EuroCC@Turkey

https://eurocc.truba.gov.tr/

This document is prepared by EuroCC@Turkey for EuroCC under GA NO 951732

CASE STUDY REPORT

CFD Simulations for Powertrain Test Labs

NCC Partner	*METU (Middle East Technical University)*
Company* ***Contact***	*Ford Otosan, Ford Otomotiv Sanayi A.Ş. - http://www.fordotosan.com.tr/* *Hande Bezci Zeren - hzeren@ford.com.tr*
NCC Expert	*Nilay SEZER UZOL - nuzol@metu.edu.tr*
Start & End Date	*15.10.2021 - 15.04.2022*

***Company** accepts that the Case Study Report is shared with the EuroCC Project and the community through the EuroCC@Turkey awareness creation activities and platforms.

1. Problem Identification

Ford Otosan (Ford Otomotiv Sanayi A.Ş.), is a publicly traded (18%) company, where Ford Motor Company (41%) and Koç Holding A.Ş. (41%) have equal shares. Ford Otosan R&D Center is the global hub for heavy commercial vehicles and related powertrains and also global spoke for light commercial vehicle development and diesel powertrain engineering. Ford Otosan R&D organization possesses all the capabilities and infrastructure required to design, develop, and test a whole vehicle, including its engine, from scratch to the complete commercial product.

Ford Otosan R&D has a group which uses CFD extensively during their design work. They also have a test group which performs related performance test for different design parts. This test group is having problems with their test rooms in their labs about satisfying and controlling the room conditions such as flow and temperature distribution and control around the test setups during the tests. They are planning to perform CFD simulations for selected test labs to understand the room conditions during the tests. The test group has limited experience on CFD usage, and they would like to learn and try TRUBA HPC infrastructure with open source CFD software for investigation of their flow and heat transfer problems in their Powertrain Labs.

2. First Suggestion

We plan to follow the following steps as the solution:

1. Going over the current tasks of Ford Otosan that need CFD analysis
2. Learning about Truba's HPC infrastructure
3. Determining the task(s) and preparing the required pre-processing grid and input files to be migrated
4. Test Case CFD Simulations: Steady-state runs
5. Parallel Performance Analysis and Flow Performance Analysis and Discussions

Milestones: (6 Months)

- 20.08.2021 – Project Definition and Preparation
- 05.10.2021 – Project Definition and Preparation Updated
- 15.10.2021 – Start of the Project
- 30.11.2021 – CFD software trials on Truba's HPC infrastructure
- 30.01.2022 – Initial CFD runs on Truba's HPC infrastructure
- 30.03.2022 – Detailed CFD results obtained on Truba's HPC infrastructure
- 15.04.2022 – Simulation procedure documents, test case files, and final report with performance analysis

Timeline	**Tasks and Milestones planned**
Oct'21 Nov'21	Orientation on the TRUBA platform Investigation of the tools that can be used for this case study Discussing the possibilities with the TRUBA team Set up the environment
30 Nov'21	Milestone: The development and runtime environment prepared
Dec'21 Jan'21	Initial steady-state CFD runs for selected test room
30 Jan'21	Milestone: Initial results and performance analysis

Feb'21 Mar'21	CFD simulation trials for test rooms with different conditions
30 Mar'21	Milestone: Results and performance analysis
15 Apr'21	Final Report and Future Recommendations

3. Solution Stage – I

1. Going over the test room conditions of Ford Otosan that need CFD analyses
2. Learning about Truba's HPC infrastructure
    a. CFD software trials on Truba's HPC infrastructure
    b. To obtain the accounts and to do initial training on how to use Truba for CFD analysis
3. Determining the task(s) and preparing the required pre-processing grid and input files to be migrated
    a. To prepare the sample geometry and grid
4. Test Case CFD Simulations: Steady-state runs
    a. To perform CFD software trials on Truba's HPC infrastructure
    b. To prepare procedure for following test cases
    c. To perform steady-state CFD results with open source CFD software obtained on Truba's HPC infrastructure
5. Trials and Discussions on CFD simulations for different test rooms and conditions
    a. To prepare final report

4. Solution Stage – II

<When a new addition is made to the first solution, different solution stages can be given by duplicating these sections respectively.>

5. Results and Achievements

In this test case study, the CFD modeling approaches for indoor environment air flow for conditions heating or cooling purposes are investigated to be able to model test rooms of Ford Otosan where they have different test setups with heating sources in the rectangular rooms with air ventilation systems. During the tests, the test setups which have inner test sections working with hot air flow heated with boilers in the room. These large rectangular boilers are the big heat emanating sources and they cause the room gets hot during the tests which prevent having steady-state test room temperature conditions cause the test to be halted until the room temperatures get lower again. The ventilation system is using the natural air from outside at the temperature of outside weather conditions that changes daily and seasonally, and it takes long time to reach desired temperature room conditions.

By using parallel CFD simulations with open source CFD software, it is aimed to obtain the test room air flow conditions with heating sources located in the room to understand the

ventilation and flow characteristics around the test setups to find a better solution for the ventilation of the test room. The problem description can be done as that in the test room problem, a room with an inlet, an outlet and a heating source in the middle will be investigated in terms of temperature distribution and flow field.

Summary of Progress and Results, Feb-June 2021:

In literature, the examples of data center cooling and hospital ventilation problems are found because of the alikeness of basic aspects such as a closed room with cool/fresh air inlet, hot/contaminated air outlet, a heating/contamination source in the room. The literature review will show the approaches about the inlet, outlet, heat source and wall boundary conditions which will be crucial in the test room problem in this test case study.

Short literature review is presented below.

Literature Review

1. Analysis of Air-Cooled Data Center

Wibron [1] studied the CFD simulations of the server rooms by using ANSYS CFX 16.0 software. The main components of the data centre are the Computer Room Air Conditioner (CRAC) units and the server racks. The server racks contain lots of working electronic components which are generating heat so that they need to be cooled down to keep them in the range of their operation temperature region. In CFD simulations, mostly the server racks are assumed as boxes with two surfaces so that one surface is the inlet to take cool air and the other surface outlet which dissipates hot air. The CRAC in the server room supplies constant temperature cool air with constant velocity from one surface and takes up the air from another surface. The different configurations of the CRAC units can also be used.

1.1 Geometry and Conditions

In the CFD simulations of Wibron [1], the original setup of the server room has two CRAC units and nine server racks arranged in three parallel rows. These all units are simplified as a rectangular black box which means the inside flow patterns are not investigated so that the boundary conditions are applied to the surfaces of these boxes. Figure 1 shows the computational domain where hexahedral structured grid is used with the maximum cell size as 5 cm for geometrical simplicity.

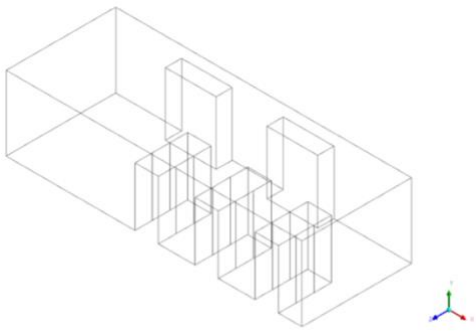

Figure 1. Geometry of server room from the CFD Model of Wibron [1]

In the CFD model [1], the walls, the floor and the ceiling are assumed as adiabatic without any leakage. The CRACs are supplying air with 0.7 m/s velocity and constant 17°C temperature from one surface and absorbing the air from another. Also, the mass flow rate is kept constant between inlet and outlet boundary condition on a CRAC. The internal fans in the server racks draw air through the front at a velocity of 0.58 m/s. Each server rack generates a heat load of 5.7 kW. It has been assumed that all the server racks generate full heat load. The heat load results in a temperature difference between the back and the front of the server rack. In the CFD simulations, the k-epsilon turbulence model is used.

1.2 Results

In the CFD simulations [1], various room settings are investigated to find the better distribution of server racks and CRAC configurations. As the temperature distribution at the inlet of server racks are important for server room problems, the temperature distribution at those surfaces are plotted as can be seen in Figure 2. Also, the streamlines of the flow is observed and tracked as shown in Figure 2 to be able to see how much of the flow enters to the server racks directly from the CRAC in percentage.

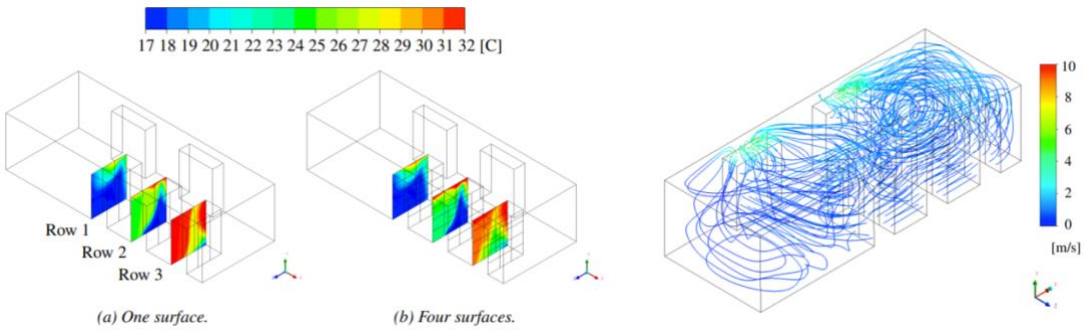

Figure 2. a) Temperature distribution at the front of server racks (left) and b) streamlines of the flow in the room (right) [1].

2.0 Temperature Monitoring and CFD Analysis of Data Centre

Another research for CFD analysis of data centre was presented by Hassan et al., 2013 [2]by using Fluent software. In this CFD model [2], the hot/cold aisle layout configuration is used as seen in Figure 3. The front faces of the server racks look at each other which form the cold aisle. From the front face to back face of the server rack, the air gets hot. There is a Computer Room Air Conditioner (CRAC) which takes hot air and delivers cold air under the raised floor.

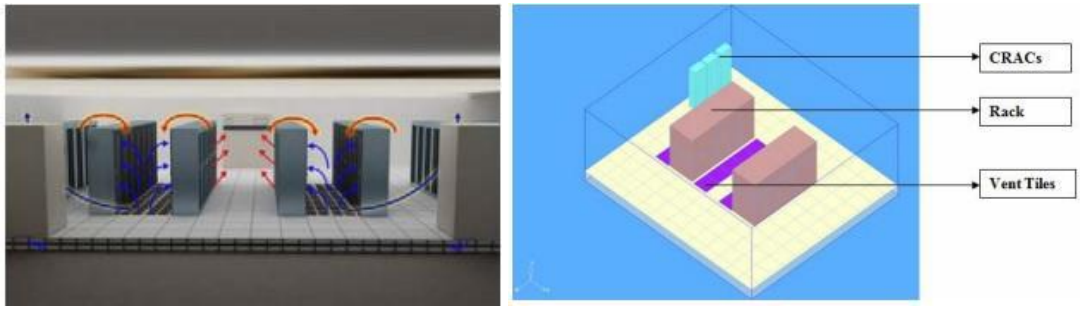

Figure 3. The hot/cold aisle layout configuration and the 3-D view of the data centre [2].

2.1 Geometry and Conditions

In the CFD simulations [2], the data centre with different configurations were investigated. Three down flow type CRAC units are used. There are two rows and each row is composed of

5 racks placed on a raised floor. The CRACs units are part of the raised floor layout with cooling air through plenum and venting in the room through vent tiles. The air flow direction and air flow rate of 0.23 m^3/s are defined for each 3.46 kW rack. Inside the racks, there is an air-moving device forcing the air straight through the rack, with a constant velocity across the front and back of the racks. The size and details of the geometry is listed in Figure 4.

2.2 Results

From the CFD simulation results [2], the temperature distributions inside the data centre can be seen in Figures 5-7. Figure 5 shows the temperature distributions at different heights at 3 and 6 ft height. The temperature rise is higher at 6 ft height at the end rack of the data centre. Also, the hot air which comes from the backsides of the racks is being recirculated into the cold aisle and it is mixing with cold air which is supplied from the vent tiles. Hence, the inlet temperature of the racks are higher than that of the supplied cold air. As can be seen in Figure 6, there are also significant temperature gradients inside the data centre. Hot air moves to the cold aisle both from the sides and top of the racks, which means different recirculation patterns occur as can be seen in the figures. As shown in Figure 7, the highest temperature zone occurs at three quarter plane along the length which corresponds to the exit of the second rank. On the other hand, the lowest temperature zone occurs at three quarter plane along the width, however, there is also high temperature zone at the end edges of the server rack since in this regions, server racks do not take air directly from the vent tiles, they take air which is composed of both room air and air exiting from the server exits.

Model ID:	Nur-3rdData_Model		
Simulation Date:	Mar 7, 2012		
Mesh Density:	Medium		
Mesh Count:	27812		
Data Center Layout:	Raised Floor with Room Return		
Data Center Components:			
	Room Dimensions	8.05 X 7.08 X 2.96	
	Room Floor Area	56.99 sq-m	
	Supply Plenum Height	0.45 m	
	Number of CRAC Units:	3	
	Total Flow Through CRAC Units:	11.704 m^3/sec	
	Supply Air Cooling Capacity:	42055.68 W	
	Number of Rack Rows:	2	
	Total Number of Racks:	10	
	Total Rack Heat Load:	41.12 kW	
	Total Flow Through Racks:	2.562 m^3/sec	
	Number Tile Rows:	3	
	Total Number of Vent Tile:	15	
	Total Flow Through Tiles:	11.707 m^3/sec	
	Ave Flow Through Each Tile Vent:	0.78 m^3/sec 0.78 m^3/sec	
Power:			
	Rack Heat Source:		41.12 kW
	Wall Heat Transfer:		0.89 kW
	Total Cooling Load:		42.01 kW
	Heat Density:		737.18 W/sq-m
	Supply Air Cooling Capacity:		42055.68 W
Temperature Details:			
	Highest Temperature in Data Center Room:		29 °C
	Highest Inlet Temperature in Racks rackrow_0_1 Rack number 1:		22 °C
Flow Through Tiles:			
	Maximum Flow Through a Vent Tile tilerow_0 Tile number 2:		1.727 m^3/sec
	Minimum Flow Through a Vent Tile tilerow_1 Tile number 4:		0.131 m^3/sec

Figure 4. The Data Centre Description [2]

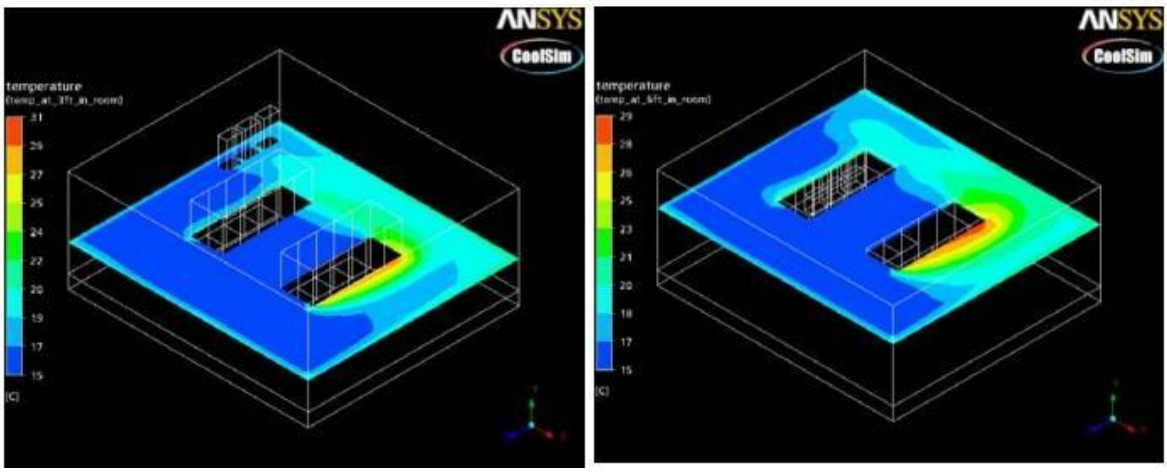

Figure 5. Temperature distribution of data centre at 3 ft and 6 ft height [2,p.555]

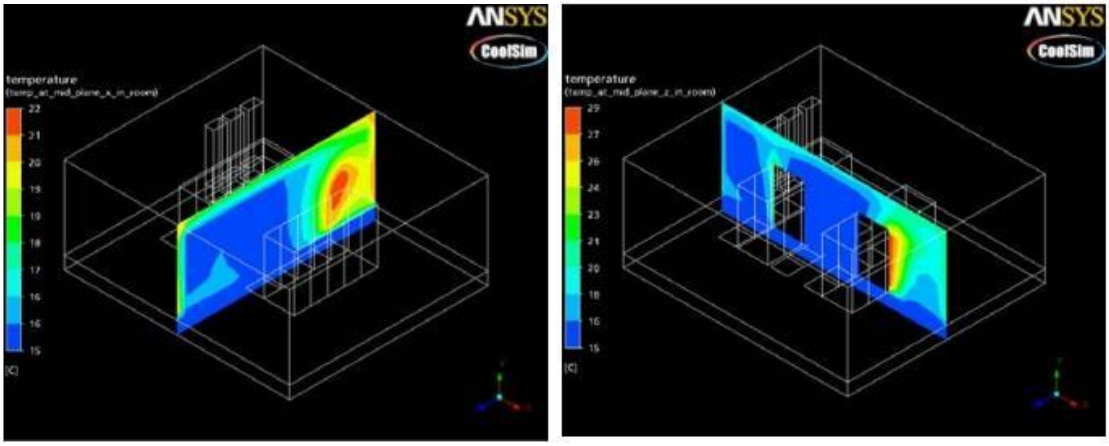

Figure 6. Temperature distribution of data centre a) at midplane along the length (left) and b) at midplane along the width (right) [2].

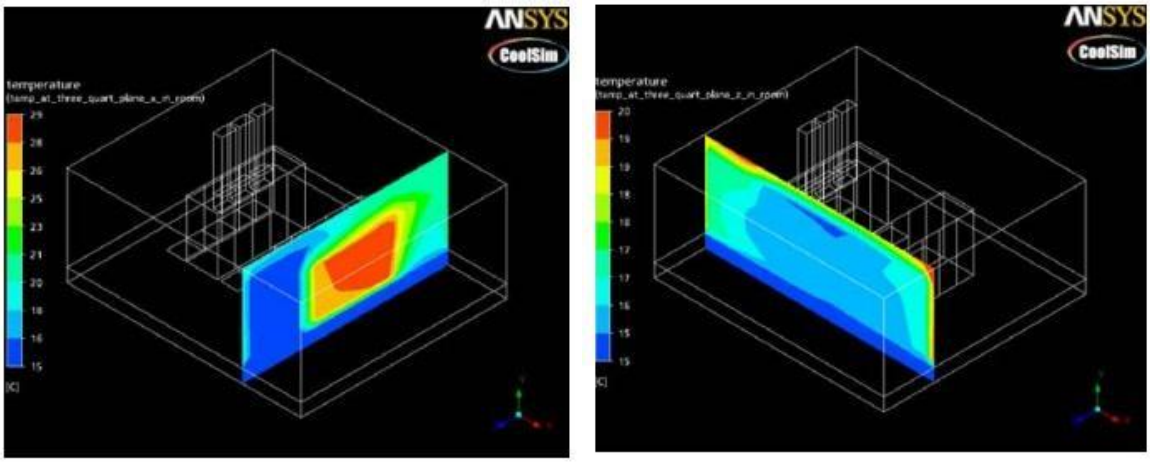

Figure 7. Temperature distribution of data centre a) at three quarter plane along the length and b) at three quarter plane along the width [2].

3. CFD Investigation of a Room with Heater

CFD investigations were performed by Sobhi and Khalil, 2019 [3] by using Fluent software to investigate the air flow patterns and thermal comfort analysis in a room with different configurations of heating systems. The aim was investigating thermal comfort with optimum heater configuration as the heating of buildings is responsible for 40% of the total energy consumption. In all cases with different radiators, the room as shown in Figure 8 was simplified to a room model consisting of a heater, a window, an air supply vent and an air exhaust vent.

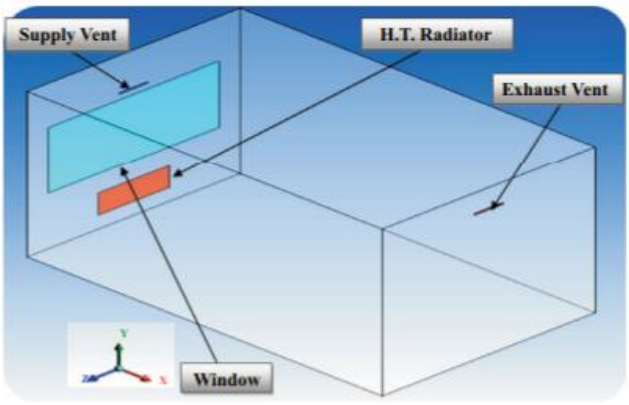

Figure 8. Room configuration [3]

3.1 Geometry and Conditions

The room dimensions considered in this study [3], as shown in Figure 8, were 5 m length, 3 m width, and 5 m height. The room had a window with dimensions of 2.4 m x 1.2 m, located at a height of 0.9 m above the floor level. They tried a couple of heater configurations such as heated wall, heated floor, radiator located under the window. The volume grid was generated by using almost 4,000,000 tetrahedral cells, using the growth rate function option, and the grid could be dense and small near the heated walls and radiators and growing far from them.

The outdoor temperature is assumed as -5°C, whereas the indoor temperature was set to be 5°C. Walls are assumed in constant temperature with zero heat flux. The ventilation air was assumed to enter the room at a temperature of -5°C, through a supply vent with dimensions 0.4 m x 0.03 m, located above the window at a height of 2.2 m above the floor. Exhaust air vent has the same dimensions of the supply vent, located on the wall opposite to the supply vent wall, at a height of 2.5 m above the floor level.

The radiation model selected in the simulation is the surface to surface radiation model (S2S model). The thermal model selected was the mixed model, where radiation and natural convection occur in the current simulation. The room is being heated by the means of the radiator with dimensions and a surface temperature of 1 m x 0.4 m and 50 °C, respectively. It rises over the floor by 0.2 m.

3.3 Results

From the CFD simulations, it was shown that the average temperature of the room is about 18.35°C and the temperature difference between heads and ankles is around 4°C. The temperature contours in the midplane of the room can be seen in Figure 9. To be able to see the differences between a couple of different heater configurations and their effects on temperature and air circulation in the room, they also checked the velocity vectors colored by temperature in the midplane of the room as shown in Figure 9. It was shown that the least amount of recirculation occurred in this case compared to the other configurations that they investigated.

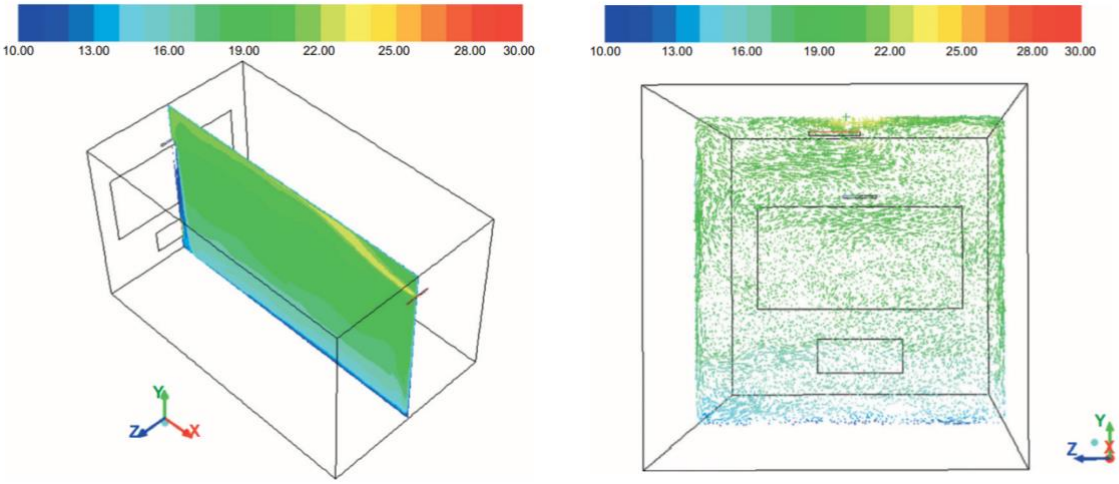

Figure 9. Temperature contours [°C] at z = 1.5 m and b) Velocity vectors at x = 2.5 m [3].

4. CFD Modeling for Ventilation System of a Hospital Room

In another study, Kermani, 2015 [4] performed CFD simulations by using COMSOL 5.1 software to understand airflow pattern and the dynamics of infectious particles due to respiratory diseases such as SARS and Tuberculosis since there are evidence that airborne bacteria may also cause infection due to inhalation of infectious particles.

4.1 Geometry and Conditions

As can be seen in Figure 10, the hospital room model is composed of a wardrobe, a bed, a patient, a doctor, a lamp, an inlet and an outlet. In this study [4], both natural convection and forced ventilation are considered. The ventilation rate is 6 ACH (Air Change per Hour). From the inlets, air enters the room with 20°C temperature and exits from the outlet. Moreover, there are heat sources such as lamp, medical equipment, also doctor and patient with constant heat flux which can be seen in Table 1. All three sides of the room and its base are assumed to thermally isolated, heat transfer between the room and outside occurs just from the ceiling and opposite side of the room from the wardrobe.

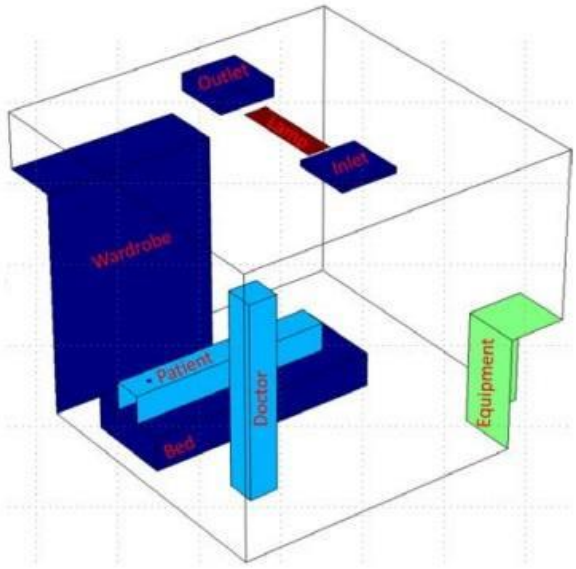

Figure 10. Hospital Room Configuration [4]

Table 1. Temperature or heat fluxes of the heat sources [4]

Body heat flux for doctor and patient	60 W/m^2
Equipment heat flux	100 W/m^2
Lamp heat Flux	200 W/m^2
Inlet temperature	20 °C
Outside temperature	5 °C

In the CFD simulations study [4], coupled Navier-Stokes, continuity and convection-diffusion equations are solved. The k-omega turbulence model with consistent stabilization is used for turbulence modelling. Moreover, the variation of density with temperature is captured with Multiphysics/Non-Isothermal Flow physics in COMSOL. Boundary layer meshes at all boundaries are also used.

4.2 Results

From the results of the CFD simulations [4], in Figure 11, both temperature distributions and velocity vectors adjacent to the doctor and the patient can be seen.

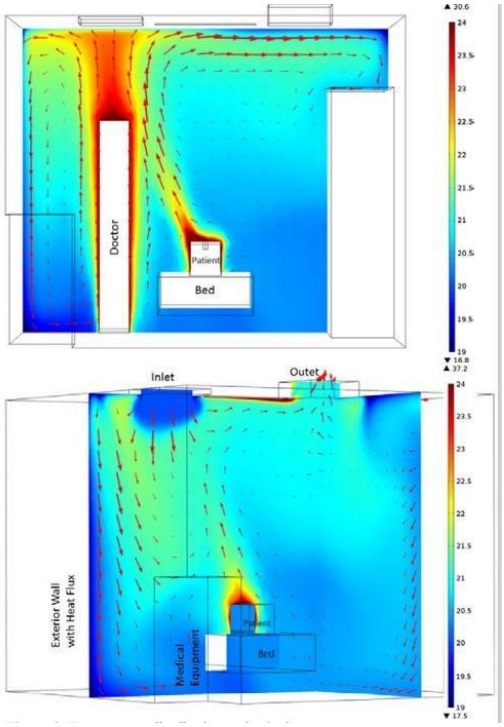

Figure 11. Temperature distribution and velocity vectors inside the hospital room [4]

References

[1] Wibron, E., *CFD modeling of an air-cooled data center,* Master's Thesis in Applied Mechanics, Department of Applied Mechanics, Chalmers University of Technology, Gothenburg, Sweden 2015.
[2] Hassan, N.M.S., Khan, M.M.K., & Rasul, M.G., *Temperature monitoring and CFD analysis of Data Centre*, Procedia Engineering, Vol. 56, pp. 551-559, 2013.
[3] Sobhi, M. & Khalil. E.E., CFD Investigation of Air Flow Patterns and Thermal Comfort in a Room with Diverse Heating Systems, Current Environmental Engineering, 6, 150-158, 2019.
[4] Kermani, A., CFD Modeling for Ventilation System of a Hospital Room, Proceedings of the 2015 COMSOL Conference in Boston, 2015.

Final Results and Discussions, Nov 2022:

1. Ford Otosan test room cooling problem and the status of their CFD analysis capability have been discussed and evaluated.
    - Ford Otosan R&D has a group which uses CFD extensively during their design work. They also have a test group which performs related performance test for different design parts. This test group is having problems with their test rooms in their labs about satisfying and controlling the room conditions such as flow and temperature distribution and control around the test setups during the tests. They are planning to perform CFD simulations for selected test labs to understand the room airflow pattern and temperature distribution conditions during the tests. The test group has limited experience and/or have limited time performing CFD simulations, and they would like to learn and try TRUBA HPC infrastructure with open source CFD software for investigation of their airflow and heat transfer problems in their Powertrain Labs.
    - The drawings of Powertrain Laboratory were obtained from Ford Otosan.
    - Generation of a simple model for the test room geometry is considered first to create the initial CFD simulation models. For this purpose, similar test cases from the literature are reviewed and summarized as above.

2. Learning about Truba's HPC infrastructure:
    - Undergraduate senior students, Bilge Şahin and Celil Sözer from the Dept. of Aerospace Eng. in METU worked on the CFD simulations for test rooms by using SU2 open source CFD flow solver.
    - They have got their accounts on METU RUZGEM HPC Cluster Poyraz and Ulakbim TRUBA HPC Cluster. During this project and study, they learned how to use HPC systems and how to perform CFD simulations.
    - First, the geometry and grid generation studies have been started in the project.
    - This pre-processing step, creation of the basic and suitable test room model, clean geometry and good grid for the computational domain were the difficult steps at the beginning of this study.
    - We prepared the basic training documents so that any new engineer can easily start using TRUBA and perform CFD simulations for this test room model, also any other problem easily.
    - SU2 CFD software installation and trial runs are performed to teach how to use them on TRUBA. The documentations for the trainings and trial runs are prepared.

3. Pre-processing steps: geometry and grid generations
    - The initial CFD steps include the pre-processing steps that require the definition of the geometry, the creation/cleaning of the geometry, and the grid generation for the CFD analysis. These steps usually take most of the time in a CFD analysis, probably at least 1/3 of the time.

- The test room geometry, the technical drawings as shown in Figure 12, and technical specifications are obtained from Ford Otosan.
- Different simple test room models are tried at the beginning as shown in Figure 13 and 15. The geometry of the test room is simplified to have a reasonable inlet outlet and a heating object. Unstructured grids are generated by using Pointwise grid generation software and also Ansys Fluent Meshing grid generation software. Sample unstructured grids are shown in Figure 14 and 16.
- For the test room models, decision of boundary conditions and placing heating sources, configurations and their heating characteristics are the important steps and parameters in input generation for the CFD simulations.

- For the test room 1, the CAD drawings are created by using Rhino are shown in Figure 13. The test room has a length of 10 m, width of 6 m and height of 5 m. The heater is shown in the middle of the room. The heater is drawn with 2 m length, 1.2 m width and 1.5 m height. The inlet and outlet are placed at the two ends of the room. They are placed 3.5 m above the ground and located in the middle in terms of width of the room which leaves 2 m from both edges. Also, the inlet and outlet are stand out from the room walls with 0.2 m. Both the inlet and outlet have dimensions of 2 m width and 0.5 m height.
- By using the described geometry, the surface mesh and volume unstructured grid are generated by using Pointwise software. Initially, the mesh is quite courser yet to get a reasonable solution, surface meshes on heater and inlet/outlet are finer then walls of the room. The surface meshes and volume grid are plotted in Figure 14. Table 2 shows the corresponding triangular and tetrahedral grid cell numbers for the surface meshes and the volume grid.
- For the test room 1 problem, the boundary conditions must be assigned. The inlet and outlet surfaces are taken as openings to the outside so that the most suitable boundary condition is chosen as “Mass Flow Inlet/Outlet” in SU2. The literature review is used in this decision as well as other boundary conditions. To satisfy mass conservation, the mass flow rate through inlet and outlet surfaces must be equal to each other. In literature review, the examples for inlet/oulet are mostly available for air conditioner types, yet in the test room case, the inlet/outlet is not quite determined in the problem. For an initial investigation, the flow speed through an air conditioner which is 0.7 m/s is used to calculate mass flow rate but in further this value can be corrected. Assuming sea-level conditions and the geometry of the inlet/outlet surfaces, the mass flow rate can be calculated by following equation:

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

- Next, the boundary conditions for the heating surfaces must be assigned. In configuration file of SU2, there are multiple types of boundary conditions available. In this case, for heater there are two suitable options such as constant heat flux and constant temperature (isothermal). Again, the conditions are not quite set so that as the boiler assumed to be in the middle

of the room, the ideal working temperature of for example a car engine can be used which is 90°C = 363.15°K.

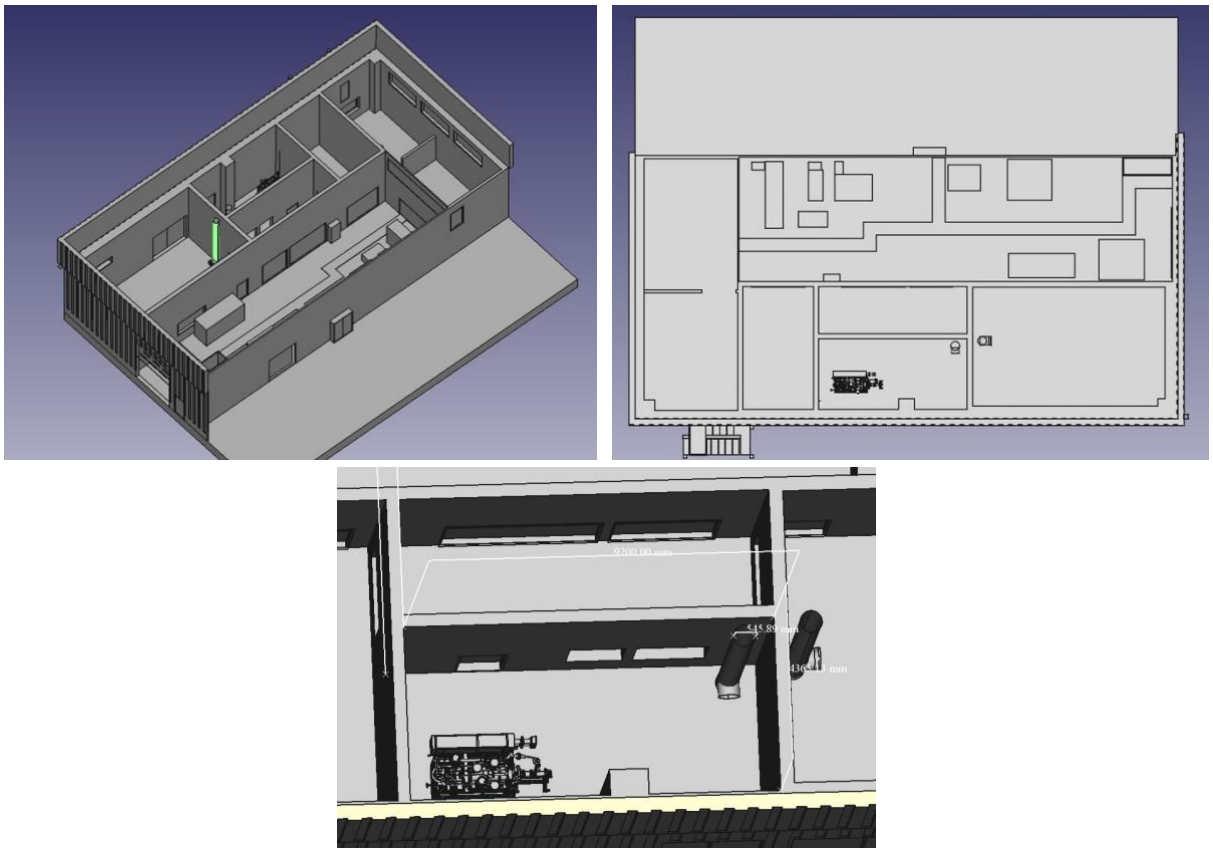

Figure 12. Laboratory geometry and test rooms

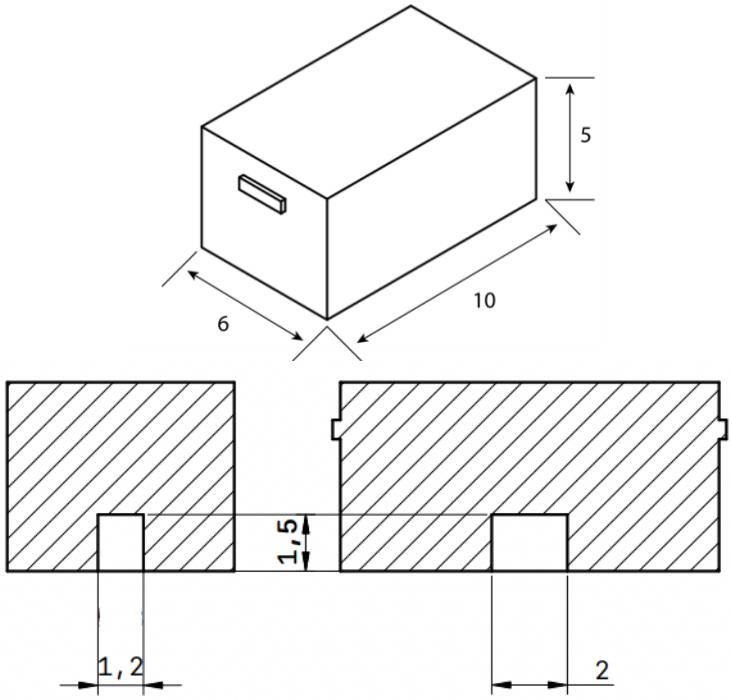

Figure 13. Test Room Model 1: Room and heater dimensions

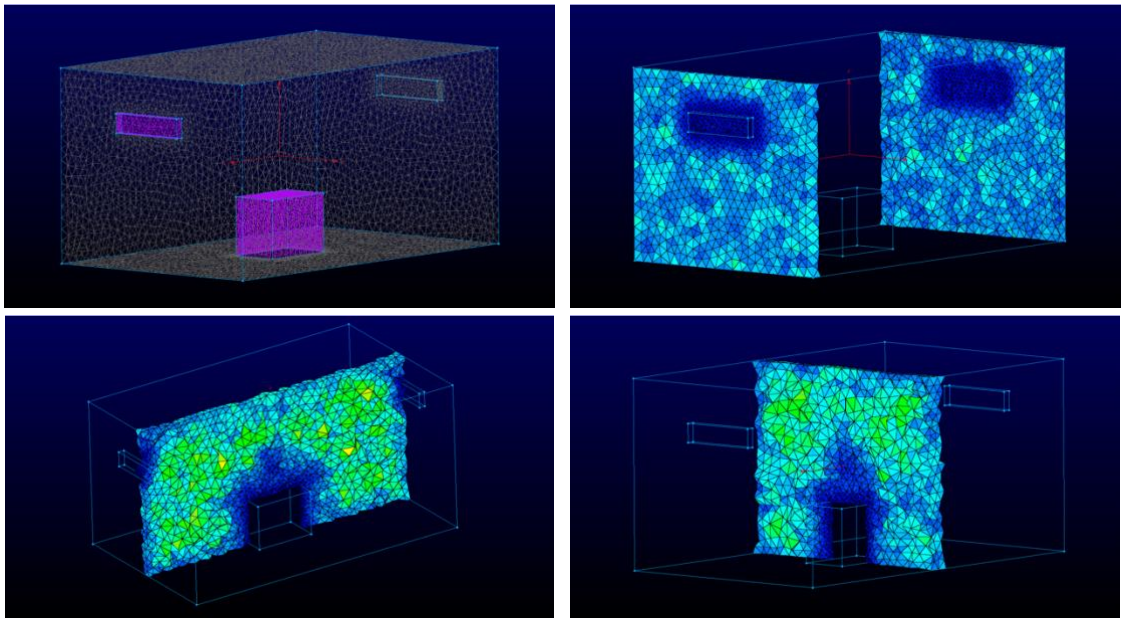

Figure 14. Computational domain and unstructured grid for the test room 1 by using Pointwise. Volume mesh slices at the ends of the room and at the middle of the room.

Table 2: Test room 1: surface and volume grid size

Name	# of Surfaces	# of Triangles
Walls	14	12006
Heater	5	3724
Inlet	1	302
Outlet	1	302

Name	# of Points	# of Cells
Volume	32,494	170,173

- For the test room model 2 as shown in Figure 15, a room with heat generator is modelled in CATIA.
- Mesh generation has been done in Fluent-Meshing. Initial coarse unstructured grid as shown in Figure 16 has 4 million cells with a max skewness of 0.74 and min orthogonality of 0.26.
- The CFD model for the test room has an inlet with 10 m/s and pressure outlet 101325 Pa as boundary conditions. A heat generator which is producing heat is applied with a temperature of 40°C and 100 W/m^2K is applied as convection heat condition.

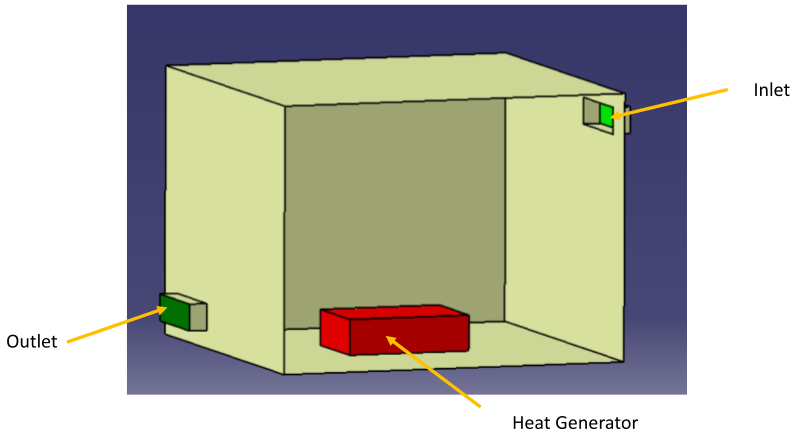

Figure 15. Test Room Model 2.

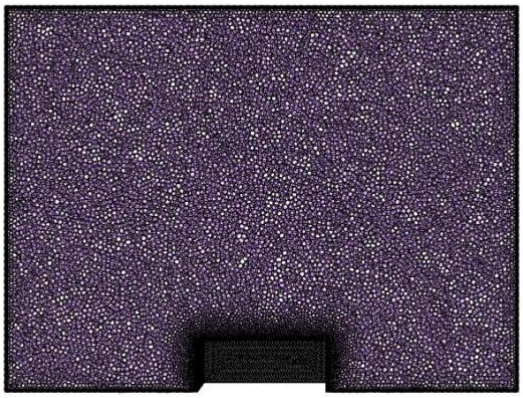

Figure 16. Computational domain and unstructured grid for test room 2 using Ansys Fluent.

4. Test Case CFD Simulations: steady-state/unsteady runs
    - The CFD test case simulations continue for different test room models and conditions.
    - The CFD trial runs are performed on Truba's HPC infrastructure and Poyraz HPC Cluster, and the sample input files for SU2 for the test room problem are prepared, and the documentation for trainings are prepared.

5. Final Discussions and Conclusions:
    - Ford Otosan group and students worked in this test case study have learned and obtained a methodology/procedure and training documentations for running their CFD simulations on TRUBA HPC clusters by using open source CFD software.
    - In addition, we will be able to continue to help Powertrain Laboratory for their detailed analysis and possible solutions for their test room conditions in the future.
    - The CFD models created will be also useful for similar indoor environment and heating/cooling studies for small SMEs and for more training activities.

6. General comments on test case studies:
    - During the test case studies for different CFD simulations, the most difficulty is to work on each problem separately for each company because the simulation models and requirements depend on the specific problem itself.
    - Another difficulty is that the time allocated for the test cases by the engineers from small groups or companies is usually very limited. Once the training materials are ready, as these materials are now prepared during these CFD test cases, it will be easy to perform training activities to different groups and companies. However, the follow up of the simulations and activities can become difficult. The use of undergraduate and graduate students allows to train the students who can go to the industry when they graduate and also allows to perform more detailed simulation for these specific test case problems. These detailed simulations become a guideline to the company engineers as well.