



*Study of the heat transmission of a liquid-solid-gas system for cooling electronic components of particle detectors.*

**FINAL REPORT ON THE INTEREST PROGRAMME**

JOINT INSTITUTE FOR NUCLEAR RESEARCH Veksler and Baldin laboratory of High Energy Physics

**STUDENT:** Urbietta Cruz Joseph

Universidad Nacional Autónoma de México, ICN.

*CDMX, 04510, México.*

**SUPERVISOR:** Dr Maribel Herrera Barrera

**PARTICIPATION PERIOD:**

February 26 – April 14, Wave 10 Dubna, 2024

## 1. INTRODUCTION

The importance of simulation in the design and operation of prototypes and detectors is paramount for accurately modeling complex phenomena such as heat transfer between different mediums, liquids, solids, and gases. Through simulation, we can understand how heat transfer influences the performance of the system under study. Additionally, simulation allows us to explore different design configurations and cooling strategies before implementing them physically, thereby saving time and resources.

The main problem of this work consists of simulating the cooling system of the Inner Tracking System (ITS) of the MPD detector using ANSYS software for subsequent analysis. The methodology developed for the work consisted of the following steps: 1. Installation of a CAD program (SolidWorks) to facilitate manipulation of the 3D models constituting the ITS system, 2. Installation of ANSYS software for analyzing the temperature of the models from the previous point, 3. Study of published documentation on the cooling system of the ITS (mainly the MPD-ITS Technical Design Report), 4. In-depth research on the operation of ANSYS software as it requires expertise to simulate what was studied in point 4, 5. Establishing the conditions to be considered in the software for simulation, 6. With the help of the CAD program, adjusting the 3D model to optimize interpretation and parameter construction in ANSYS, 7. Mesh modeling, 8. Establishing boundary conditions for simulation, 9. Simulation, 10. Obtaining results, and 11. Analyzing the results.

The complexity of the experimental environment and the need to maintain stable operating conditions have driven the application of advanced simulation tools like ANSYS to model the thermal behavior of the cooling system. The findings of this simulation will provide crucial information about the cooling system's performance, identifying areas for improvement and optimization that could lead to significant enhancements in the efficiency and accuracy of ITS measurements during heavy ion collision experiments.

Within the scope for the established timeframe of this work is having prepared the 3D model of the cooling system for importation into ANSYS software, establishing the parameters of the materials to be simulated, creating the meshing of the model, and setting the boundary conditions.

## **2. PROJECT OBJECTIVES**

The main objective of this work is to create a guide on using ANSYS software, specifically the Thermal or Fluent analysis modules, to simulate temperature and water (or air) flow conditions in the cooling system. This will be achieved through a detailed study of the Technical Design Report of the Inner Tracking System (ITS) detector of the MPD detector.

## **3. SCOPE OF WORK**

The scope of the report, as determined by the established timeframe, focuses on the following points:

- Introduction to the use of ANSYS Workbench Software: An overview of ANSYS Workbench software will be provided, highlighting its integration of multiple simulation modules. It will explain how the Thermal or Fluent analysis modules were selected to address the specific cooling system problem.
- Description of the model: The process of obtaining the 3D model of the cooling system, including the selection of corresponding carbon fiber sheets, pipes, and flow channels, will be detailed. It will explain how the geometries were configured in the ANSYS software.
- Meshing and establishment of boundary conditions: A detailed description of repairing the 3D model and meshing process to discretize the model into finite elements will be provided. Additionally, the established boundary conditions for simulating temperature and water (or air) flow conditions in the cooling system will be detailed.
- Limitations and next steps: The report will discuss study limitations, such as the lack of simulation and subsequent analysis due to time constraints. It will identify areas for improvement and potential next steps to complete the simulation, such as executing the simulation, analyzing results, and validating with experimental data or theoretical models.

## 4. Methods

Below are the steps followed during the simulation process with ANSYS Workbench.

### 4.1 Study of the Technical Design Report documentation of the Inner Tracking System (ITS) detector of the MPD detector.

Chapter three of the TDR document titled "Detector Layout" describes the materials, thermal conductivity, densities, and important information that constitutes the cooling system.

Screenshots of the tables and schematic from the Technical Design Report of the Inner Tracking System (ITS) detector of the MPD detector.

Table 1. highlights the material contributions

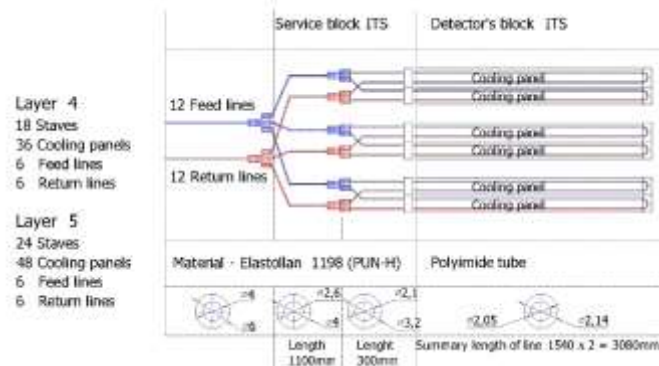
Stave element	Component	Material	Thickness (µm)	$N_0$ (cm)	$N_0$ (%)
Module	FPC Metal layers	Aluminium	50	8.896	0.056
	FPC Insulating layers	Polyimide	100	28.41	0.035
	Module plate	Carbon fibre	120	26.08	0.046
	Pixel Chip	Silicon	50	9.369	0.053
	Glue	Ecobond 45	100	44.37	0.023
Power Bus	Metal layers	Aluminium	200	8.896	0.225
	Insulating layers	Polyimide	200	28.41	0.070
	Glue	Ecobond 45	100	44.37	0.023
Cold Plate	Cooling tube wall	Carbon fibre	40	100.80	0.004
		Carbon paper	30	26.56	0.011
		Polyimide	44	28.41	0.013
	Cooling fluid	Water		35.76	0.105
	Carbon plate	Carbon fibre	120	26.08	0.046
	Glue	Ecobond 45	100	44.37	0.023
Space Frame		Carbon roving		0.080	
<b>Total</b>					<b>0.813</b>

Among the relevant information, Table 1.2 indicates the operating temperature of the detector chips.

Technical specifications	Detector Staves	RU Electronics
	OB: 84 pcs (Half Staves)	
Power load	From the detector 1555 W* Bus 63 W Power Cable 22 W	7.5 kW
Stave pressure difference and water consumption	OB: $\Delta P = 0.2 \text{ Kg/cm}^2$ $Q = 6.31 / \text{h}$ 52530 l/h	OB: $0.3 \text{ Kg/cm}^2$ Stave $Q = 111 / \text{h}$
Chip / valid temperature Range	20 °C to 30 °C	20 °C to 40 °C
Chip / working temperature	22 °C ± 1 °C	30 °C ± 5 °C

(\*) Calculation based on Alpha 4; OB = 28 mW/cm<sup>2</sup> + 50%.

Figure 1. illustrates a schematic of water circulation through polyamide tubes.



## 4.2 Adjustment of the 3D model of the cooling plate.

Despite the 3D model having real dimensions, it was decided to section it. The reason for this is that the subsequent steps to integrate the Mesh and the simulation require significant computational capacity. A desktop computer lacking sufficient resources is therefore forced to use an incomplete model to proceed. One of the particularities of ANSYS is its capability to import geometries created in other programs, expediting the creation of 3D models from programs such as SolidWorks.

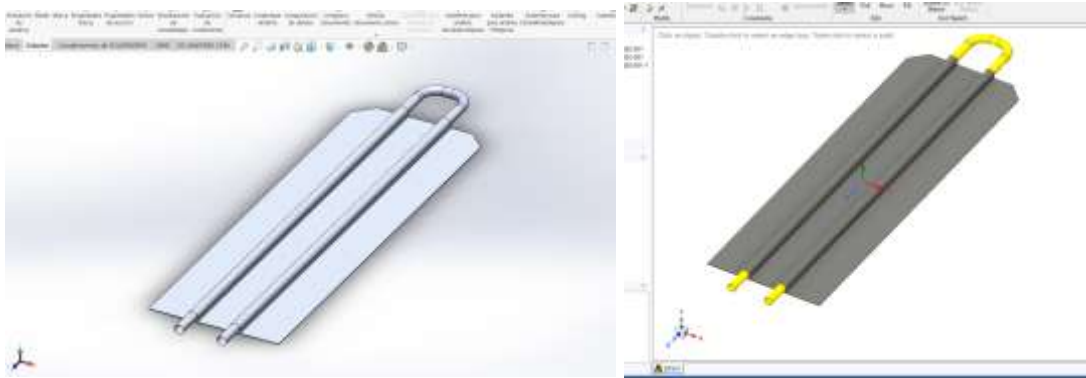


Figure 2 y 3. Section of the 3D model of the cooling system. Left: view from SolidWorks, Right: view from SpaceClaim.

## 4.3 Importing the model

To import the models, simply right-click on Geometry, select Import Geometry, and Browse... Then, choose the file in .STEP or .IGS format. See figure 3.



Figure 4. Section Materials.

## 4.4 Repairing the 3D model

An important step before creating the mesh in the model is to ensure that the imported model is repaired. This is done to avoid errors such as open geometries or surfaces that are difficult to handle.

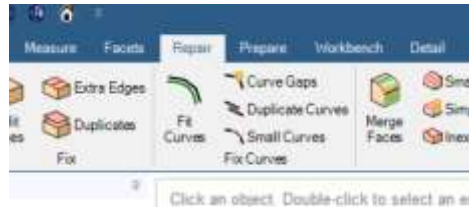


Figure 5. Repairing.

#### 4.5 Discretize model into a control volume

Through the Ansys Student software, the implementation of a control volume for the transfer of the coolant fluid was proposed. Its sole aim at present is to elucidate the behavior of various variables such as velocity, pressure, but specifically temperature, to comprehend the heat transfer from the plate to the carbon fiber coated with polyamide. It is crucial to discretize the heat transfer through conduction and convection in the case study.

Table 2. Specifications. Parameters to consider in the simulation

<b>Fluid: Water</b>	
<b><math>T_1 = 299.15K [K]</math></b>	<b><math>\mu = 0.001003 [Pa \cdot s]</math></b>
<b><math>q_0 = 20 [W/m^2]</math></b>	<b><math>C_p = 4182 [J/Kg \cdot K]</math></b>
<b><math>R = 0.1 [m]</math></b>	<b><math>\rho = 998.2 [Kg/m^3]</math></b>
<b><math>L = 13 [cm]</math></b>	<b><math>K = 0.6 [W/m^2 \cdot K]</math></b>
<b><math>Re = 500</math></b>	

It is proposed a very large  $z$ , where the gradient of  $T$  with respect to  $z$  tends to be constant, based on the equation:

$$T = T_1 + q_0 R \left[ \frac{2z}{\rho c_p R^2} + \frac{1}{k} \left( \frac{r}{R} \right)^2 - \frac{1}{4k} \left( \frac{r}{R} \right)^4 - \frac{7}{24} \right] \text{ ecuación (1)}$$

$$Nu = hD/k = 4.364$$

[1] Incropera, F. P., Sección 7.2, N.A., Introduction to Heat Transfer (Sexta Edición), Lienhard & Lienhard.

Through the Ansys Workbench software, the Fluid Flow module is utilized for the discretized modeling of a pipe section, which is imported from Solidworks and observed in Space Claim to confirm that the geometry is undamaged and solid repair is not required. The extruded control volume is observed, noting that fluid transport for cooling the plates will occur within this control volume. In the image, parameters such as diameter, area, and perimeter are visible in figure 6.

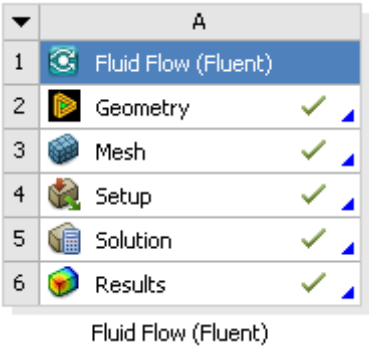


Figure 6. Operation Fluid Flow (Fluent)

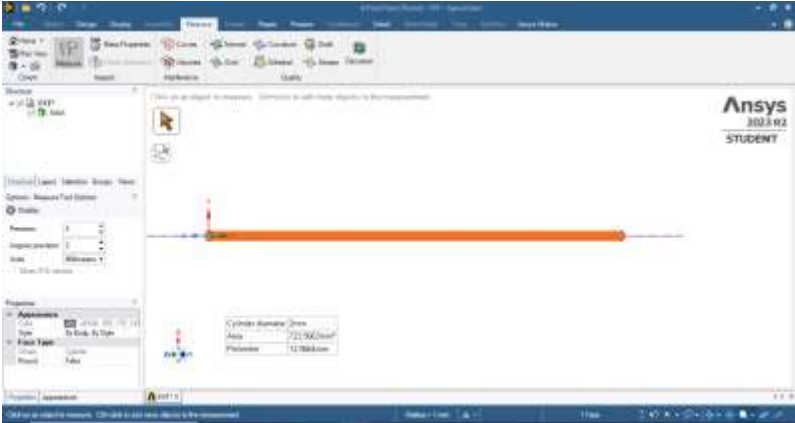


Figure 7. Discretized model.

After analyzing the geometry of the extruded volume, we turned to the Mesh module, where we can analyze the meshing of our geometry under study. It is important to mention that we require a small mesh sizing of around  $1.e-004$  in the body section. Additionally, Named Selection was performed to identify the boundary conditions that will be revisited later in the process, namely: inlet, outlet, and transport. It's worth mentioning that a Face Sizing of around  $1.e-005$  was applied at the inlet and outlet to enhance the solution behavior of the numerical method. Mesh independence was performed until convergence in the solution was achieved.

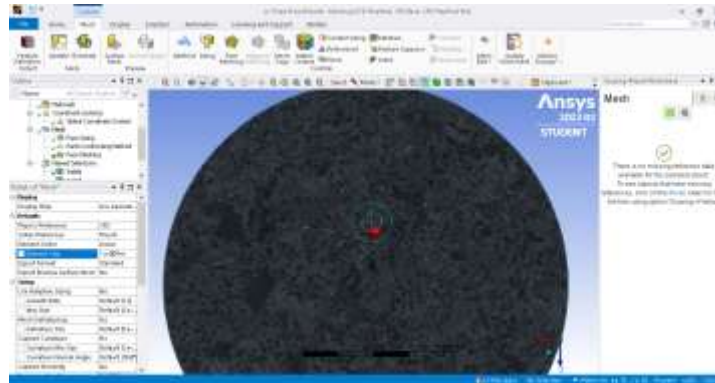


Figure 8. Mesh model.

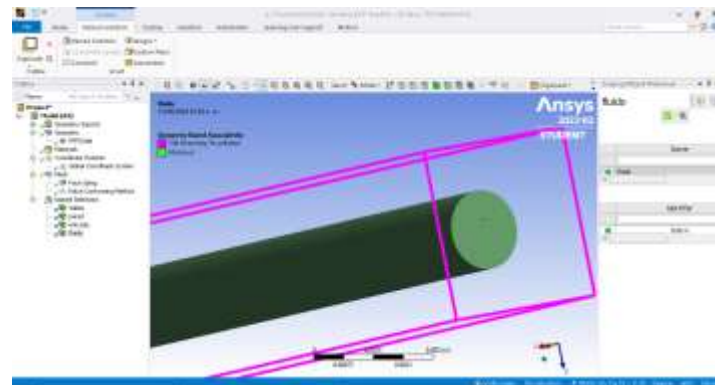


Figure 9. Named Selection, conditions.

In the Setup module, important parameters for the analysis are entered. It's worth mentioning that any erroneous changes at this point can completely alter the result.



Figure 10. Setup.

When opening the Fluent Launcher interface, the important conditions to consider in the study model are as follows:



- Specifications at this point: Stationary, neglecting gravity, assuming laminar flow, and considering the energy conservation equation.
- Fluid properties: Selecting properties for liquid water by adding their properties.
- Boundary conditions: Utilizing the Named Selections previously mentioned for:
  - Inlet
  - Transport
  - Outlet
- Flow behavior: Implementing developed flow with a parabolic velocity profile.
- Velocity magnitude: Utilizing equation (1) to determine velocity magnitude.
- Heat flux condition: Applying a constant heat flux condition provided by the wall.
- Initialization methods: Utilizing hybrid initialization methods.
- Iteration settings: Setting the number of iterations to 400 and the reporting interval to 1.

It's crucial to note that these parameters significantly influence the accuracy and reliability of the simulation results. Any deviations or errors in these settings can lead to significant alterations in the predicted behavior of the system under study.



Figure 11. Laminar fluent.

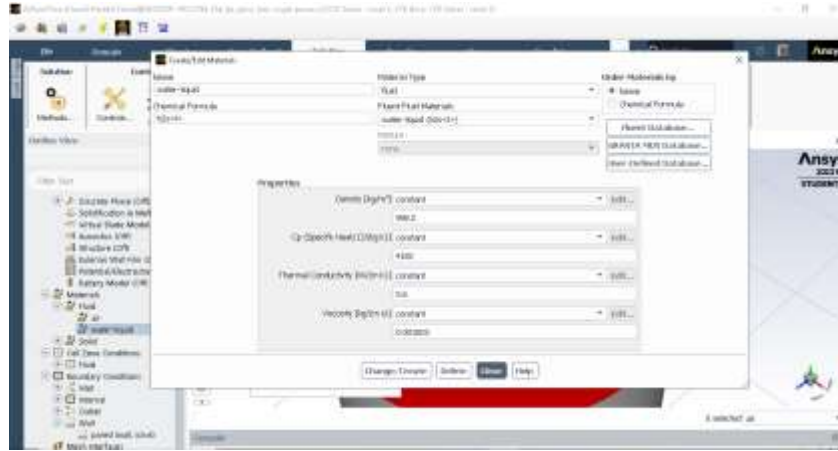


Figure 12. Properties liquid.

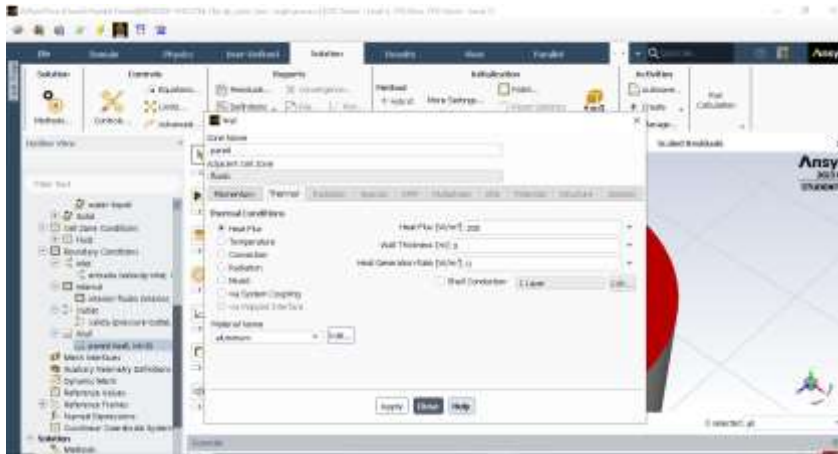


Figure 13. Heat flux system.

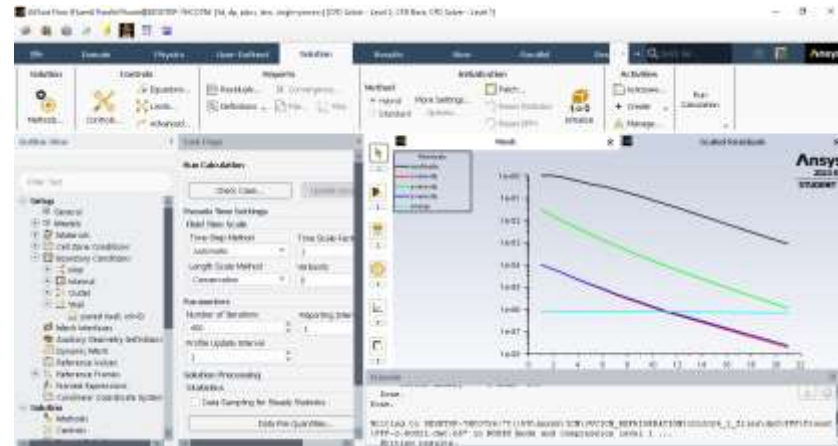


Figure 14. Solve model.

#### 4.6 Diagrams

In Figure 15, the behavior of the fluid and its interaction is observed, including the flow directions and heat transfer. There is a heat flow around the wall or perimeter of the pipe, which heats through convection. Additionally, the velocity profile in a pipe is depicted, showing a developed laminar velocity profile. The reference system is placed at the moment it passes through  $T_1$ , marking the onset of fluid heating. It's worth mentioning that this diagram is intended for educational purposes to aid understanding.

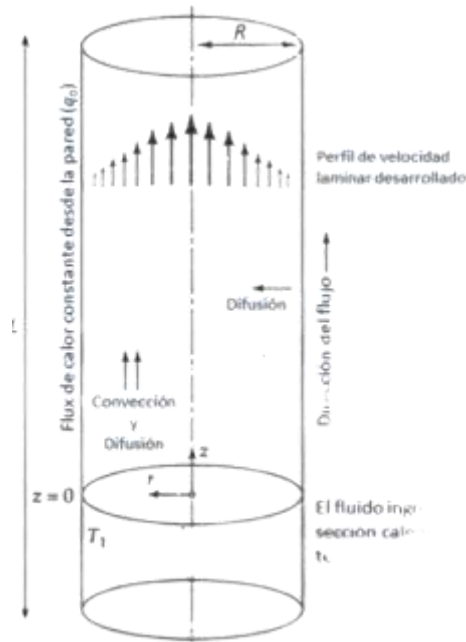


Figure 15. Model System.

[2] Sección 10.8, Bird et al. 2006 Fenómenos de transporte, 2 edición  
Bird, R. (2006), N.A, Transport Phenomena, Segunda Edición, LIMUSA

## 5 RESULTS

In figure 16 and 17, starting from the left side where our flow originates, it can be observed how the temperature of the flow begins at a low value, then increases to about  $17.85^{\circ}\text{C}$ . This confirms, to some extent, that there was a heat transfer from the fiber plate to our flow. This transfer occurred through a phenomenon known as conduction, where heat is transferred from a body of higher energy to one of lower energy. In conclusion, what is observed in the image confirms that the cooling system is functioning properly.

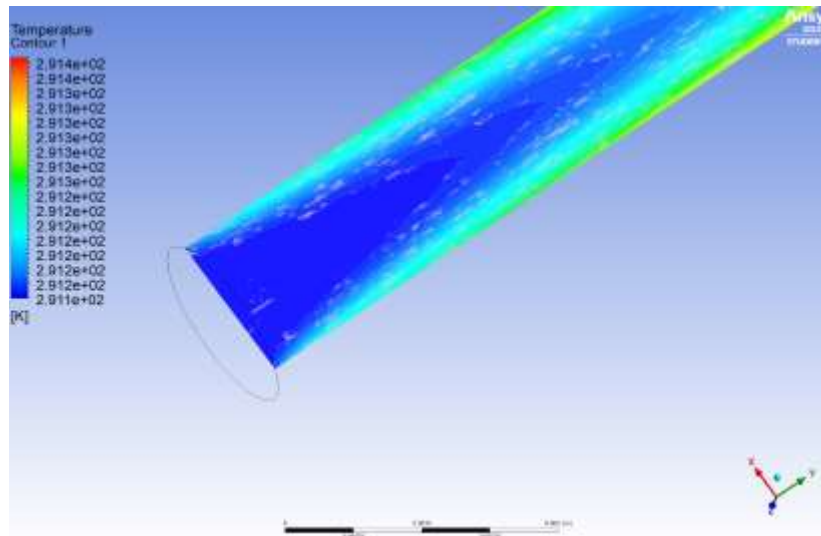


Figure 16. Solve temperature.

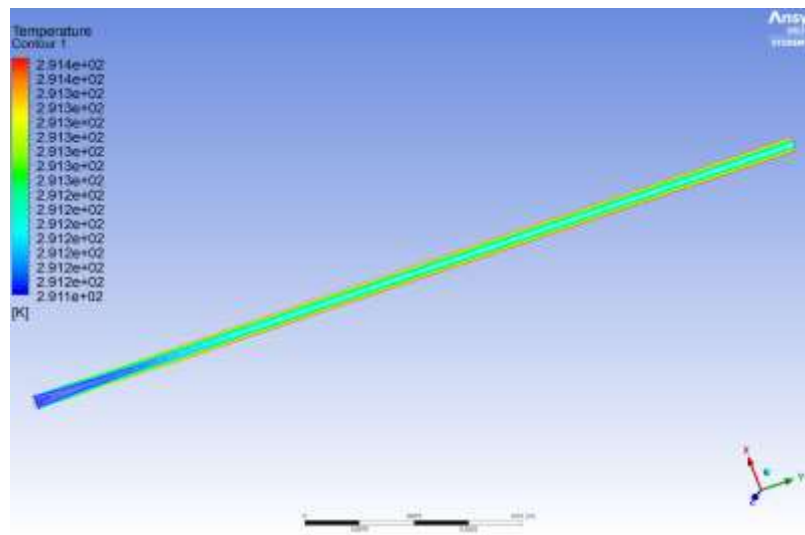


Figure 17. Solve temperature.

What can be observed in this image is how the flow velocity behaves. The most notable aspect in this image is that the velocity at the center of the tube is higher compared to the beginning and end of the tube. This is because the velocity profile in laminar flow tends to be parabolic, with the highest velocity at the center. This profile is a result of the conservation of momentum equation for steady laminar flow in a duct of constant cross-sectional area.

According to this equation, the fluid velocity at the center of the tube is maximum due to the lower resistance to flow in that region. As we move away from the center towards the walls of the tube, the fluid velocity decreases due to frictional effects with the walls, in figure 18 and 19.

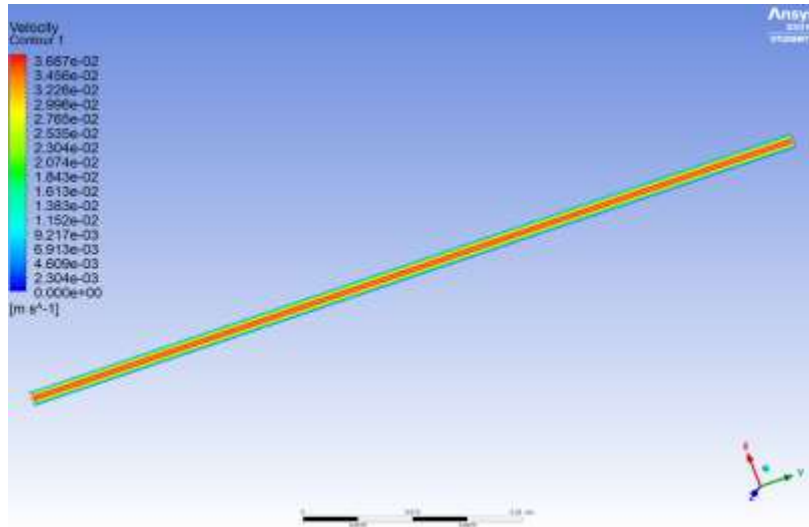


Figure 18. Solve Velocity.

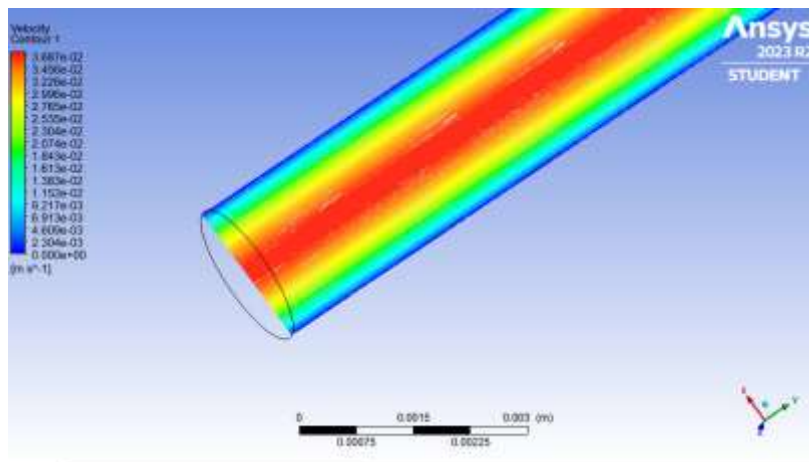


Figure 19. Solve Velocity.

## 6 CONCLUSION

Upon completing the research work, we were able to realize several things. Firstly, the overall objectives were successfully achieved as we managed to create a CAD model from which we conducted a simulation using the ANSYS program. Through this simulation, we were able to observe and better understand the behavior of a fluid inside one of the tubes of the carbon fiber piece.

In analyzing the simulation results, several key points were observed. Starting with temperature, we noted a moderate increase in the flow temperature, a phenomenon known as conduction. This explains the heat transfer between solid bodies, from one of higher energy to one of lower energy. Additionally, based on theoretical knowledge, we infer that a portion of the heat within the tube will transfer to the environment. This transfer occurs through convection, where temperature is transferred from a solid body to a fluid, in this case, air.

It's worth mentioning that the model was discretized into a selected control volume, as it is easier to work with that model. However, it is considered that a second iteration of conduction directly on the plate is required since we could compare the values and see the relationship between the entire system under study. While the result of this simulation is good, we cannot confirm it as the complete solution for the entire system.

## **7 REFERENCES**

[1] Incropera, F. P, Sección 7.2, N.A , Introduction to Heat Transfer (Sexta Edición), Lienhard & Lienhard.

[2] Sección 10.8, bird et al.2006 Fenómenos de transporte, 2 edición  
Bird, R.(2006), N.A, Transport Phenomena, Segunda Edición, LIMUSA