Using BIM and Flow Simulations to Improve Air Flow in Server Rooms

Stephen McNamara

TUS

Aim of the Project

The aim of this project is to design a Server Room using data from Moylish's comms room to help create a custom model in **Revit**. This model is then simulated in **SolidWorks** to determine the airflow of the current layout. Different layouts are created and simulated to determine the best possible layout to extract the most amount of heat from the servers.

Objectives

- ❑ Research the background, designs, and layouts of data centres.
- ❑ Observe, measure, and collect information on the Moylish's comms room.
- ❑ Create a **Building Information Management** (BIM) model of a server room using the research and data collected as a guideline.
- ❑ Perform **Computational Fluid Dynamics** (CFD) simulations on a **SolidWorks** model of the server room.
- ❑ Multiple simulations are done on different layouts to determine the best layout.

Figure 1: Moylish Comms Room

Server Rooms

Server Rooms are areas which hold computers that can store data, perform complex calculations, train and store **Artificial Intelligence** (AI) software. These applications produce a ton of heat that emits from the servers, if this heat is not regulated or properly treated, it can result in overheating which can cause damage to the servers or cause a fire hazard. To minimize this risk, an optimal cooling system must be in place to take away as much heat as possible from the servers. Not only is there different options to optimize cooling, but the layout of the systems are also important.

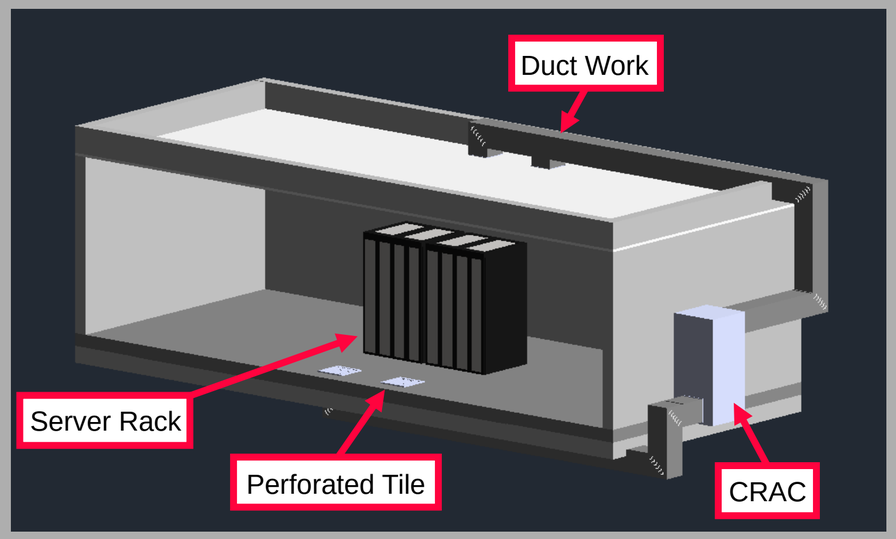

Figure 2: Data Centre Revit Model

Revit

Revit is a **BIM** software that allows users to create accessible, updated models on a building. In this dissertation, Revit is used to create all layouts of the model as it allows further customization of the equipment used. It also allows the creation of the ducting system.

SolidWorks

SolidWorks is a **CAD** software that allows users to create models. Models created for this dissertation were the perforated tiles, and the **Computer Room Air Conditioner** (CRAC). These models can then be transferred into Revit to use in the model.

CFD Simulation

A CFD simulation is performed on different layouts to determine the best air flow trajectories. These were performed on SolidWorks by converting the Revit model. The data acquired from the Moylish comms room was used to set up the simulations. The following data was used for one of the test:

Temperature = 19.5, 27.7 ºC	**Flow Rate= 8, 15 l/s**
Humidity = 31.5% RH	**Measurements = 446 x 1117 cm**

Many simulations were done where the temperature, flow rate of the air, and layout of the equipment were altered to produce different results.

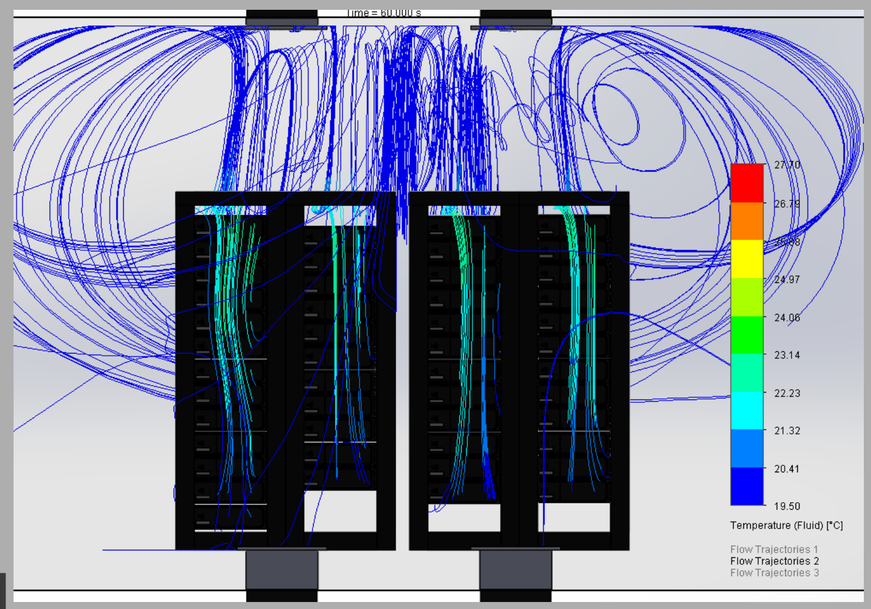

Figure 3: CFD Simulation with Flow Trajectories

Conclusion

As of the making of this poster, the best measurement for the data centre is to have the air intake and outtake on different sides of the room, with a flow rate of 8 l/s at 19.5ºC. The temperature from the servers at 27.7 ºC, and a flow rate of 15 l/s. This gave an air temperature of around 23 – 24ºC outside the servers and an outtake of 19.5ºC.