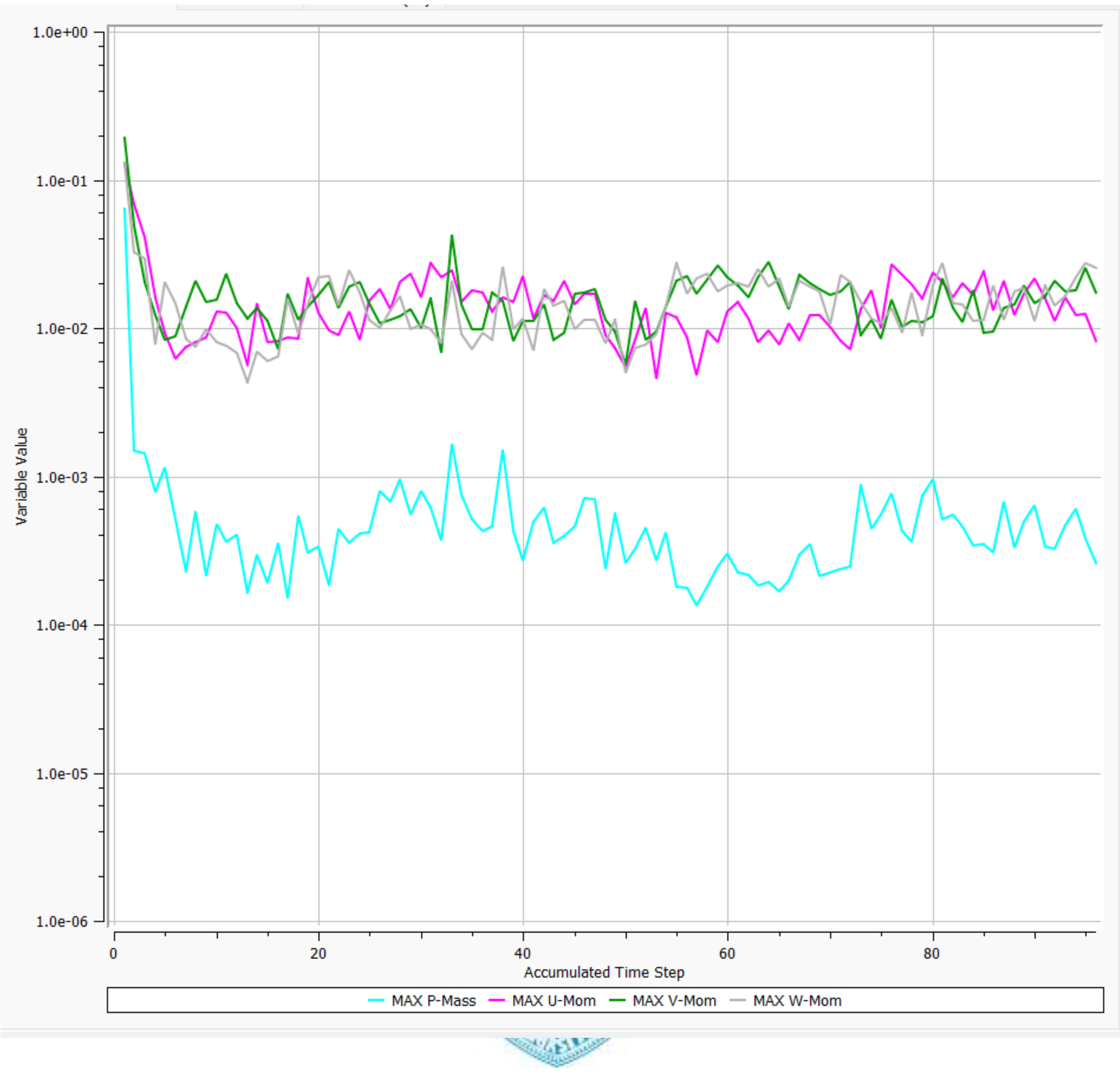

Figure 40: Failed convergence of momentum and mass

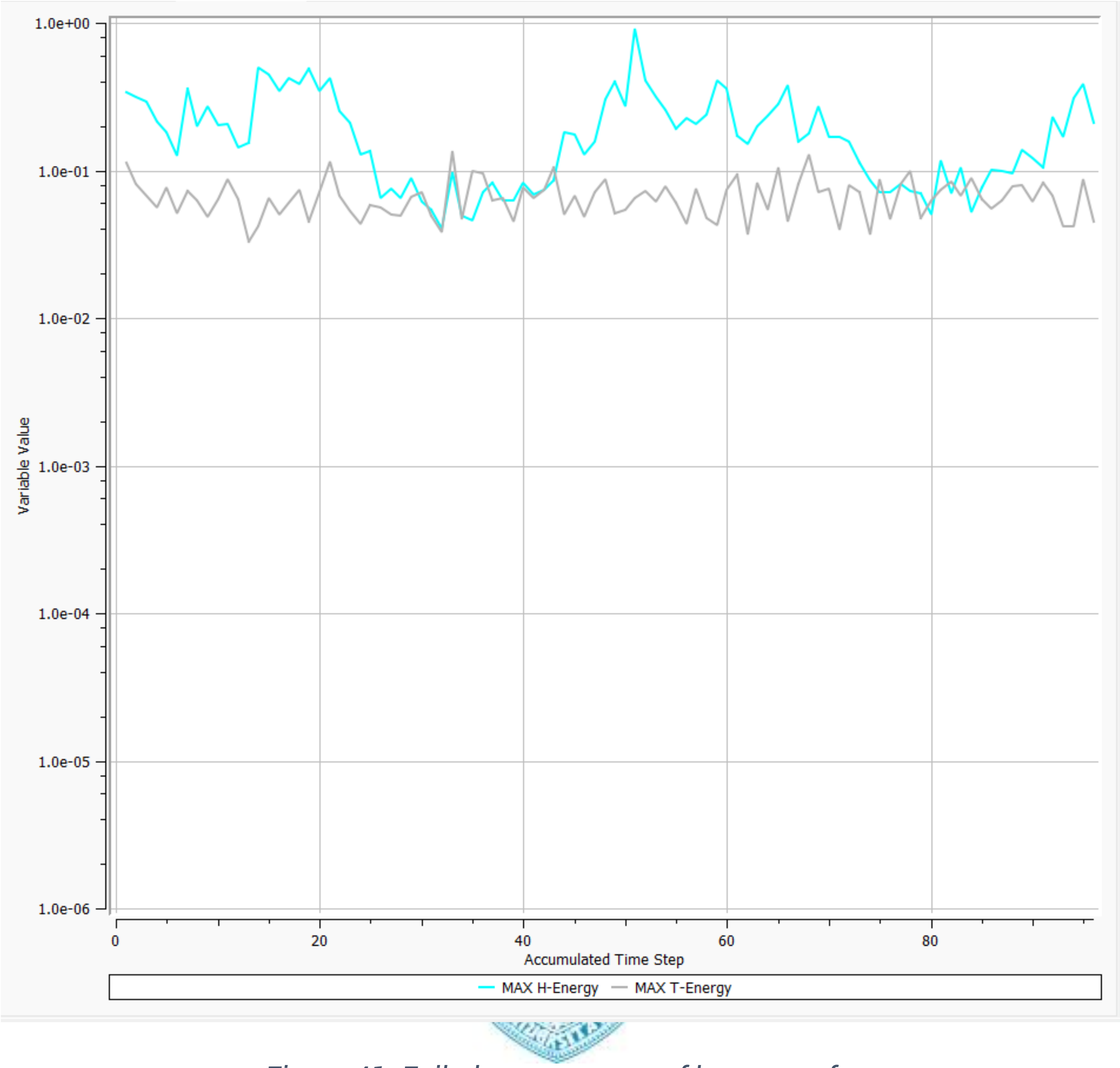

Figure 41: Failed convergence of heat transfer

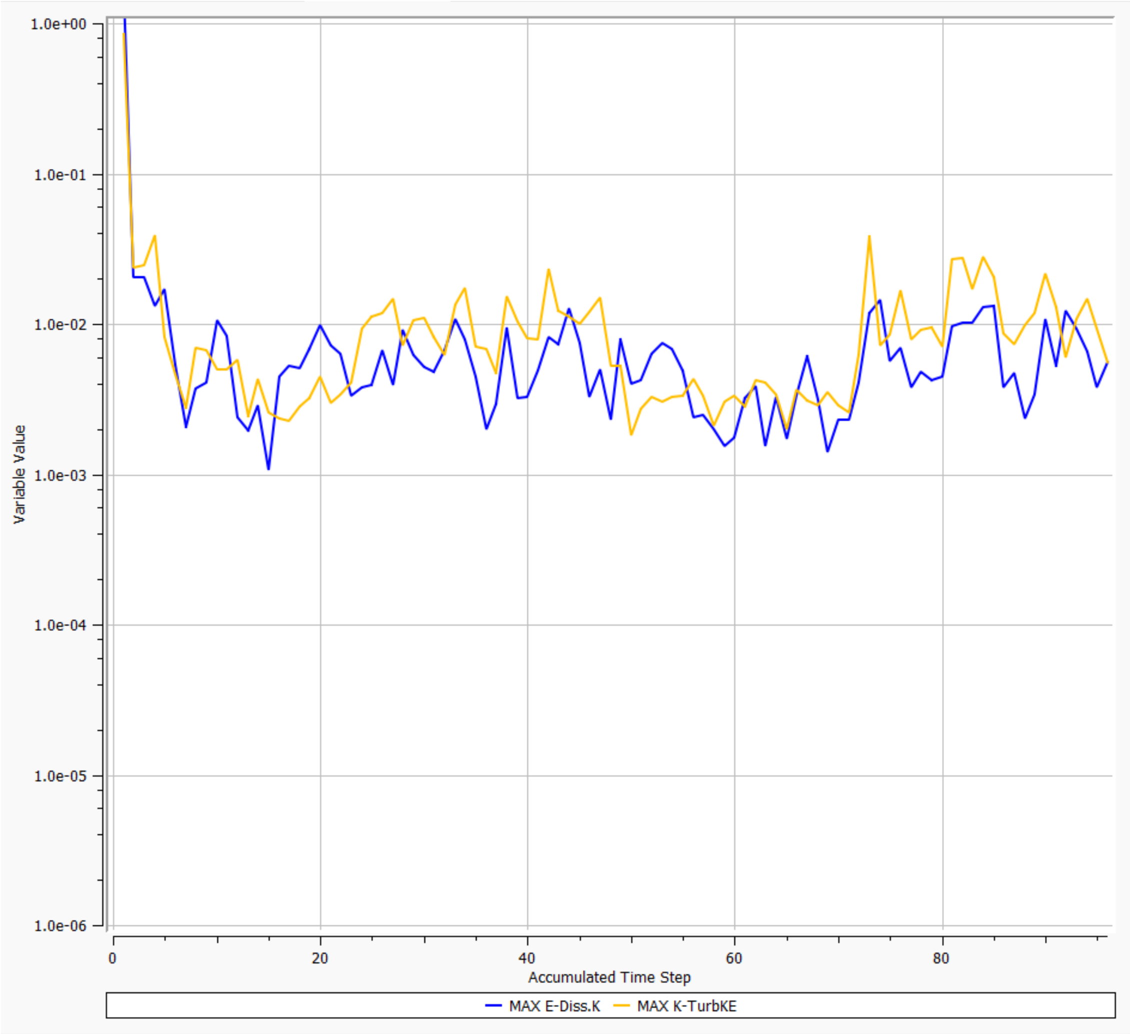

Figure 42: Failed convergence of turbulence

Analyzing the temperatures obtained at the thermostat, there are certain times when the temperature reaches high values far from the comfort temperature. To improve this, the inlet temperature needs to be lowered. The solution may involve increasing the size of the tube in the earth-air heat exchanger or installing another exchanger, operating both simultaneously, or increasing the diameter of the tube, thus increasing the contact area between air and soil.

4.9 Results for the summer day

In figure 43, the temperature distribution for the final moment of the simulation is observed.

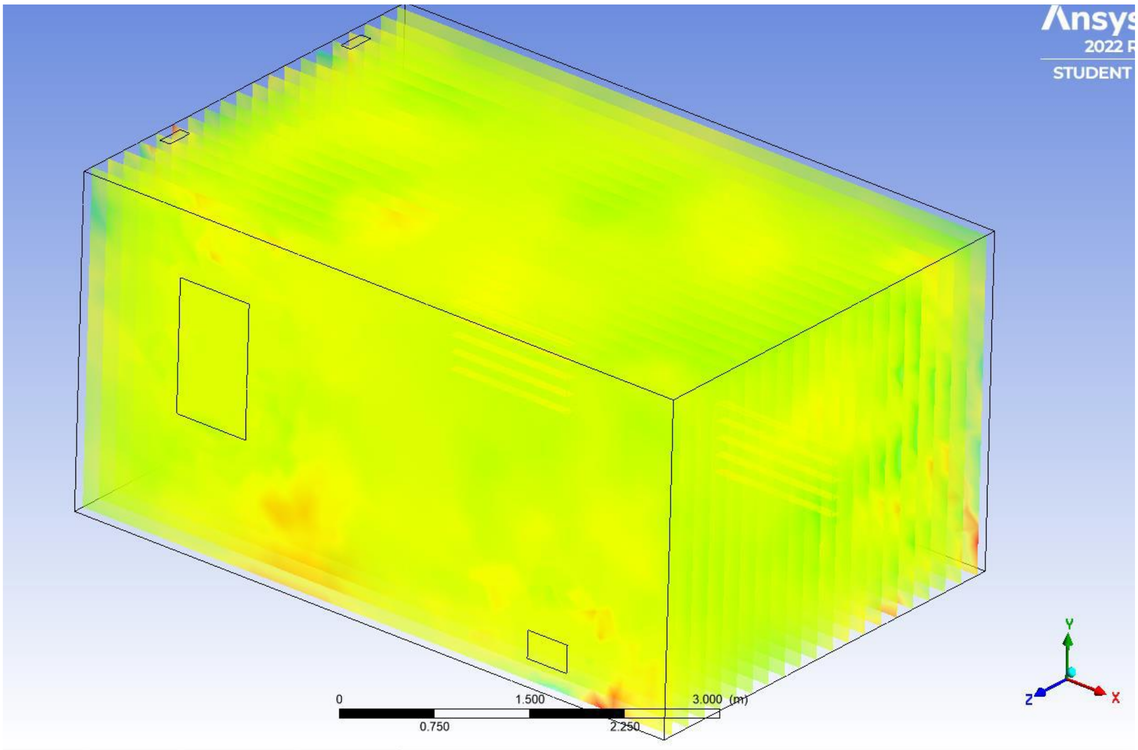

Figure 43: Temperature distribution at the final timestep

In the following figures 43 and 44, the airflow for the first timestep and the final timestep is observed. The decrease in flow lines at the final timestep may be due to lack of convergence in that step.

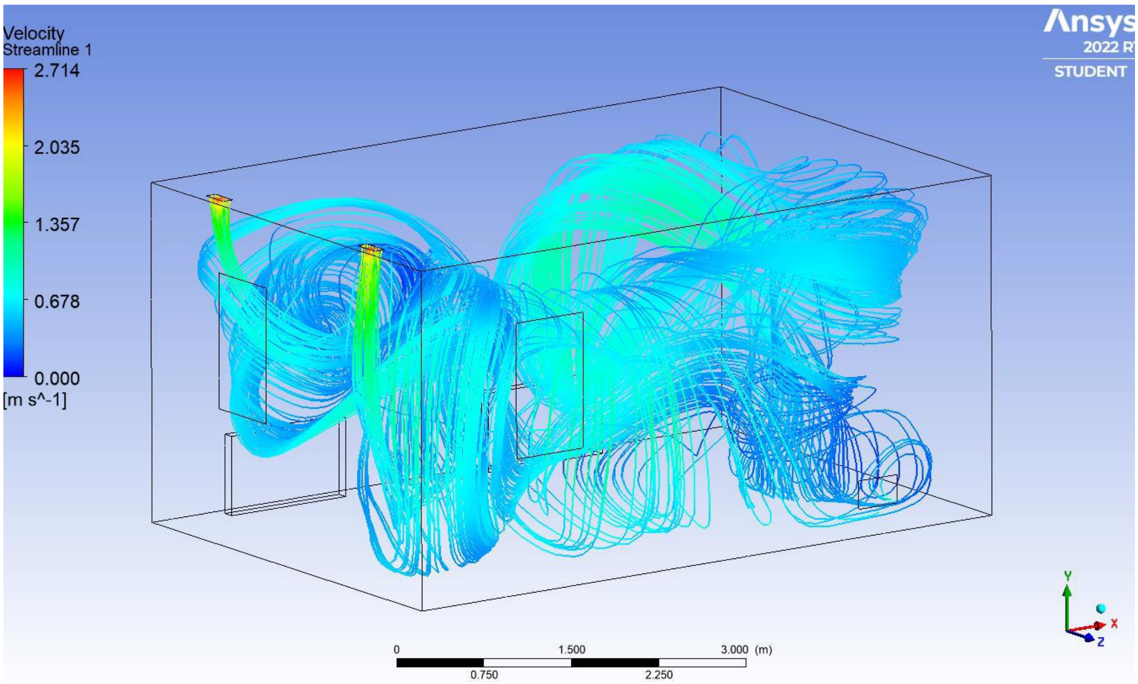

Figure 44: Flow at the first timestep

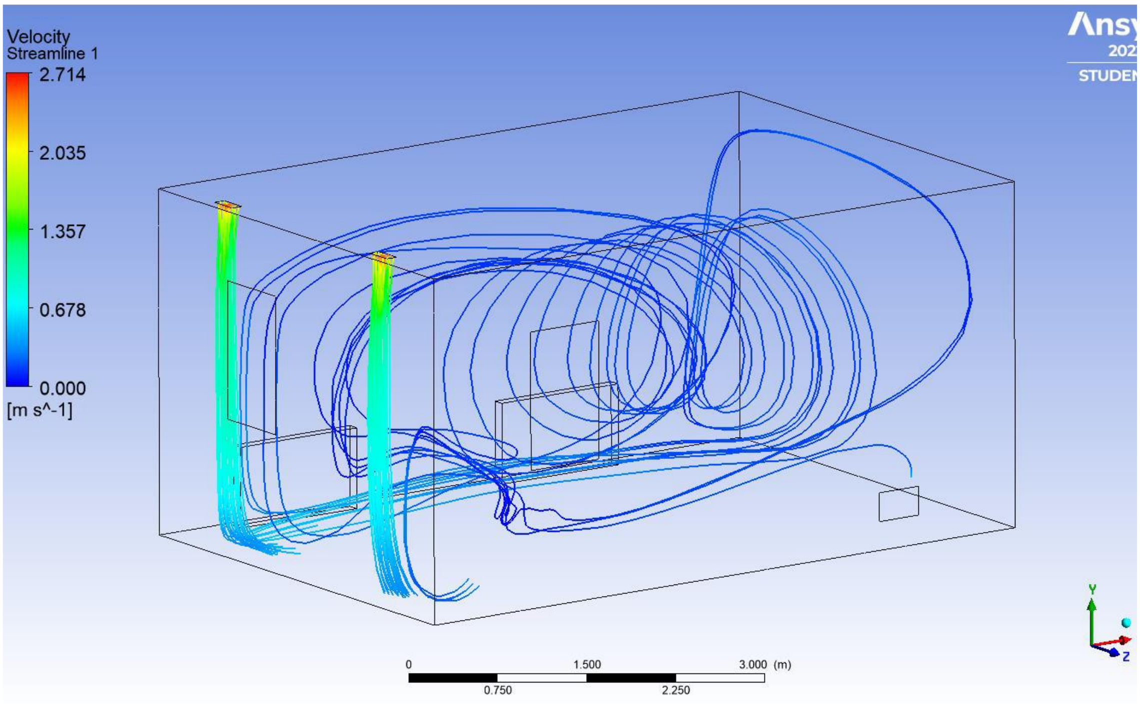

Figure 45: Flow at the final timestep