Environmental Monitoring in the ATLAS ITk Detector

M. P. Connell[1], M. Bhamjee[1], T. P. Mafa[2], S. H. Connell[1], L. Leeuw[2]

1. Department of Mechanical Engineering Science, University of Johannesburg, Auckland Park, Johannesburg, South Africa. 2. Department of mathematical Sciences and UNISA - ATLAS High Energy Physics Group, University of South Africa, Johannesburg, 1710, South Africa 2. Department of Physics and Astronomy, University of the Western Cape. Belville 7535, Cape Town. South Africa

E-mail: rsamcon@gmail.com

Abstract. In the design of the proposed Inner Tracker upgrade to the ATLAS detector, humidity and temperature sensors will be placed throughout the volume to monitor environmental conditions. To assist in the placement of these sensors and to better understand the internal fluid environment of the detector under the nitrogen flushing, the Inner Tracker is modelled using Computational Fluid Dynamics. Fluid flow and heat transfer simulations have been completed, revealing key features of the flow distribution during nitrogen flushing. The models also indicate a uniform temperature distribution throughout most of the volume.

1. Introduction

The ATLAS Inner Tracker (ITk) is a planned upgrade to the ATLAS detector at the Large Hadron Collider (LHC) in CERN, Switzerland. The purpose of the ITk is to plot the positions and trajectories of charged particles close to the particle interaction point [1]. The ITk will replace the current Inner Detector, which will have suffered accumulated radiation damage, while at the same time improving the coverage, radiation hardness and triggering capabilities in time for the High Luminosity LHC [1].

The internal components of the particle detectors are susceptible to damage due to high humidity: when is high enough, condensation can occur and sensitive electrical equipment can become corroded over time resulting in loss of sensitivity and even complete loss of detection, greatly reducing the effectiveness of the detector [1]. Because the internal environment of a detector is kept at $-20°$C for optimal performance, any moisture introduced can be dangerous.

This problems is solved in the ITk in two ways: continual flushing with dry nitrogen (N_2) which will keep the environment dry and humidity and temperature sensors placed throughout the detector which will be able to monitor the environmental conditions and warn if conditions lead to condensation. These "leaks" could be back-diffusion in the dry N_2 flush exhaust, leaks at the service penetrations. A fault condition in the cooling is envisaged to drop the local temperature, meaning the specification of very low humidity must be sufficient that the dew-point is below the lowest temperature reached in this extreme fault condition ($T_{dp} < -60°$C).

In order to know the best placements for sensors and better understand the fluid environment, a Computational Fluid Dynamics (CFD) simulation is used to calculate flow, temperature and humidity distributions throughout the volume. This paper will address the CFD modelling up to the point of modelling temperature and flow distributions. The results of these models are presented and the next steps in order to model humidity are discussed.

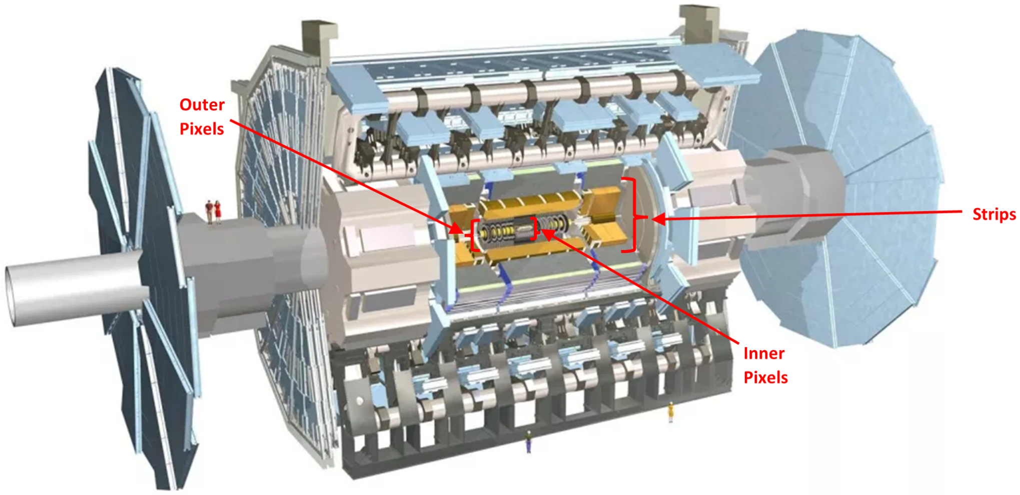

Figure 1. Overview of the ATLAS detector showing the Inner Detector to be replaced in the Upgrade by the Inner Tracker (ITk). [1]

2. Methodology

The CFD simulation design and execution has 4 stages: simplification and adaption of real geometry to suit the CFD, meshing, executing the simulation and post-processing.

2.1. Geometry

For simulation purposes, the ITk volume is split into 3 sections, from outermost to innermost: the Strips, the Outer Pixels and the Inner Pixels, shown in figure 1. The work presented in this paper concerns only the ITk Outer Pixels.

The Outer Pixels can be thought of as a pipe filled with sets of dozens of discs, which detect particles, as well as support cylinders. The main section of the Outer Pixels can be divided into 3 parts: the Left $(+z)$ endcap, the outer barrel, and the Right $(-z)$ endcap, as can be seen in Figure 2. It widens at each end into the two Patch Panel 1 (PP1) flanges. There are 12 flushing inlets of 10mm wide on the left flange mirrored by 12 similarly sized outlets on the right flange. The outer barrel services envelop the endcaps and then insert into the outer barrel. The Pixel Service Tube (PST) separates the Outer Pixels from the Strips and the Inner Service Tube (IST) separates the Inner and Outer Pixels.

The geometry is made complex by the transitions from large to small length scales. There are many channels on the *mm* scale typically between the detector discs and support structures. This becomes especially challenging when meshing for CFD as it requires sudden changes in cell size around these narrow channels.

A cross section of these components in the yz-plane is shown in Figure 2. The geometry is simplified as much as possible without changing any fundamental features. The Outer Pixels are symmetrical across the yz-plane so only half of the volume requires meshing. The y and z axes are not symmetric due to gravity and flow direction respectively. However, the ITk is still in the design phase, thus, the geometry is always subject to changes.

2.2. Meshing

The mesh for the ITk is large at around 60 million cells, which are predominantly tetrahedral. To ensure mesh quality, all cells were required to have a skewness below 0.9. To achieve this the average cell size was set to be 10mm and face meshing was applied to the narrow channels, so that each channel was at least 2 cells wide.

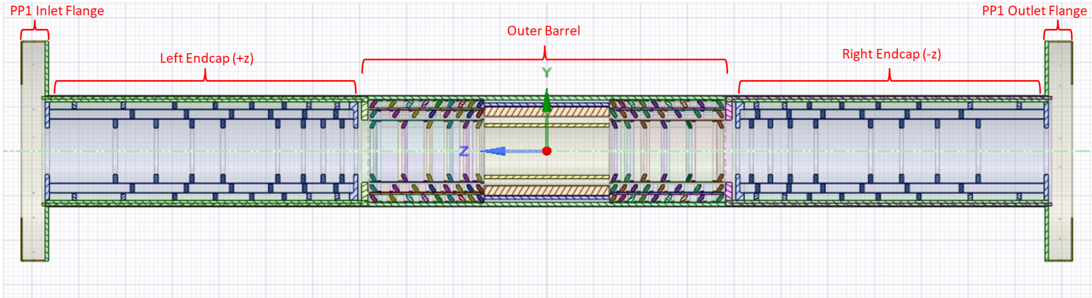

Figure 2. Cross section of the ITk Outer Pixels showing the solid components.

2.3. Simulation

2.3.1. Governing Equations The finite volume method [2] is used to simulate the fluid environment inside the ITk. The governing equations are shown in in equations (1) and (2). Equation (1) is the continuity equation and represents conservation of mass. Equation (2) represents conservation of momentum in x, y and z directions [2], [3], [4]. Equation (3) represents the heat transfer equation for the fluid and equation (4) represents the heat transfer equation for solid regions.

$$\nabla(\rho\vec{u}) = 0 \tag{1}$$

$$\rho\nabla.(\vec{u}_i\vec{u}) = -\frac{\partial p}{\partial x_i} + \nabla.(\mu\nabla\vec{u}_i) - B_i \tag{2}$$

for $i = x, y, z$. where the body forces are $B_x = B_z = 0$ and $B_y = -\rho g$.

$$\nabla \cdot (\rho\vec{u}) = -P\nabla \cdot (\vec{u}) + \nabla \cdot (k\nabla T) \tag{3}$$

$$\nabla \cdot (\vec{u}\rho h) = \nabla \cdot (k\nabla T) \tag{4}$$

2.3.2. Boundary Conditions The boundary conditions for the inlets and outlets are listed in Table 1. The inlets are designed to achieve a flow rate of 500l/hr throughout the full volume.

Table 1. Inlet and Outlet boundary conditions for the Outer Pixels simulation

Boundary	**Type**	**Flow Rate** (kg/s)	**Temperature (C)**
Inlets	Mass Flow Inlets	7.9×10^{-5} $(6.59 \times 10^{-6} kg/s$ per inlet$)$	10
Outlets	Pressure Outlets	Atmospheric pressure	Fluent Calculated

The temperature boundary conditions are shown in the Table 2. Most solids are excluded from the simulation to simplify the model and are instead represented as constant boundary conditions. The outer barrel services and bulkheads are deemed to be thick enough to be modelled, thus they are meshed and affect the heat transfer simulations. A coupled boundary condition is applied where the solid regions are in contact with other meshed regions and constant temperature boundary conditions are used where they face an external surface.

All internal detector components are maintained at a temperature of -20°C to ensure the optimal functioning of electronics and these are represented by constant temperature walls in

Table 2. Temperature boundary conditions for the Outer Pixels simulation

Boundary	Type	Heat Transfer Conditions	Temperature [C]
Flanges	Wall	Const. Temperature	25
PST	Wall	Const. Temperature	20
IST	Wall	Const. Temperature	-20
Detector disks	Wall	Const. Temperature	-20
Bulkhead	Solid region	Const. Temperature	25 on external face
	(meshed)	Coupled	Fluent calculated on Outer Pixels side.
Outer Barrel	Solid region	Const. Temperature	20 on PST side
Services	(meshed)	Coupled	Fluent calculated on Outer Pixels side.

the simulation. Temperature boundary conditions at the Outer Pixels boundaries to Strips [5] and Inner Pixels are taken from results of CFD simulations of these regions.

First a flow-only simulation is run to get a flow distribution for the volume. The energy equation is turned off and there is no heat transfer between components. After this a heat transfer simulation is run, giving both flow and temperature distribution. This can effect the flow and allows for the possibility of effects like buoyancy driven flow.

2.3.3. Convergence For the flow-only simulations, the standard residuals are used: continuity and velocity in the x, y, and z directions. For the heat transfer simulations, energy is also used as a residual. For both simulations mass imbalance is monitored. Mass imbalance is the difference between net mass in through the inlets and out through the outlets. In some cases continuity may not converge because the mass imbalance is non-zero – however, as long as the mass imbalance is stable the simulation can be considered converged. The tolerance values used for continuity, x, y, and z velocities is 10^{-3} and for energy it is 10^{-6}

Once simulations were set up and initialised, they were run on the Centre for High Performance Computing (CHPC) clusters in South Africa. Both flow-only and heat transfer simulations are found to converge within 300 iterations.

2.3.4. Solver Settings The solver uses double precision so these values are calculated halfway between each mesh node as well as at each node. Momentum and pressure at nodes shared between cells must match, meaning the equations must all be solved simultaneously. The solver's initial solution is a guess which is then used in the next calculation. The simulation is iterated in this way until it certain variables, called residuals, fall below a set tolerance level. Residuals are dimensionless and can be roughly understood as the difference between a given quantity from one iteration to the next. Once every residual is below its tolerance value in a given iteration, the simulation is considered converged.

Flow is assumed to be incompressible and, for flow and heat transfer simulations, at constant density. A viscous model is used where the flow is assumed to be laminar. This assumption is based on measurement of the flow through the inlets, which is calculated to be laminar. Gravity is set at -9.81m/s in the y-direction. Solid components are modelled as graphite and the fluid environment is assumed to be entirely N_2 for the flow and heat transfer simulations.

3. Results

3.1. Flow Results

Velocity magnitude plots indicate low velocity flow throughout the Outer Pixels. The velocity magnitude plot for the left endcap in Figure 3 shows flow is between $1 - 5mms^{-1}$ through the majority of the volume. Layer 4, the outermost layer of the endcaps marked by 1 in Figure 3, demonstrates that this layer has the highest velocity magnitude. This can be seen more clearly in the right-hand image depicting velocity plots for the yz-plane. This high velocity is due to Layer 4 having more space for fluid to flow through, while Layers 2 and 3 have only narrow $1-2mm$ channels. This could be problematic as Layers 2 and 3 may not be flushed as effectively as Layer 4. This pattern of flow is mirrored for the right endcap, while the Outer Barrel region does not show significant flow features for any layer.

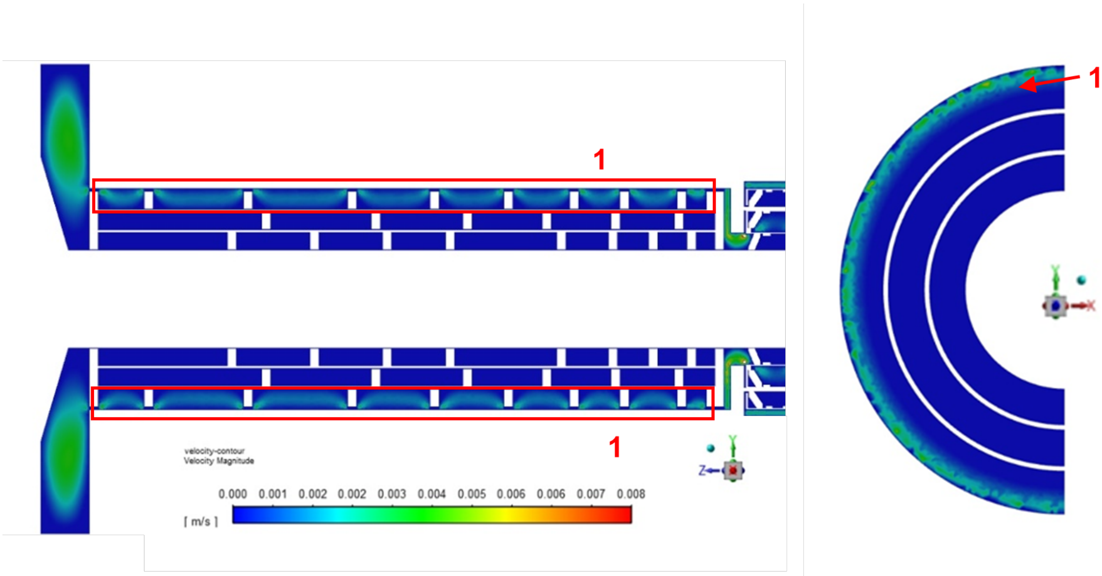

Figure 3. Velocity magnitude contours in the Outer Pixels endcap. Left: yz-plane Right: xy-plane

3.2. Temperature Results

Temperature plots for the Outer Pixels indicate that the temperature is mostly uniform at -20C throughout the volume. This is seen most easily in the right-hand image of Figure 4. This uniform distribution is due to cooling from the detector components: each component has its own cooling system.

Layer 4 of the endcap and the outer barrel services are labelled by 1, and this region exhibits a higher temperature due to the external boundary conditions on the services, which conducts heat down to Layer 4. However, this area is subject to a geometry update that will isolate Layer 4 from the services. Once this update is implemented it is expected that Layer 4 will be have the same uniform distribution as Layers 2 and 3 as it will be thermally isolated. The same temperature pattern is mirrored in the right endcap and the outer barrel discs are also at uniform temperature.

These temperature distributions would indicate that there is no preferred position for sensors to be placed based on properties of the fluid environment. As it stands, it would be advised sensors placement can proceed according to considerations of convenience, such as what areas have space, whether it is easy to insert sensors there, and a need to have maximum coverage of the Outer Pixels volume. In regards to the latter, sensors will be placed only in the endcaps and not the outer barrel. This is because leaks are only expected at the flanges and not along the volume. Any moisture due to leaks will have to pass through the endcaps before reaching the outer barrel. However, the addition of air leaks, and the subsequent humidity, into the simulations could affect temperature and flow distributions. Subsequently sensor placement may need to be reconsidered based on future model predictions.

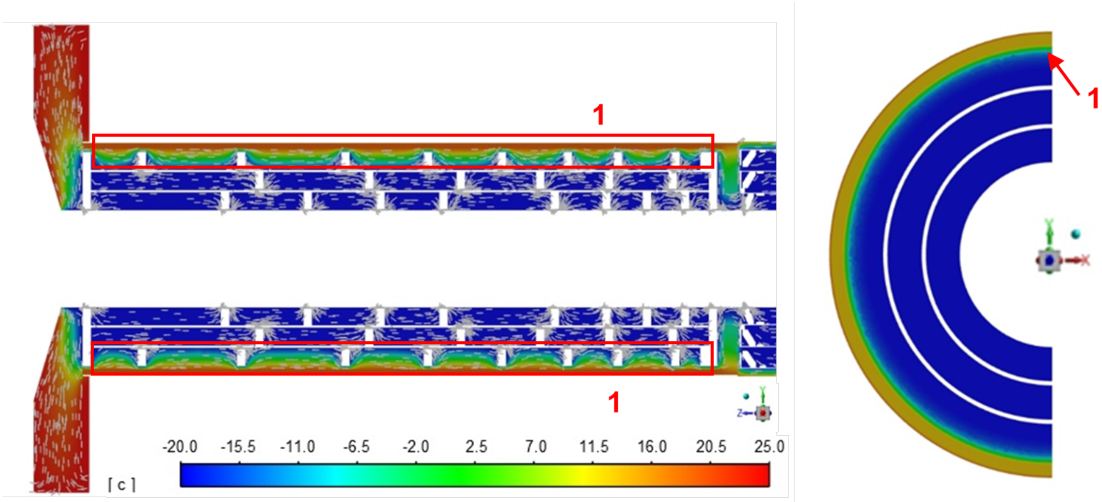

Figure 4. Temperature contours in the Outer Pixels endcap. Grey arrows indicate velocity vectors. Left: yz-plane Right: xy-plane

4. Conclusion

The aim of this work was to model the internal fluid environment of the ATLAS ITk using CFD in order to advise on the placement of sensors. Although flow and temperature distributions have been modelled, humidity must still be modelled before we can confidently assess the optimal placement of sensors. Flow distributions have indicated that there may be some areas in the geometry which may hinder flow. On the other hand, temperature distributions are almost entirely uniform throughout the inner layers, implying that there is no preferred location for temperature sensors. However, these adding humidity to the simulations could still affect these results. This will be done by introducing humid air as a new species entering the volume via "leaks" on the flanges and the distribution of densities for each species will be calculated using the species transport equation, allowing calculation of humidity distributions. A user-defined function can be created that takes temperature and humidity values at each point in the volume in order to calculate the Dewpoint. This distribution will allow us to understand at what locations condensation is likely to occur.

5. Acknowledgments

The Centre for High Performance Computing (CHPC) is acknowledged for the use of the CHPC clusters.

References

[1] ATLAS Collaboration 2017 ATLAS Inner Tracker Strip Detector: Technical Design Report URL `https://cds.cern.ch/record/2257755`

[2] Versteeg H K and Malalasekera W 2007 *An Introduction to Computational Fluid Dynamics: The Finite Volume Method* (Pearson)

[3] Bhamjee M, Nurick A and Madyira D 2013 *Energy and Buildings* **57** 289–301 ISSN 0378-7788

[4] ANSYS Inc 2021 ANSYS Fluent Theory Guide

[5] Mafa T P, Bhamjee M, Connell S H and Leeuw L L 2022 Fluid Flow and Humidity in the ATLAS ITk Studied by CFD