BNL-224172-2023-TECH
EIC-ADD-TN-046

Computational fluid dynamic modeling to determine the indoor environment of an electron-ion collider service building

L. Norman, R. Srinivasan

April 2022

Electron-Ion Collider Project

Brookhaven National Laboratory

U.S. Department of Energy

USDOE Office of Science (SC), Nuclear Physics (NP) (SC-26)

Notice: This technical note has been authored by employees of Brookhaven Science Associates, LLC under Contract No.DE-SC0012704 with the U.S. Department of Energy. The publisher by accepting the technical note for publication acknowledges that the United States Government retains a non-exclusive, paid-up, irrevocable, world-wide license to publish or reproduce the published form of this technical note, or allow others to do so, for United States Government purposes.

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or any third party's use or the results of such use of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof or its contractors or subcontractors. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

Computational fluid dynamic modeling to determine the indoor environment of an electron-ion collider service building

Logan Norman
Physics Department, University of Texas at Arlington, Arlington, TX 76013

Ram Srinivasan
Electron-Ion Collider Infrastructure Department, Brookhaven National Lab, Upton, NY 11973

Abstract

The design for the Electron-Ion Collider (EIC) calls for several service buildings that house various power supplies and control electronics for the collider ring itself. In order to operate within specified conditions, the ambient air entering the power supplies needs to be within a certain temperature range while dissipating the heat from losses. Proper cooling is therefore a necessity in the service buildings to ensure that every aspect of the EIC works as intended. Since the EIC is in the design stage, we are evaluating the indoor environment of the service building using the current design specifications. We have researched multiple forms of literature and performed the necessary calculations to compile a list of boundary conditions that accurately represent the situation at hand. We are using computational fluid dynamics modeling to solve the conservation equations for mass, momentum, and energy (Navier-Stokes). This allows us to perform a finite element analysis which will give us the flow distribution in the room as well as temperature profiles throughout the building. We have obtained a simulation result giving us the temperature profiles for the building and it shows that the placement of the racks and supply vents are essential to obtaining an even temperature distribution. This model will provide a basis for design decisions which will affect the overall cooling of the service building without extending the schedule and avoiding a costly reworking of the cooling system.

Introduction

Brookhaven National Laboratory has a long history of excellence in the advancement of science through the numerous experiments and facilities they operate. One such facility that stands out not only for the scientific advancements it has made but for its massive size and scale is the Relativistic Heavy-Ion Collider (RHIC). RHIC hosts scientists from around the world and is used to perform experiments that have given great insight into the structure of nucleons and subsequently quark-gluon interactions. To probe further, an upgrade is currently undergoing its design phase which will transform RHIC into the Electron-Ion Collider (EIC). This machine will be able to answer many of the questions in quark-gluon physics that were unobtainable before.

There are many people working on designing every aspect of the EIC, with the work being split between several teams working on different aspects. I collaborated with the infrastructure department of the EIC, and their responsibilities include ensuring the proper design and implementation of the necessary buildings, power lines, and any additional construction that the project requires. The design contains several service buildings that are to be built around the ring to house the some of the various power supplies and controls electronics that are needed to operate the collider. The ambient air entering the power supply fan inlets needs to be maintained within a certain range of temperature to ensure a consistent power output from the power supplies. The temperature variation range given by the contracting engineering firm in the concept design review is +/- 3°F.[1]

This paper focuses on determining the indoor environment of the service building through the construction and simulation of a digital model. The indoor environment encompasses the temperature distribution throughout the building and airflow patterns. If we can observe the temperature of the ambient air entering the inlets of the power supplies, we can determine if it is

within an appropriate range. The construction of this model will allow one to observe a simulated indoor environment without having to construct a physical building or implement an HVAC system. A model such as this is applicable in the design phase of projects to provide a basis for design decisions that would affect the overall cooling of the building.

Method

The main method/tool used to evaluate the indoor environment was a computational fluid dynamic simulation of the service building. In our model a few reasonable assumptions had to be made in order to keep the computation time at a reasonable level and to approximate incredibly complex processes. One of the biggest assumptions to be made for this simulation was how the power supplies themselves should be modeled. The power supplies involve complex geometry that does not have a significant effect on the overall environment of the building.[2] Therefore, the details do not need to be modeled as they would only needlessly increase the computational load. The simplified model we used for the power supply can be seen in Figure 1 and consists of a hollow tube that runs through the racks with a heat source block located in the middle and a fan boundary condition on the end forcing air to flow through it. This hollow tube model is similar to the approach taken by Jefferey Rambo and Yogendra Joshi.[3] This assumption allowed us to split the heat loads and corresponding fans up into smaller more realistic pieces, thereby striking a balance between complexity and computation time.

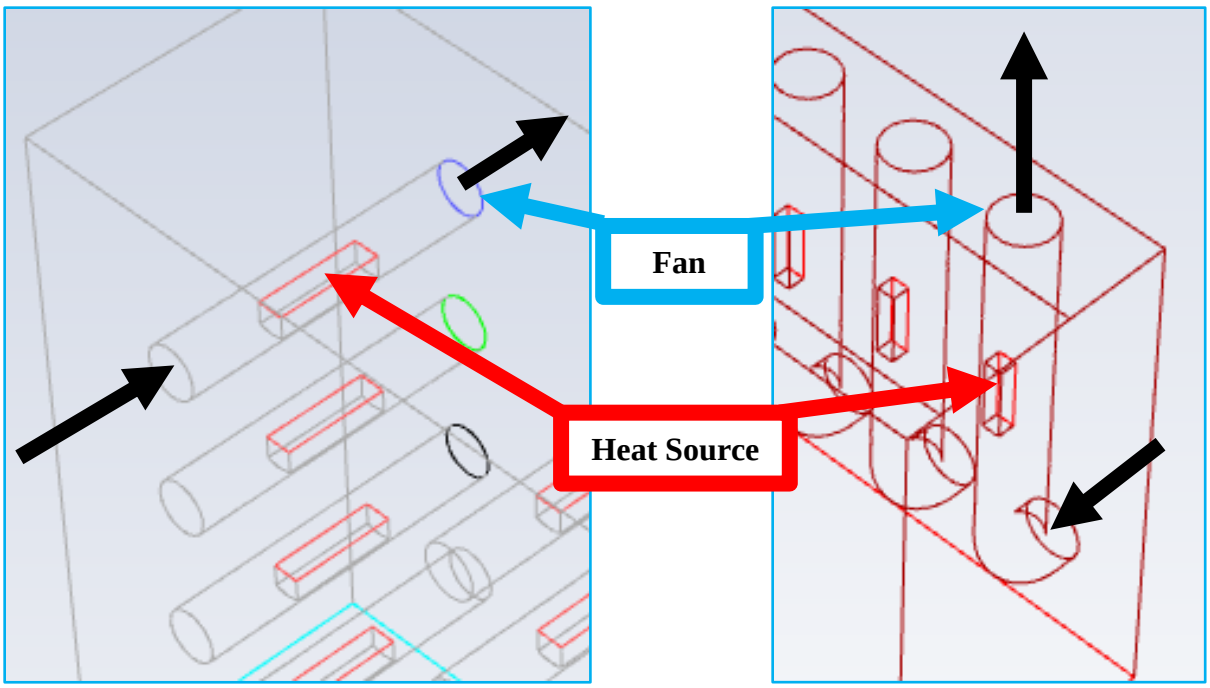


Figure 1. Simplified Power Supply Model with Horizontal Exhaust (Left) and Vertical Exhaust (Right)

In Figure 2 you can see the entire model setup, which illustrates a few of the other general assumptions made. For instance, we considered the power supply racks to be simple rectangular prism solids with several of the power supply tube models in them, with the number of tubes corresponding to the overall heat load of the rack. The cable trays were assumed in the model to just be solid blocks that generate a volumetric heat load based on the losses associated with the various power cables running through them. The building dimensions itself were taken from the EIC CD-1 design for the alcove 9 building, with the assumption that each room was separated with no doors or windows. There are 9 supply vents in the main room and 7 supply vents in the controls room, indicated by the blue arrows pointing into the rooms in Figure 2. There is an equal number of return vents in each room, indicated by the red arrows pointing out of the room, and both the supply and return vents are 2 ft. x 2 ft. squares.

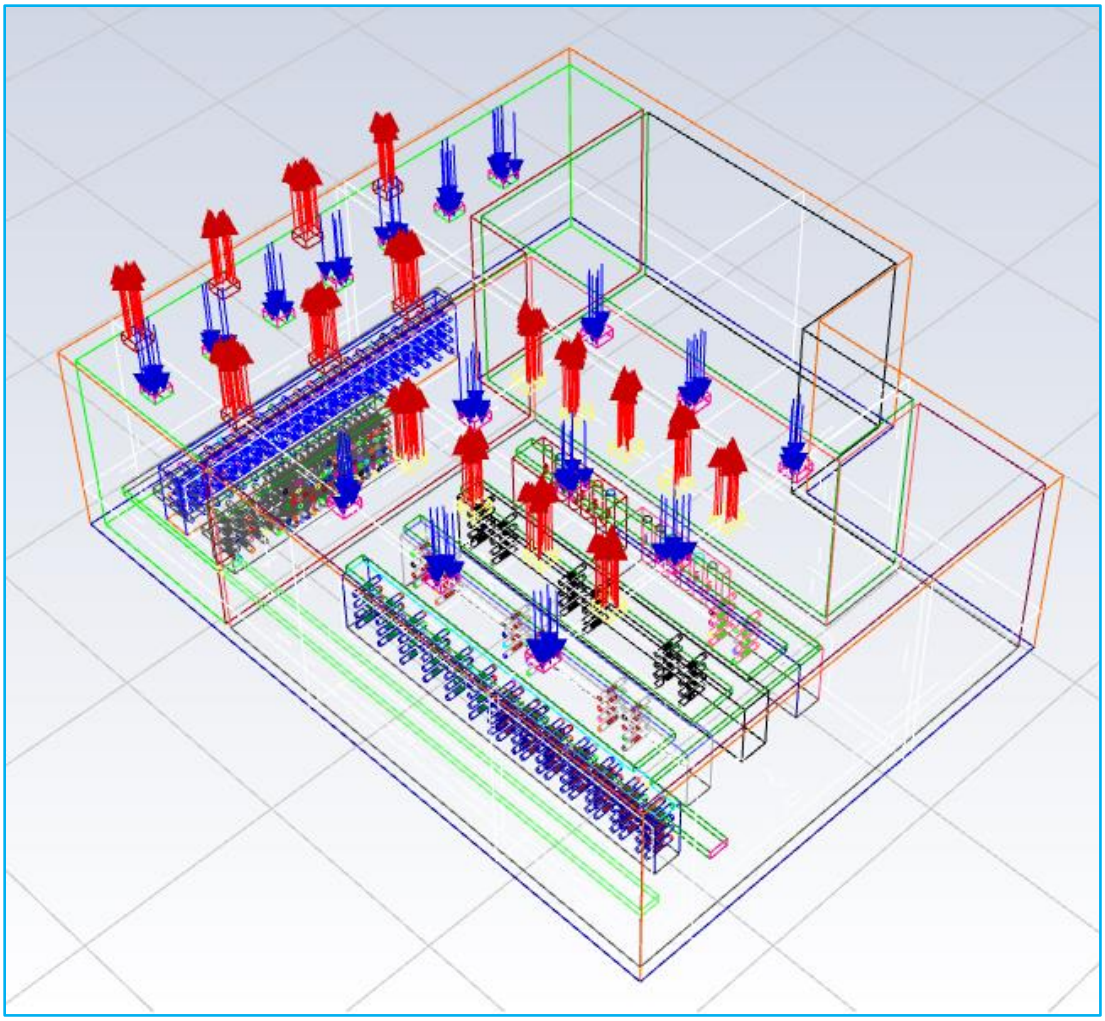


Figure 2. Overall Model Setup

In any simulation the boundary conditions are the base point from where a finite element calculation begins and propagates through, and they are one of the few user inputs. Therefore, the boundary conditions must represent the most realistic scenario within a reasonable uncertainty. The supplies were set to a reasonable number for the space and set to a standard temperature of 55°F.[4] The necessary supply air velocity was determined using a calculation of the mass flow rate from the specific heat equation[5], shown below as equation 1.

$$\dot{Q} = \dot{m} C_P \Delta T \quad (1)$$

The power, $\dot{Q}$, was taken from the total heat load for the building by summing the heat loads of the racks (provided by the EIC Power Supply Group), the solar heat load from the walls, and the heat load from the cable trays. The isobaric specific heat value, $C_P = 0.24\ \frac{BTU}{lb\cdot°F}$, is a constant taken from the PE Mechanical Reference Handbook[5], and the ΔT was assumed to be the difference between a standard room temperature 75°F and the standard supply air temperature of 55°F.[4] The mass flow rate was substituted with equation 2 below in order to use variables relevant to design and simulation inputs such as the supply air velocity and area of the vents.[5]

$$\dot{m} = \rho V A \tag{2}$$

The area, A, as stated before consisted of 4 ft^2 vents and then multiplied by the number of vents in the room. The density of air is constant, $\rho = 0.0763\ \frac{lb}{ft^3}$, in this case and taken from the PE Mechanical Reference Handbook[5], and the equation can be solved for velocity as shown below in equation 3.

$$V = \frac{\dot{Q}}{\rho A C_P \Delta T} \tag{3}$$

The calculation resulted in an average velocity of around 1000 ft/min which is the velocity we used for each individual supply vent for each room. The supply vents also had a 15° angular component off the vertical axis on each half to simulate the effects of a diffuser. The return vents were set to be at a constant zero-gauge pressure outlet. The heat sources are set to heat flux inputs based on the power supply loss data given to us by the EIC Power Supply Group and proportional to the area of the heat source geometry. We also added a convective heat transfer boundary condition to the outer walls to simulate the heat transfer from the outside air during a summer day. For the variables involved in the convective heat transfer we used a convective heat transfer coefficient of 12 $\frac{W}{m^2K}$ and a free stream temperature of 90 °F taken from

literature sources.[6] We used the Shear Stress Transport (SST) k-ω model as our choice of turbulence model within the ANSYS Fluent solver with gravity and viscous heating included. We also utilized the steady state solver to reach a solution and not the transient solver, however, we did compare the steady state solver to the transient solution ran for an hour of simulated time and found little difference between the two.

Results

The simulation was run for a steady state solution over a total of 2000 iterations and the residuals shown in Figure 3 below show that the simulation did reach a steady state. We determined it had reached a steady state when the residuals, which represent the percent difference of the various simulation parameters between iterations, began to vary a steady and minimal amount between iterations.

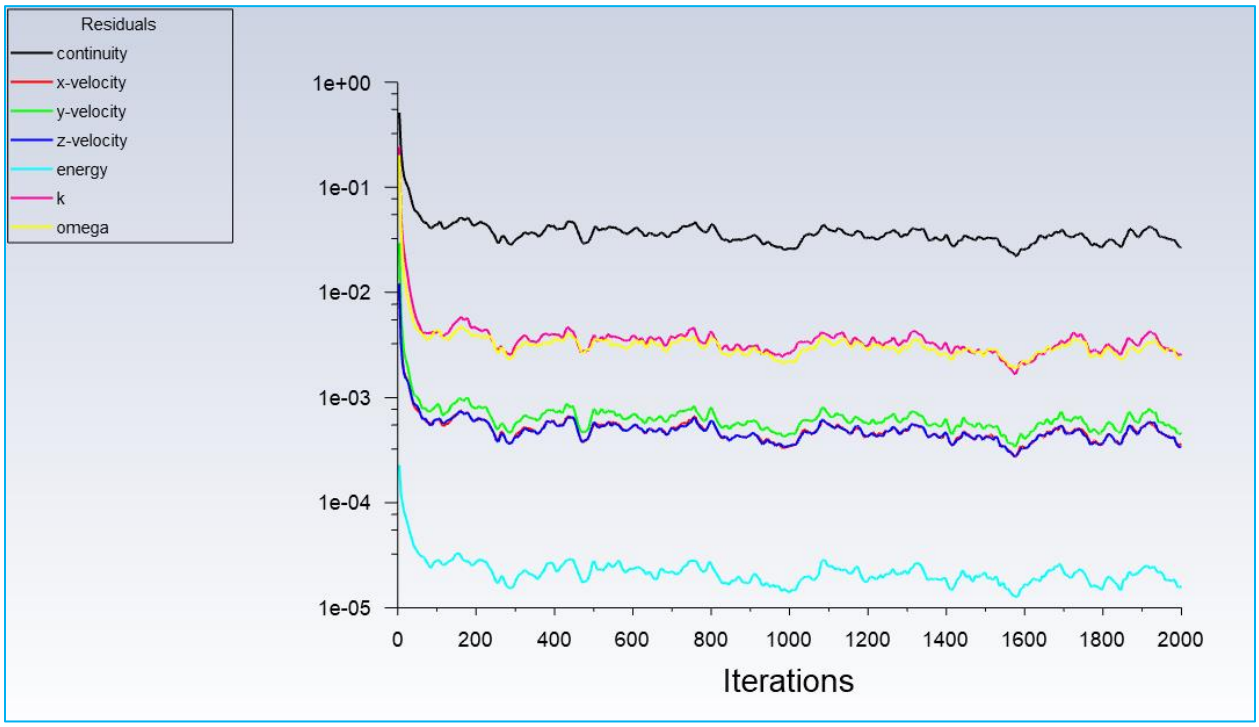


Figure 3. Residuals of steady state simulation

In Figure 4 below there are two images of the temperature profiles that the simulation gave as an output from the steady state simulation for the building. The image on the left includes an isometric view of the entire building with select cross-sectional planes which show the temperature distribution throughout the building. The image on the right is a view of the building looking down from the ceiling at a cross sectional plane at an elevation of 5 feet from the ground, standard height of a thermostat. The temperature profiles that were generated give a visual guide as to how the temperatures vary throughout the building.

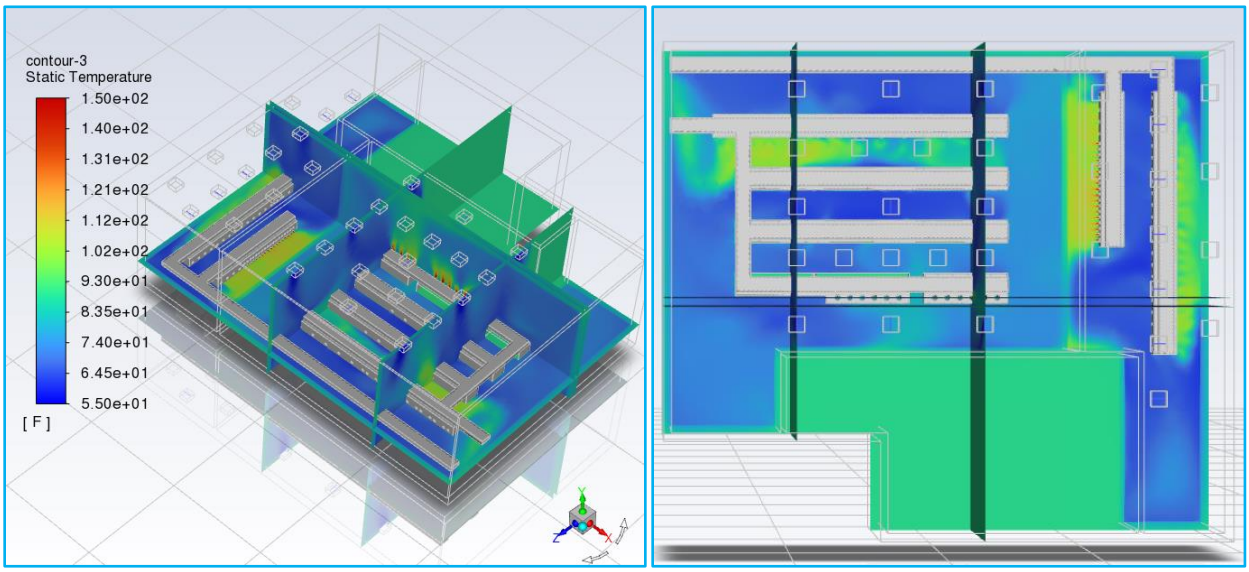


Figure 4. Service building temperature profile isometric view (Left) and plan view at an elevation of 5 ft. (Right)

The key points of this investigation are the front faces of the power supplies, or more specifically, the area in which air flows into the power supply vents for cooling. To investigate these areas in the simulation we created planes that were bounded at the front faces of each row of power supplies as depicted below in Figure 5.

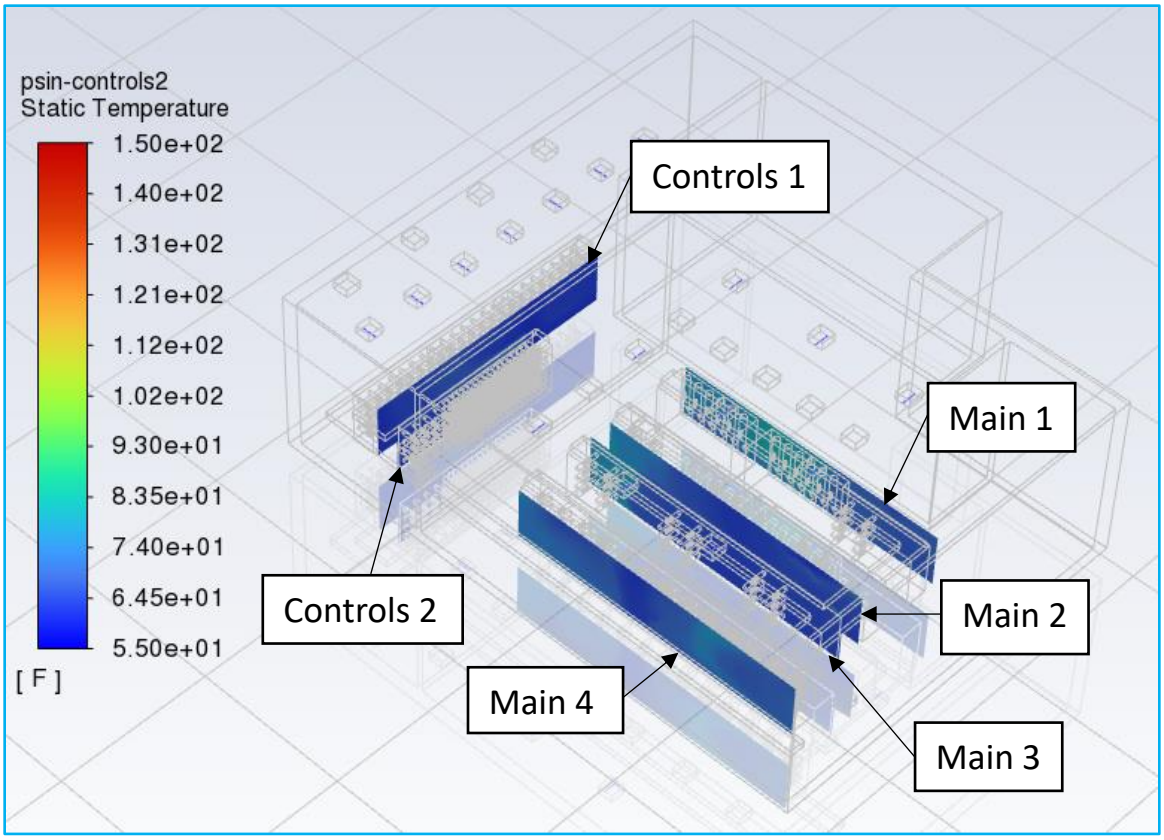


Figure 5. Temperature profiles for the front faces of all six racks

The temperature profiles depicted in Figure 5, like the cross-sectional planes depicted in Figure 4, are for a visual guide as to the planes locations as well as the temperature distributions throughout the planes. To further investigate the exact temperature ranges and variation within the front rack faces we plotted the temperature of each mesh node for each rack face and produced Figure 6 below. The labels of the rack faces are the same between Figures 5 and 6 to give a physical reference for where the data in the plots corresponds to in the building model itself. We also plotted two temperature bands to represent the design criteria in red of +/- 3°F and a slightly larger band for reference in blue of +/- 5°F.

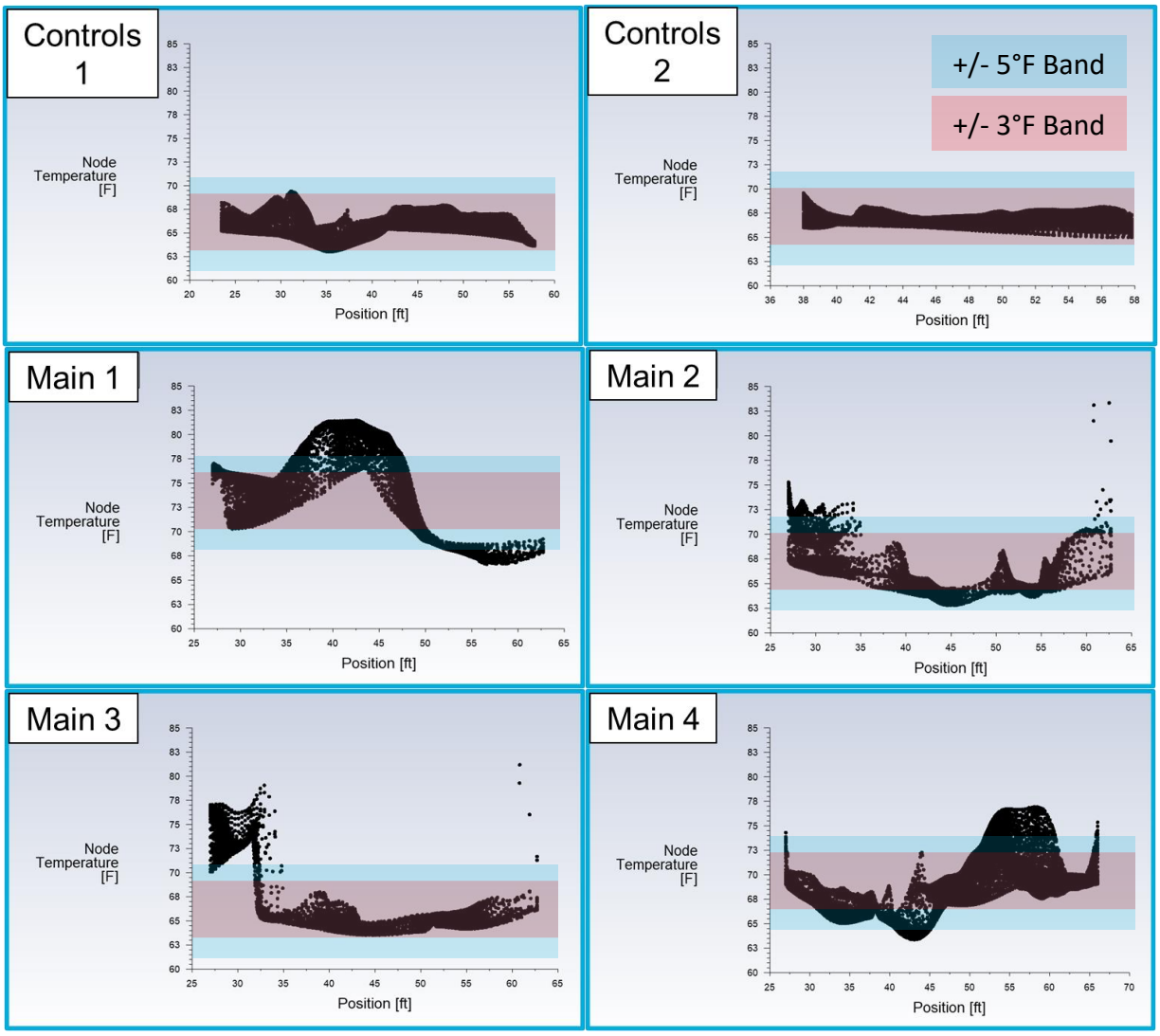


Figure 6. Temperature plots for each mesh node along each front rack face with a +/- 3°F band plotted in red and a +/- 5°F band plotted in blue for each plot

The temperature in the control electronics room does not vary outside of the 3-degree band and is relatively consistent within the band as well for both inlet planes. The main room comparatively has more variation in temperature, most likely due to the amount of power supply heat generated in the room is larger and the room size is harder to cool. All four inlet planes show temperature variation outside of the 3-degree band, with some deviating more severely than others. The row indicated by main1 has the most deviation, however, main1 also contains

the largest power supplies which do not require as stringent of a temperature band for their consistent output.

Conclusion

We have created a computational fluid dynamics model that solves the Navier-Stokes equations to determine the indoor environment of an EIC service building. This model has allowed us to predict the temperature profile of the service building which houses critical electronic equipment that requires temperature control. The analysis gave us insight into how the placement of the power supply racks affect the temperature distribution of the room as well as the placement of the supply and return vents. The temperature contours also reveal the locations of hotspots which tell us where we can assign different zones to the room and reduce the flow rate coming into a cooler zone and increase the flow rate in the hotter zones. The simulation showed that the controls electronics room does not deviate beyond the 3-degree band while the main room has some locations in which it does. This could be due to artificial reasons within the simulation, an inaccuracy in the boundary conditions, or it could be the real results. Further steps can now be taken to refine the model's boundary conditions to continue the development of the model and improve accuracy. For instance, improved data can be obtained for flow rates of the individual rack power supply fans from manufacturers. The mesh can also be refined so that any mesh dependency is reduced to an insignificant amount. A zone coverage simulation could be implemented to more effectively even out the temperatures in the rooms. As the design of the EIC progresses the model can be updated to match current vendor data and building design. It would also prove useful to benchmark the model with a physical building already in operation using operational data. This model is a tool for determining how a design decision, such as

swapping power supply locations or adjusting the air conditioning flow rates, will affect the overall indoor environment of the service building.

References

[1]"Concept Design 100% Review Submittal", Nov. 6th, 2020, HDR

[2]Hermansen, Knud A. "Validation of simplified rack boundary conditions for numerical data center models." *Diss. University of Colorado at Boulder*, 2011.

[3]Rambo, Jeffrey, and Yogendra Joshi. "Modeling of data center airflow and heat transfer: State of the art and future trends." *Distributed and Parallel Databases* 21.2 (2007): 193-225.

[4] "Handbook, A.S.H.R.A.E. Fundamentals", *ASHRAE–American Society of Heating Ventilating and Air-Conditioning Engineers* (2017).

[5]NCEES. *PE Mechanical Reference Handbook*. Version 1.2, NCEES, 2019.

[6]Popovici, Cătălin George. "HVAC system functionality simulation using ANSYS-Fluent." *Energy Procedia* 112 (2017): 360-365.

Acknowledgements

I would like to thank my mentor, Ram Srinivasan and Charlie Folz for their full support and advice throughout this project.

This project was supported in part by the U.S. Department of Energy, Office of Science, Office of Workforce Development for Teachers and Scientists (WDTS) under the Science Undergraduate Laboratory Internships Program (SULI).