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

FINAL REPORT ON THE INTEREST PROGRAMME

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

Supervisor:

Dr Maribel Herrera Barrera

Student:

Godínez Herrera Tonatiuh Aktán

Mexico City

Universidad Nacional Autónoma de México

Participation period:

February 26 – April 14, Wave 10

Dubna, 2024

INTRODUCTION

Simulation plays a crucial role in designing and operating prototypes and detectors by accurately modeling complex phenomena such as heat transfer between different mediums. It helps us understand how heat transfer affects system performance and allows us to explore different design configurations and cooling strategies before physical implementation, saving time and resources.

This project focuses on simulating the cooling system of the Inner Tracking System (ITS) of the MPD detector using ANSYS software for analysis. The methodology involves several steps: installing a CAD program (SolidWorks) for 3D model manipulation, installing ANSYS for temperature analysis, studying documentation on the ITS cooling system, researching ANSYS operation, establishing simulation conditions, optimizing the 3D model, mesh modeling, setting boundary conditions, conducting simulation, and analyzing results.

The complex experimental environment and the need for stable operating conditions have led to the use of advanced simulation tools like ANSYS to model the cooling system's thermal behavior. The simulation results will provide crucial insights into the cooling system's performance, identifying areas for improvement and optimization that could enhance the efficiency and accuracy of ITS measurements during heavy ion collision experiments.

The project's timeframe includes preparing the 3D model of the cooling system for import into ANSYS, establishing material parameters, creating meshing, and setting boundary conditions.

PROJECT OBJECTIVES

The primary aim of this project is to create a guide for using ANSYS software, particularly the Thermal or Fluent analysis modules, to simulate temperature and fluid flow in a cooling system. This goal will be achieved by thoroughly examining the Technical Design Report of the Inner Tracking System (ITS) detector of the MPD detector.

SCOPE OF WORK

The report's scope, within the established timeframe, includes the following areas:

Introduction to ANSYS Workbench Software: Providing an overview of ANSYS Workbench software, emphasizing its integration of multiple simulation modules and explaining the selection of the Thermal or Fluent analysis modules for addressing the cooling system problem.

Description of the Model: Detailing the process of obtaining the 3D model of the cooling system, including selecting carbon fiber sheets, pipes, and flow channels, and configuring geometries in ANSYS software.

Meshing and Establishment of Boundary Conditions: Describing the repair of the 3D model, the meshing process to discretize the model into finite elements, and detailing the established boundary conditions for simulating temperature and fluid flow conditions.

Limitations and Next Steps: Discussing study limitations, such as the inability to complete simulation and subsequent analysis due to time constraints, and identifying areas for improvement and potential next steps to complete the simulation, such as executing the simulation, analyzing results, and validating with experimental or theoretical models.

Methods

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

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. Among the relevant information, Table 3.1 highlights the material contributions, Table 3.3 indicates the operating temperature of the detector chips, Table 3.4 describes the parameters of air circulation, and Figure 3.13 illustrates a schematic of water circulation through polyamide tubes.

Table 3.1: Estimated contributions of the Outer Layer Stave to the material budget.

Stave element	Component	Material	Thickness (μm)	X_0 (cm)	X_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	Eccobond 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	Eccobond 45	100	44.37	0.023
Cold Plate		Carbon fleece	40	106.80	0.004
		Carbon paper	30	26.56	0.011
	Cooling tube wall	Polyimide	64	28.41	0.013
	Cooling fluid	Water		35.76	0.105
	Carbon plate	Carbon fibre	120	26.08	0.046
	Glue	Eccobond 45	100	44.37	0.023
Space Frame		Carbon rowing			0.080
Total					0.813

Table 3.3: Technical requirements for Staves and electronics

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: ΔP=0.2 Kg/cm² Q=6.31/h Σ5301/h	OB: 0.3 Kg/cm² Stave Q=111/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

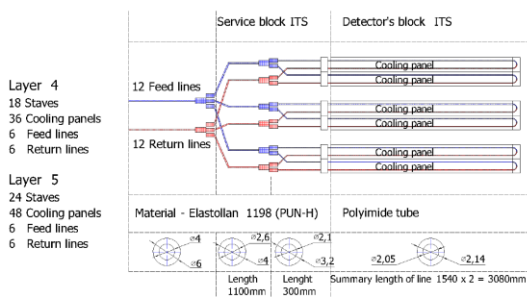(*)Calculation based on Alpide 4; OB = 28 mW/cm² + 50%.

Figure 3.13: Modularity specification of the ITS OB Layers.

Table 3.4: ITS air cooling sysms basic characteristics.

Parameter	Air circulation
Air flow	Q=30 m³/h±6 m³/h OB airflow: 20 m³/h Service unit: 7 m³/h
Temperature	T_{in}=20 °C
Humidity	RH_{out}=10 % to 35 % (RH to be set in this range)
Flow direction in the layers*	Layer 5 from A to C Layer 4 from C to A
Flow rate	<2 m/s (Detector housing)

(*)See figures 3.14 and 3.15.

Figure 1. Screenshots of the tables and schematic from the Technical Design Report of the Inner Tracking System (ITS) detector of the MPD detector.

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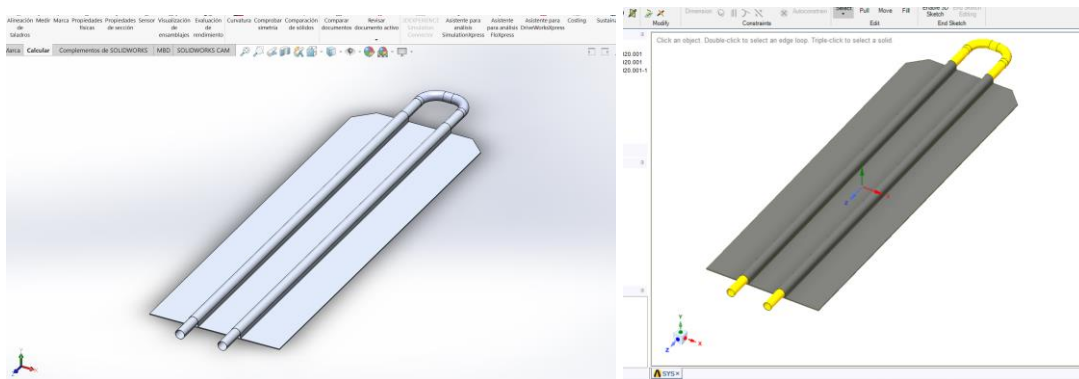
Figure 2. Section of the 3D model of the cooling system. Left: view from SolidWorks, Right: view from SpaceClaim.

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.
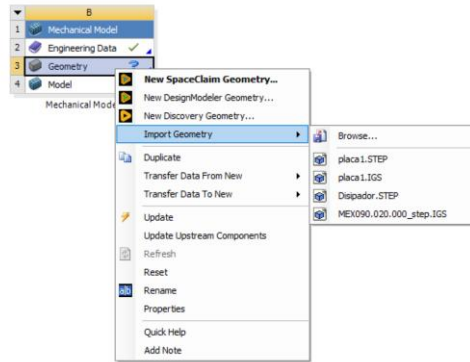
Figure 3. Section Materials.

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.
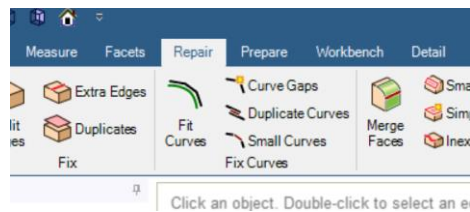

Figure 4. Repairing.

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.

Specifications:

Fluid: Water

T1: 299.15K [k]

q0= 20 [W/m^2]	μ=0.001003 [Pa s]
R= 0.1 [m]	Cp= 4182 [J/Kg K]
L= 13 [cm]	ρ= 998.2 [Kg/m^3]
Re= 500	K=0.6 [W/m^2 K]

Table 1. Data.

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

$$T = T1 + q0\,R\,[2z/\rho cpR^2 + 1/k(r/R)^2 - 1/4k(r/R)^4 - 7/24] \quad \textbf{ecuación (1)}$$

$$Nu = hD/k = 4.364$$

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.
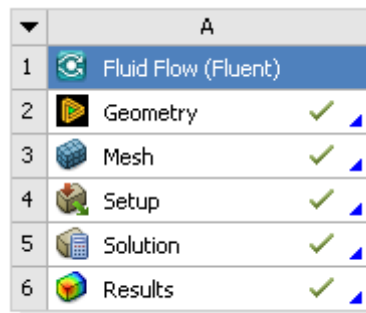
Fluid Flow (Fluent)

Figure 5. Operation Fluid Flow (Fluent)
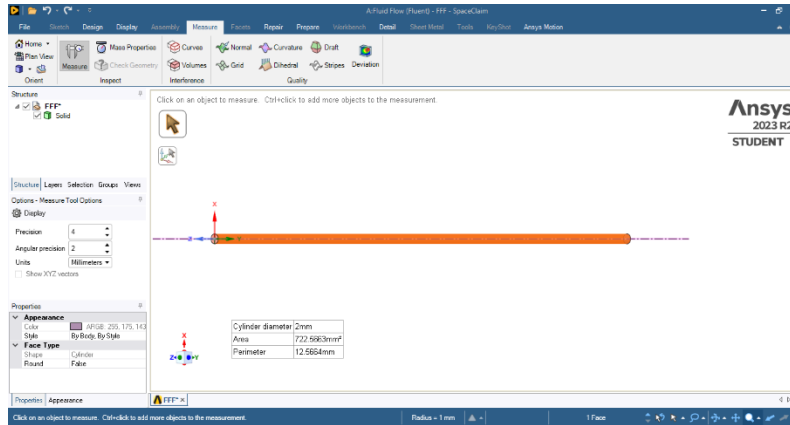
Figure 6. Discretized model.

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.
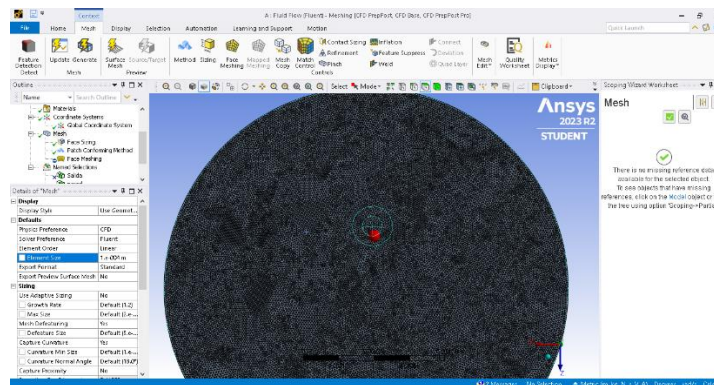
Figure 7. Mesh model.
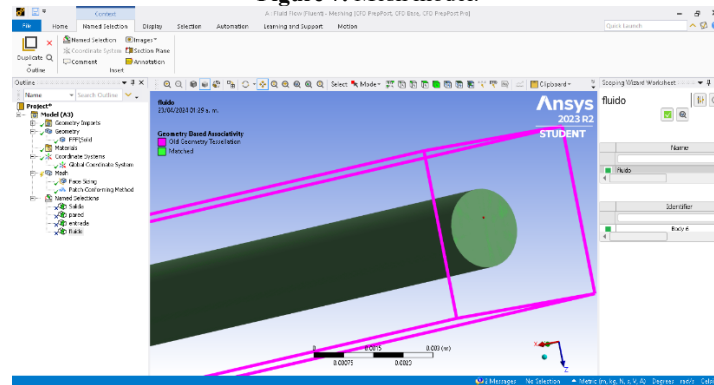
Figure 8. Named Selection, conditions.

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.
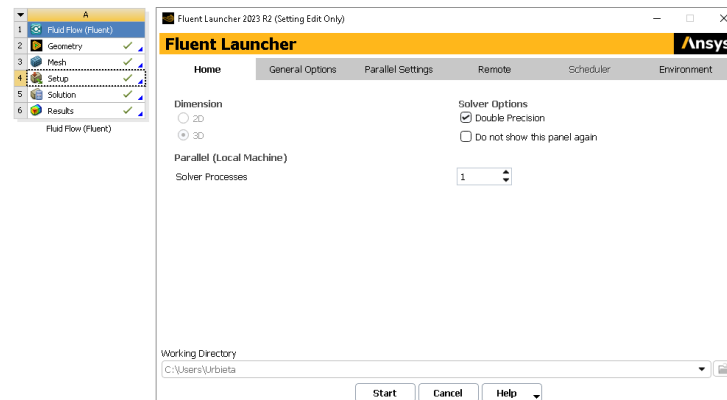

Figure 9. Setup.

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.
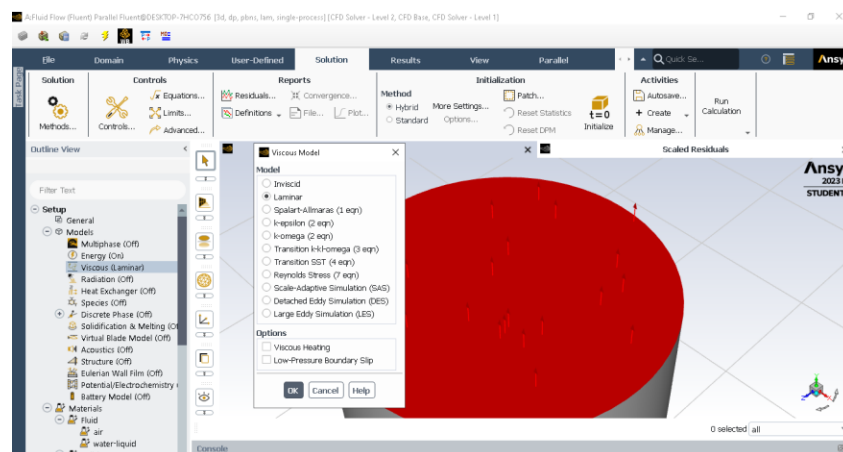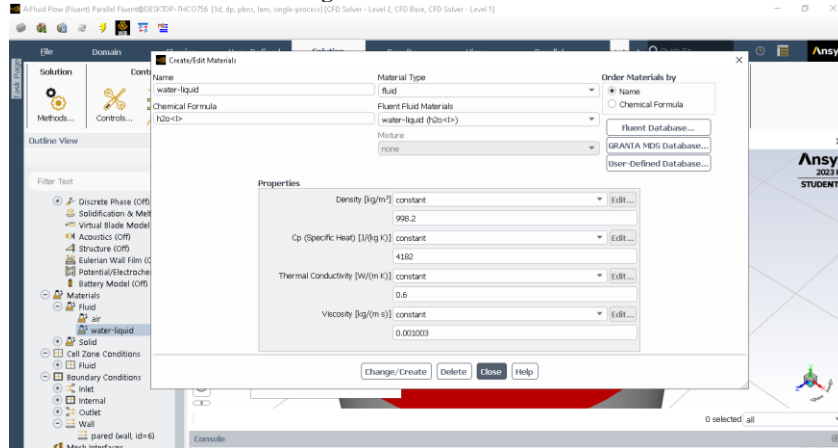
Figure 10. Laminar fluent.
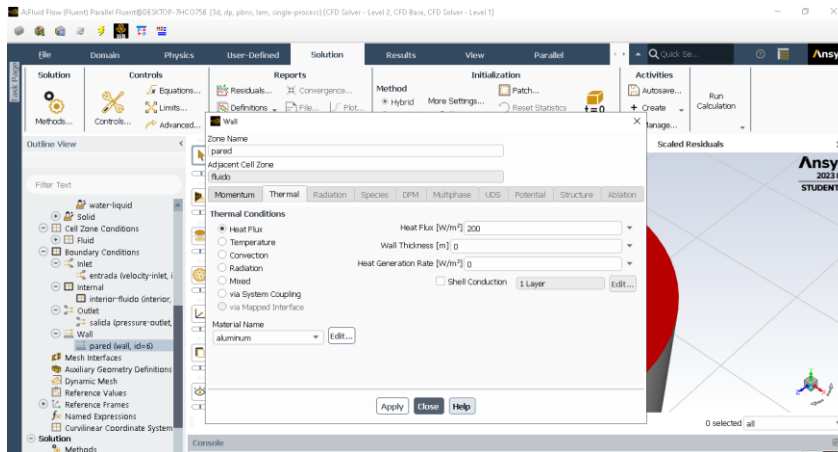
Figure 11. Properties liquid.
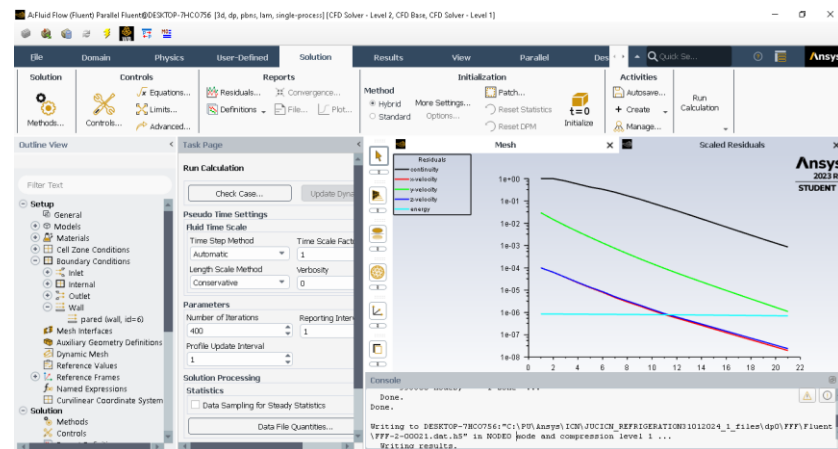
Figure 12. Heat flux system.
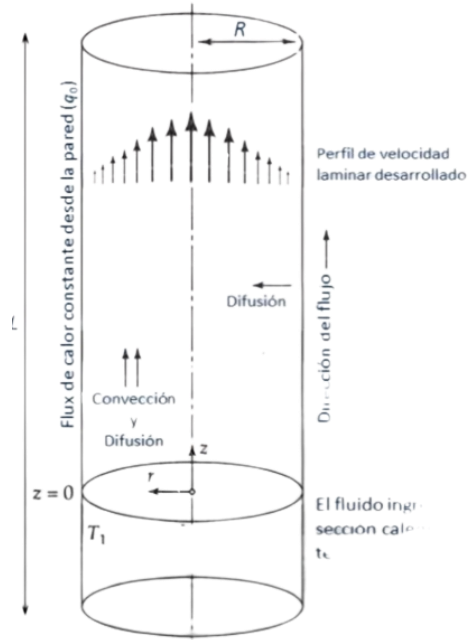
Figure 13. Solve model.

Diagrams

Diagram 1. Model System.

RESULTS

This image illustrates the flow from the left, showing the flow's temperature starting at a low value and increasing to approximately 17.85°C. This indicates some degree of heat transfer from the fiber plate to the flow, likely through conduction, where heat moves from higher to lower energy bodies. Overall, the image suggests that the cooling system is operating effectively.
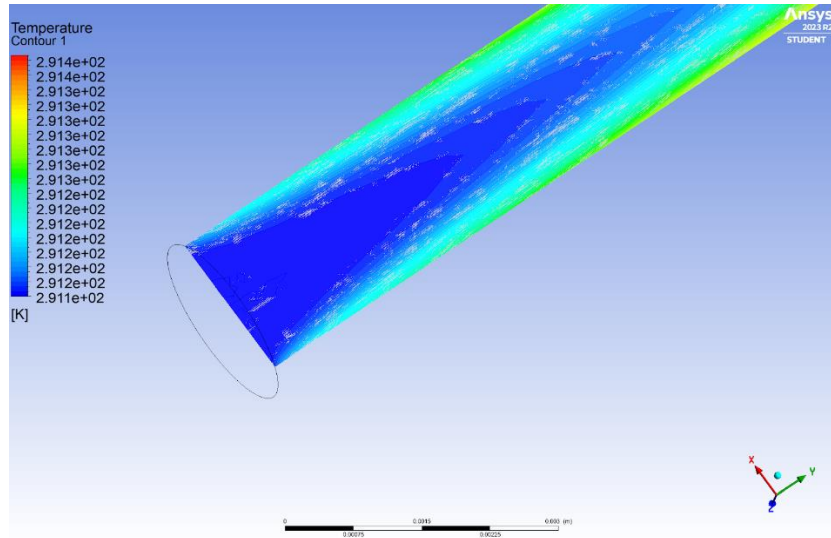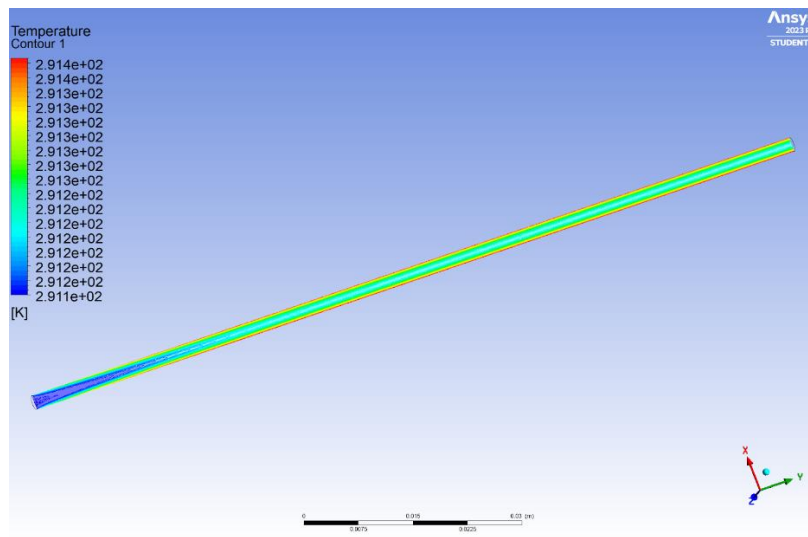
Figure 14. Solve temperature.

Figure 15. Solve temperature.

This image illustrates the behavior of flow velocity. The most noticeable aspect is that the velocity at the center of the tube is higher compared to the ends. This is because the velocity profile in laminar flow tends to be parabolic, with the highest velocity at the center, as dictated by the conservation of momentum equation for steady laminar flow in a duct of constant cross-sectional area.

According to this equation, the fluid velocity is highest at the center due to lower resistance to flow in that region. Moving away from the center towards the walls, the velocity decreases due to frictional effects with the walls.
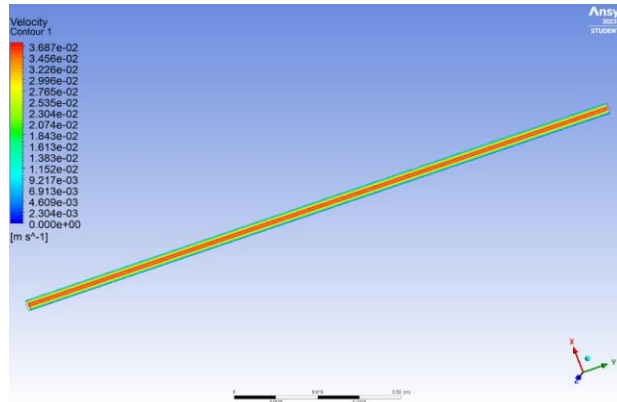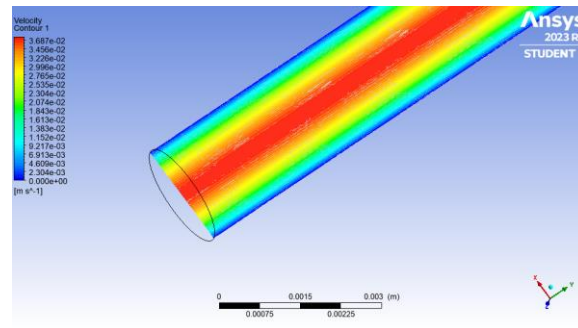
Figure 16. Solve Velocity.

Figure 17. Solve Velocity.

CONCLUSION

After completing the research, several outcomes were realized. The primary objectives were achieved by creating a CAD model for simulation in the ANSYS program, enabling observation and understanding of fluid behavior inside a tube of the carbon fiber component. Analysis of the simulation revealed several key findings. Regarding temperature, a moderate increase in flow temperature, known as conduction, was observed, indicating heat transfer between solid bodies from higher to lower energy. Theoretical knowledge suggests that some heat within the tube transfers to the environment through convection, where temperature transfers from a solid body to a fluid, such as air.

The model was discretized into a selected control volume for ease of use, but a second iteration of conduction directly on the plate is considered necessary for comparison and understanding of the entire system. While the simulation's results are promising, they do not constitute a complete solution for the entire system.

REFERENCES

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

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