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:
Alvarado Hernández, Mauricio
Mexico City
Universidad Nacional Autónoma de México

Participation period:
February 26 – April 14, Wave 10

Dubna, 2024

1. INTRODUCTION

Simulation plays a crucial role in both designing prototypes and operating detectors, particularly in accurately representing intricate processes like heat transfer across various mediums including liquids, solids, and gases. It helps in comprehending how heat transfer impacts the performance of the system being analyzed. Furthermore, simulation enables the exploration of diverse design setups and cooling techniques before their physical implementation, thus conserving 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) [1].
4. In-depth research on the operation of ANSYS software as it requires expertise to simulate what was studied in point 3.
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. Meshing is the necessary process to begin with any simulation option in ANSYS.
8. Establishing boundary conditions for simulation
9. Simulation, using the necessary conditions for the simulation to be as realistic as possible.
10. Obtaining results
11. Analyzing the results.

The intricate nature of the experimental setup and the necessity to uphold consistent operational parameters have prompted the utilization of sophisticated simulation software such as ANSYS to simulate the thermal dynamics of the cooling mechanism. The results from this simulation hold vital insights into the performance of the cooling system, pinpointing areas ripe for refinement and optimization. These enhancements could substantially elevate the efficiency and precision of ITS measurements during experiments involving heavy ion collisions.

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 GOALS

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 report's scope, defined by the established timeframe, centers on the following aspects:

- **Introduction to ANSYS Workbench Software Usage:** An overview of ANSYS Workbench software will be presented, emphasizing its integration of multiple simulation modules. It will elucidate the rationale behind selecting the Thermal or Fluent analysis modules to tackle the specific cooling system issue.

- **Model Description:** The process of creating the 3D model of the cooling system, including the selection of relevant carbon fiber sheets, pipes, and flow channels, will be outlined. It will elucidate how the geometries were configured within the ANSYS software.

- **Meshing and Boundary Condition Establishment:** A comprehensive explanation of repairing the 3D model and the meshing process to discretize the model into finite elements will be provided. Additionally, the defined boundary conditions for simulating temperature and fluid (water or air) flow conditions in the cooling system will be elaborated upon.

- **Limitations and Future Steps:** The report will address study constraints, such as the inability to conduct certain simulations and subsequent analyses due to time limitations. It will highlight areas for enhancement and potential future steps to complete the simulation, including executing the simulation, analyzing outcomes, and validating against experimental data or theoretical models.

4. Methods

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

The steps mentioned below are the result of several meetings among the members of the team involved in this project, as well as research on websites and tutorial videos regarding the use of ANSYS Workbench. It is important to highlight that appropriate equations for interpreting data and units used were also researched in specialized heat transfer books [2][3].

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. Figure 1 displays the most relevant information found, 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.

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
Power load	OB: 84 pcs (Half Staves) From the detector 1555 W* Bus 63 W Power Cable 22 W	7.5 kW
Stave pressure difference and water consumption	OB: $\Delta P = 0.2\,\mathrm{Kg/cm^2}$ $Q = 6.3\,\mathrm{l/h}$ $\Sigma 530\,\mathrm{l/h}$	OB: $0.3\,\mathrm{Kg/cm^2}$ Stave $Q = 11\,\mathrm{l/h}$
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 = $28\,\mathrm{mW/cm^2}$ + 50%.

Table 3.4: ITS air cooling sysms basic characteristics.

Parameter	Air circulation
Air flow	$Q = 30\,\mathrm{m^3/h} \pm 6\,\mathrm{m^3/h}$ OB airflow: $20\,\mathrm{m^3/h}$ Service unit: $7\,\mathrm{m^3/h}$
Temperature	$T_{in} = 20\,°\mathrm{C}$
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	$<2\,\mathrm{m/s}$ (Detector housing)

(*)See figures 3.14 and 3.15.

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

Figure 2 show a schematic of the flow planned for the ITS detector. The significance of this diagram lies in its ability to describe the behavior and direction of water flow in the simulation. ANSYS Workbench enables working with both gas and liquid flows, making simulation using this software acceptable.

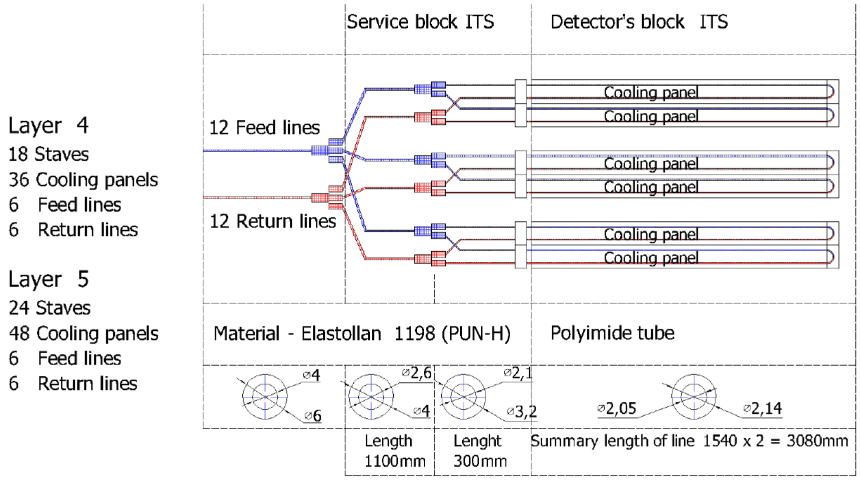

Figure 2. Screenshots of diagram of the water circulation through the cooling system from the Technical Design Report of the Inner Tracking System (ITS) detector of the MPD detector.

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 simulation require significant computational capacity. Figure 3 shows a section of the complete model, with an additional U-shaped component added to serve as the water flow return. A desktop computer, lacking sufficient resources, is forced to use an incomplete model to proceed. One of the particularities of ANSYS is its capability to import geometries created from other programs, which streamlines the creation of 3D models from programs like SolidWorks.

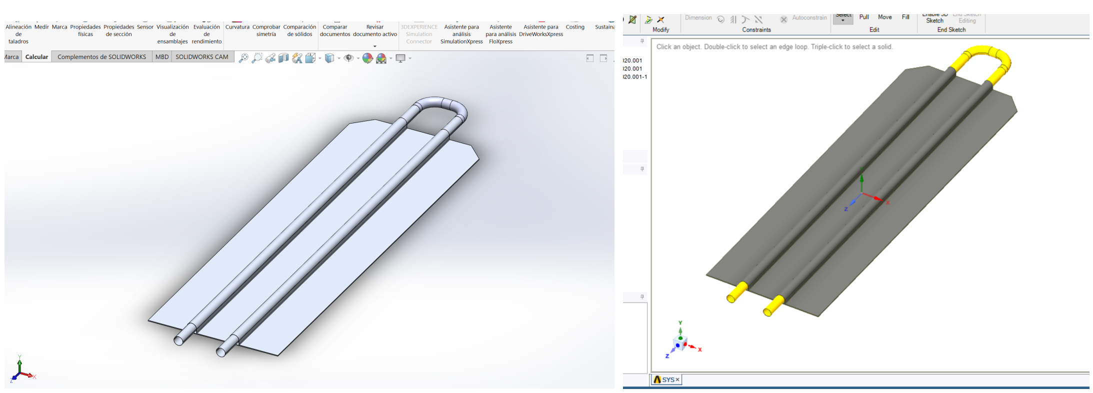

Figure 3. Section of the 3D model of the cooling system. Left: SolidWorks view, right: SpaceClaim view.

Importing the model

Importing 3D models into ANSYS is crucial for analysis and simulation. It serves as an alternative to using SpaceClaim since many designers prefer using other programs and sharing their files in more compatible formats. By importing 3D models, users can conduct finite element analysis (FEA), computational fluid dynamics (CFD), electromagnetics, and other simulations.

Figure 4 show the steps to import the models, simply right-click on Geometry, select Import Geometry, and then Browse… Choose the file in .STEP or .IGS format.

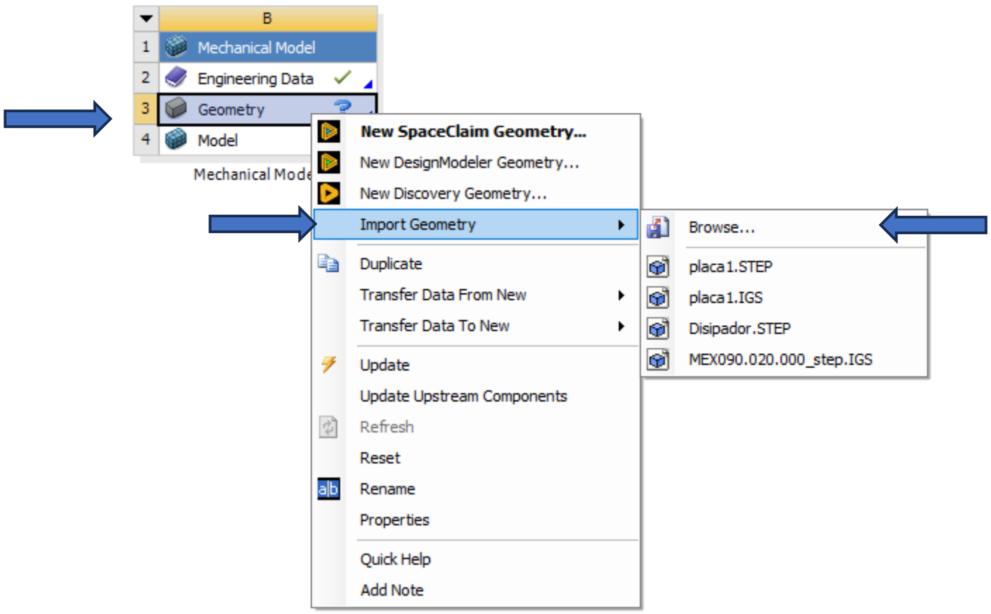

Figure 4. Geometry import menu.

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. Repairing a 3D model in ANSYS is crucial because it ensures the accuracy and integrity of the analysis. 3D models may contain imperfections such as poorly defined geometries, holes, unexpected intersections, or poorly connected surfaces, which can lead to inaccurate results or errors in simulations. By repairing the 3D model, these issues are corrected to ensure that the geometry is suitable for numerical analysis.

In Figure 5, the right side displays the tab where the option to repair the 3D model is selected. On the left side, the imported model is shown, highlighting an assembly error between parts. Repairing parts helps identify potential assembly issues.

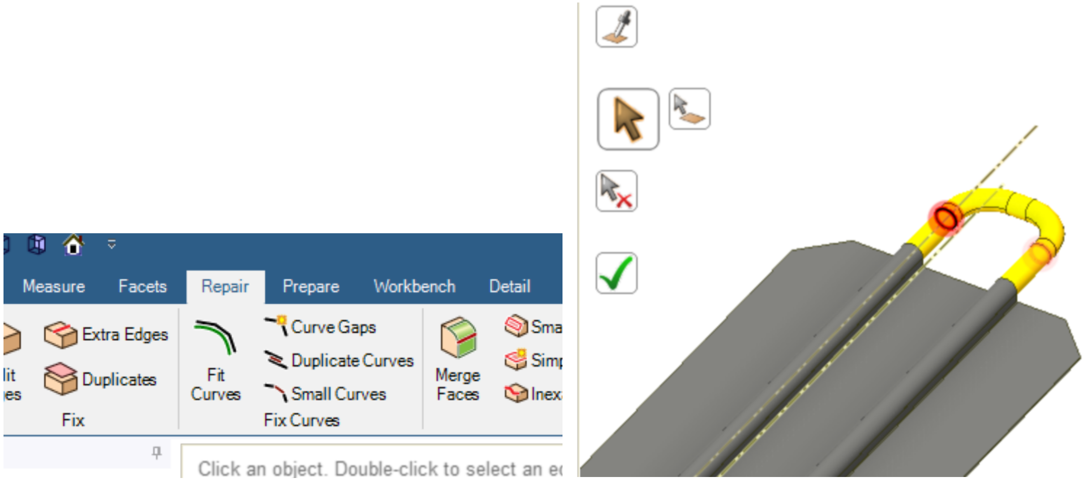

Figure 5. Sample of junctions that need to be repaired. On the left side, the repair section is displayed, while on the right side, a view of an error in the model due to part junction is shown.

Editing materials

Each element of the 3D model can be assigned a type of material with the thermal conductivity characteristics described in the TDR of the ITS-MPD. As seen in the following figure 6, right-click on Engineering Data, then select Edit.

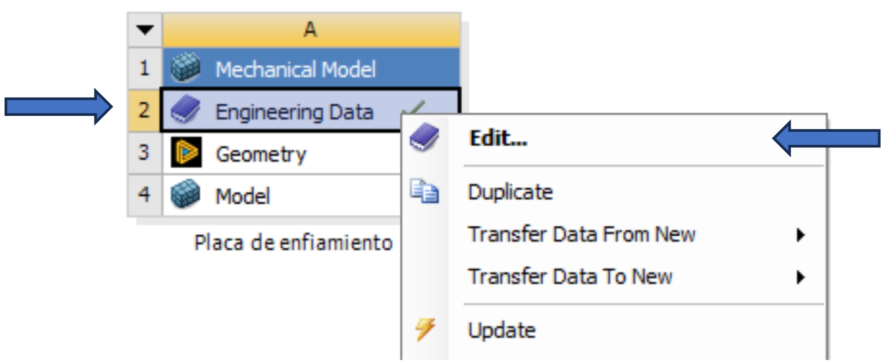

Figure 6. Menu where the material list can be edited.

Adding and editing material properties in ANSYS is essential for conducting accurate and realistic analyses. Each material possesses unique characteristics that influence its behavior under various loading and environmental conditions. By adding and editing material properties in ANSYS, the behavior of components and structures can be simulated more accurately.

A list of available materials will be displayed; however, a new material with the desired characteristics can be created. Figure 7 displays the list of available materials.

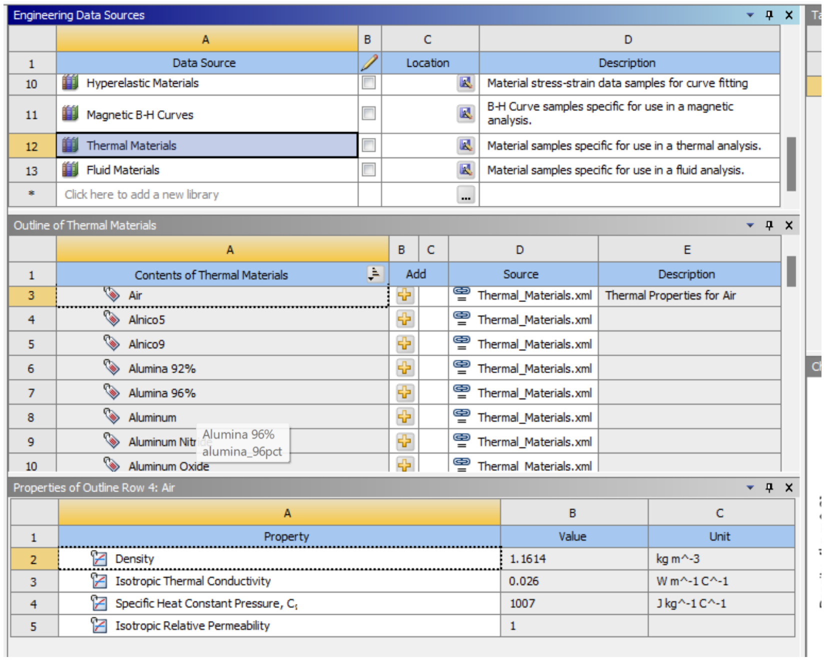

Figure 7. List of available materials in ANSYS.

To associate the material with the element of the 3D model, simply right-click on the model and then select Edit (figure 8). When the workspace with the model opens, select the piece, and from the list of materials, choose the appropriate material (figure 9).

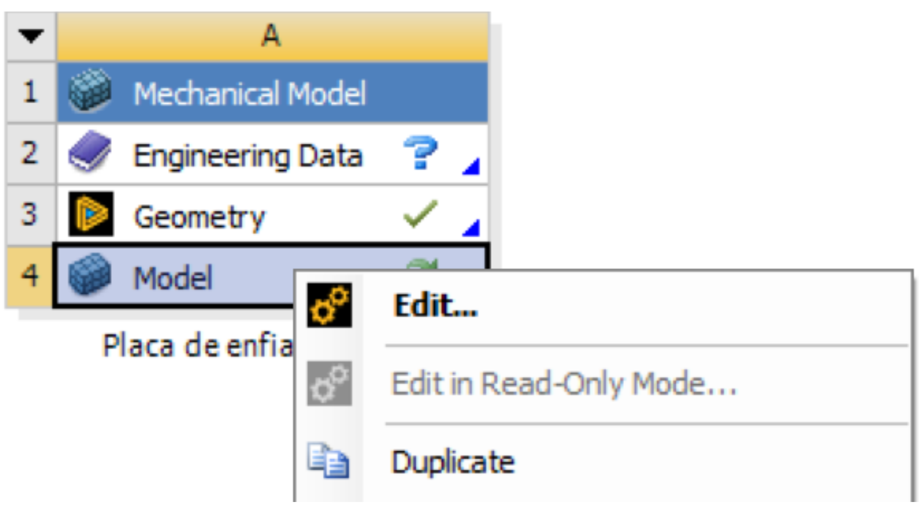

Figure 8. Menu for editing the 3D model.

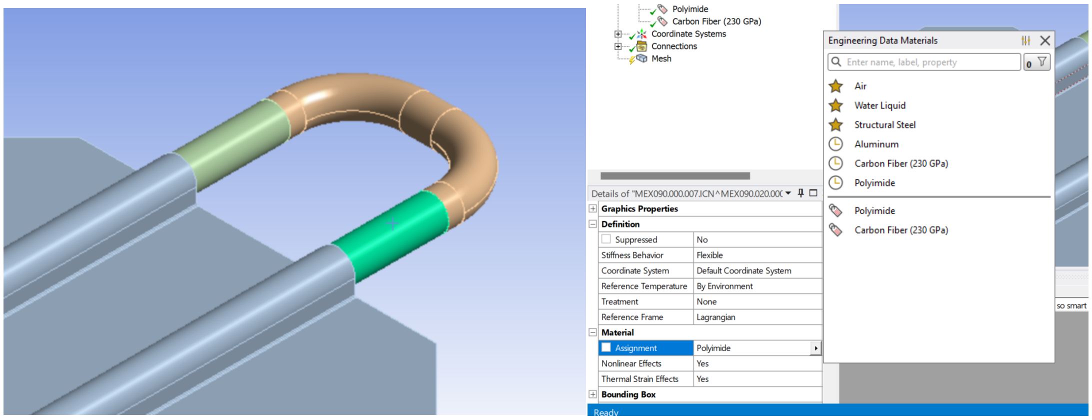

Figure 9. Material assignment to a 3D model element. On the left side, the selection of parts of the 3D model is displayed, while on the right side, the appropriate material is selected.

Creating Mesh

Once the 3D model is prepared, meshing is carried out. To do this, go to the menu on the left side labeled Mesh, then right-click to display another menu, where Insert is selected, followed by sizing. Using the cursor, choose the faces of the 3D model that you want to work with. It is important to choose an appropriate number for meshing; generally, 1e-005 is sufficient to avoid generating errors (as figure 10).

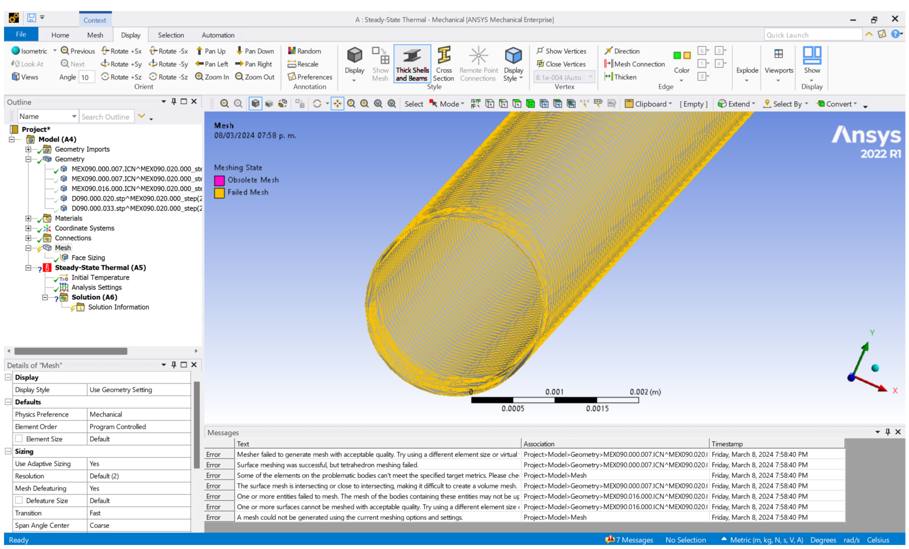

Figure 10. Example of mesh in a 3D model.

Extracting the volume of a geometric body

By extracting a volume, one can isolate parts of the geometry for finite element analysis, specifically in this case for computational fluid dynamics. This process is useful for studying the behavior of individual components and evaluating critical flow areas or simplifying a complex model to reduce calculation time.

The geometry must be open in SpaceClaim, then you proceed to the Prepare tab. Select "Volume Extract" as show in figure 11.

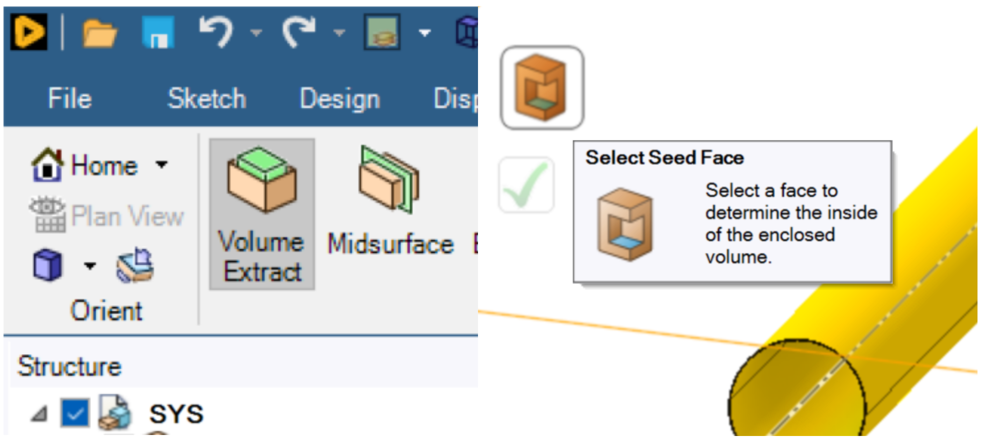

Figure 11. Volume extraction in the 3D model. On the left side, the option that needs to be selected is shown, while on the right side, the option for the internal face is displayed.

4 RESULTS

The expected results from simulations typically involve the model changing colors on a scale ranging from blue to red, with red indicating the highest impact. Through simulation, ANSYS Thermal provides valuable data on temperature distributions, heat flows, and thermal gradients within the model (figure12).

Despite only conducting a simple simulation representing temperature behavior in a very basic model, it was possible to obtain a representation of temperature behavior in a system controlled by simulation parameters. This is evident from the different color zones observed in the 3D model.

ANSYS Thermal allows for the evaluation of different cooling strategies as the behavior reflected in the model is indicative of material management.

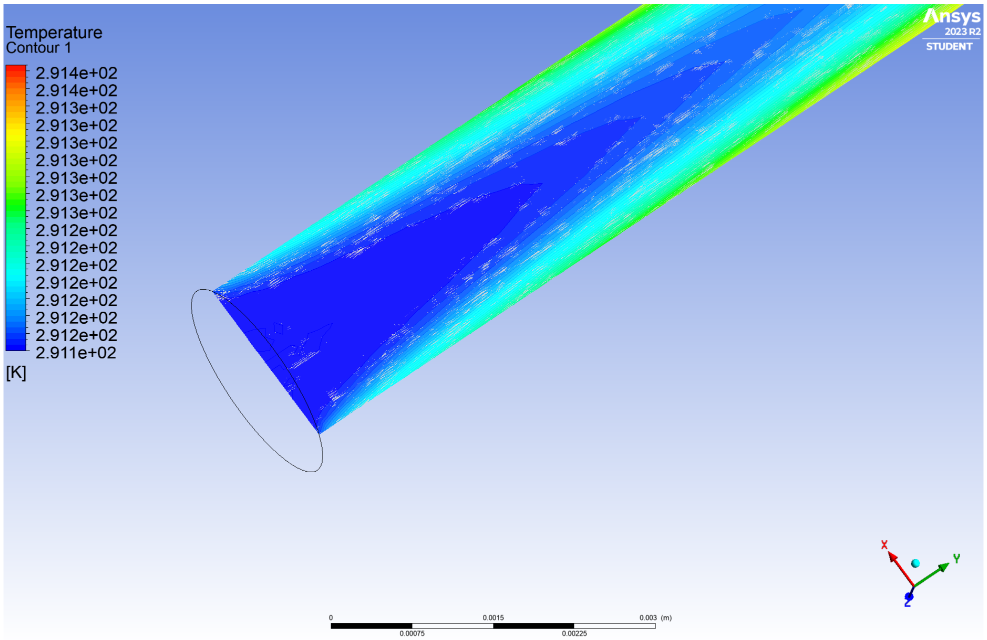

Figure 12. Example of the results yielded by the simulation.

5 CONCLUSION

Simulation is an essential tool in the design and prototyping processes, saving resources and time in projects. In this work, we undertook the task of learning to use the ANSYS software to conduct simulations that would help verify that the cooling system of the ITS-MPD detector functions as expected. Although we couldn't achieve that goal due to the limited time we had, as ANSYS software is complex, we were able to take the first steps in importing, simulating, and obtaining results from simple models.

Not only did we create a representative model in form, but we also managed to establish the parameters of specific materials that we learned from the documentation of the TDR of the ITS detector. This may seem minor, but in any finite element simulation software, parameters are indispensable in simulation behavior.

The results obtained may be preliminary and need to be discussed further, even requiring more detailed interpretation. From this learning experience, even more can be gleaned with the help of this software, as it allows for the analysis of models by combining temperature exchange criteria between materials and fluids. Unfortunately, the time we had was short, but there is no doubt that detailed simulation results can be achieved in the future.

6 REFERENCES

[1] MPD-ITS Technical Design Report, February 2022.

[2] F. P. Incropera y D. P. DeWitt, *Fundamentals of Heat and Mass Transfer*. Wiley Sons, Inc., John, 2000.

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