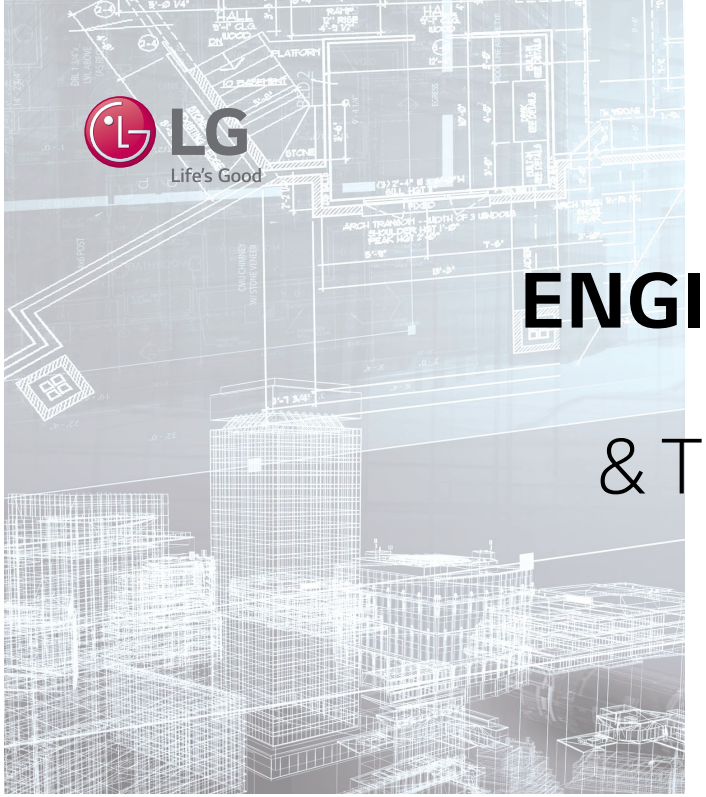


ENGINEERING TOOLS & TECHNICAL SOLUTION

01

LG PROGRAMS

- LATS CAD
- LATS HVAC
- LATS ENERGY
- LATS REVIT

02

ENERGY MODELING

- eQUEST®
- EnergyPlus™
- TRNSYS

03

CFD ANALYSIS

· FLUENT

CFD Analysis is applied in areas of estimating: indoor airflow and temperature distribution while operating VRF products, outdoor airflow distribution. LG HQ engineers support diverse installation environment simulation at Design development and construction stage through Fluent, a commonly used CFD analysis program.

By running a simulation before construction, engineers estimate possible issues and find optimal solutions of malfunction that could occur after construction.

04

NOISE ANALYSIS

- LMS Raynoise
- Rayacoustics

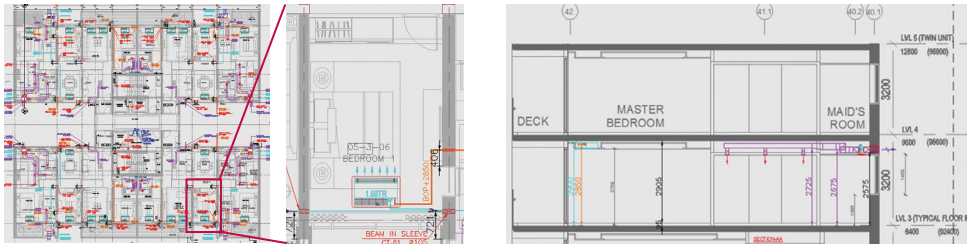
I-1. Indoor Unit: Optimal Capacity Decision Through Air Flow Analysis

HOTEL

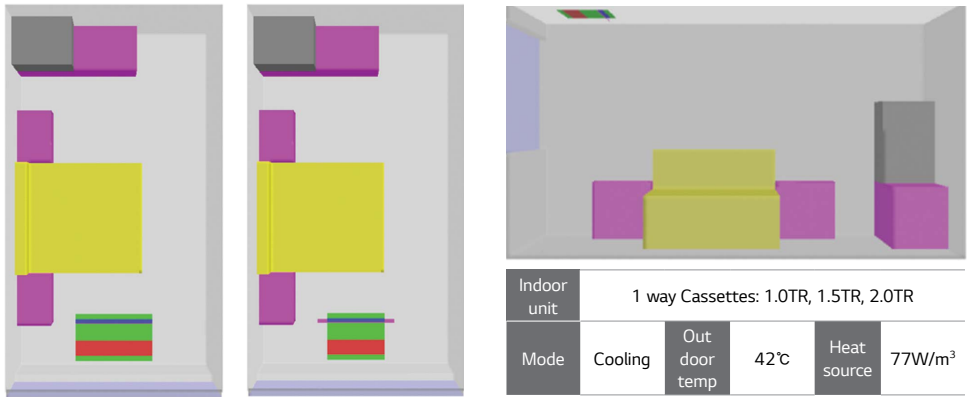
Project Scope

The project is to decide optimal capacity to hotel room in India. The CFD analysis simulation was conducted to the air flow and temperature in case of the indoor units operation respectively. Based on this simulation result, LG advanced the opinion on the optimal capacity to this project.

CAD Drawing



3D Modeling



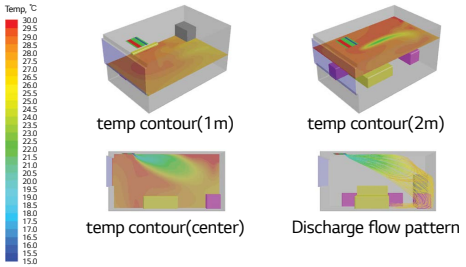
Indoor unit	1 way Cassettes: 1.0TR, 1.5TR, 2.0TR				
Mode	Cooling	Out door temp	42℃	Heat source	77W/m³

Case 1

- Unit: 1 way Cassette, 1.0 TR
- Condition : Discharge air flow 10CMM (2.66m/s), 12℃

Results

The capacity is expected to be less than actual load, resulting in lack of cooling performance

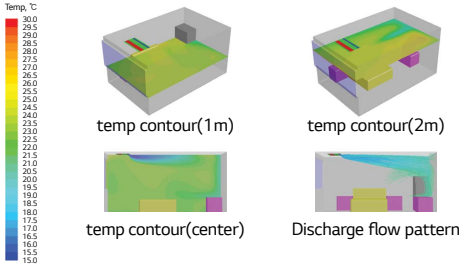


Case 2

- Unit: 1 way Cassette, 1.5 TR
- Condition : Discharge air flow 13.3CMM (2.64m/s), 12℃

Results

Due to bigger capacity than 1.0TR, the average indoor temperature is expected to be 4.1℃ lower

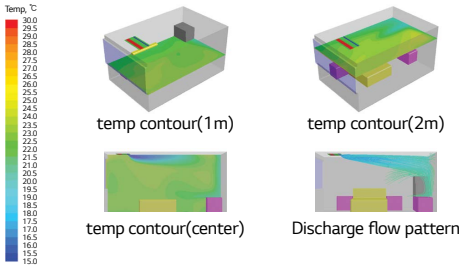


Case 3

- Unit: 1 way Cassette, 2.0 TR
- Condition : Discharge air flow 14.6CMM (2.9m/s), 12℃

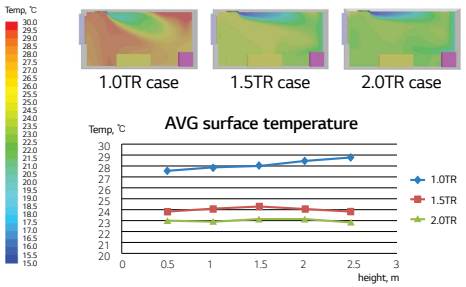
Results

Due to bigger capacity than 1.5TR, the average indoor temperature is expected to be 1℃ lower



Summary

LG advanced an opinion on the optimal capacity. Above the CFD analysis result, the temperature distribution of case 1 is not good in comparison with the case 2 & case 3. Case 2 is similar to case 3 in temperature distribution, and discharge air flow velocity of case 2 is more mild than case 3. So, in LG's opinion, case 2 is considered as the optimal capacity.



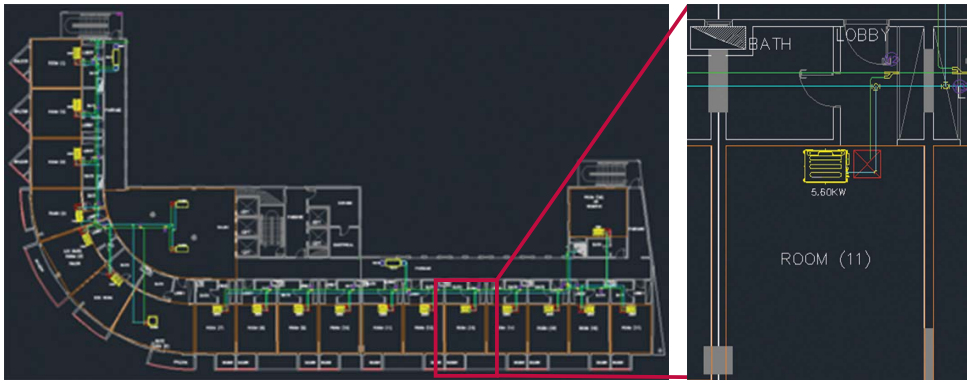
I-2. Indoor Unit: Different Types

HOTEL

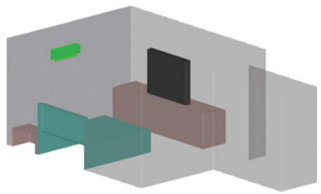
Project Scope

The project is to select indoor units to hotel in Congo. The CFD analysis simulation was conducted to the air flow and temperature in case of the indoor units operation, wall mounted type and concealed ceiling duct type respectively. Based on this simulation result, LG proposed the solution which indoor unit type is more suitable for this site.

CAD Drawing

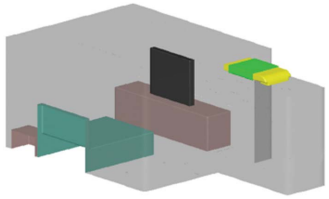


Case 1 – Residential Wall Mounted



ISO view

Case 2 – Ceiling Concealed Duct



ISO view

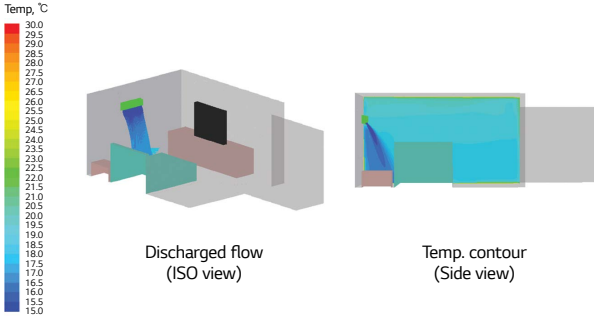
A wall mount type is installed in the outside wall and a ceiling concealed duct type is installed in the ceiling of inside door.

CFD analysis simulation to room temperature and air distribution, comparison to each case.

Case 1 Result - Residential Wall Mounted

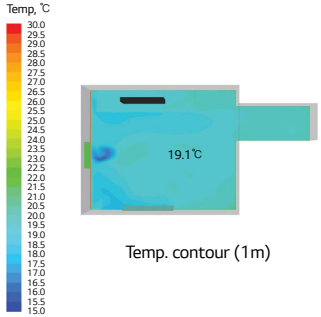
Results

Cooling capacity is expected to be enough. But due to wall mounted type, the reach of airflow may be short, resulting in uneven room temperature, and cold draft to guests.



Discharged flow (ISO view)

Temp. contour (Side view)

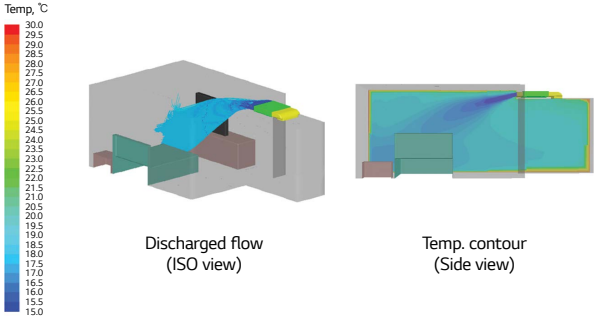


Temp. contour (1m)

Case 2 Result – Ceiling Concealed Duct

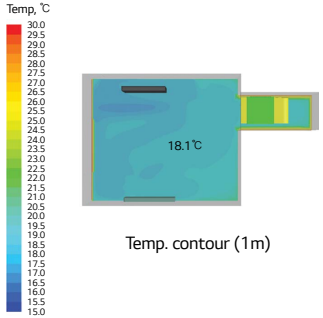
Results

Cooling capacity is expected to be enough, and room temperature is expected to be even compared to other indoor types. Also, cold draft does not occur to quests.



Discharged flow (ISO view)

Temp. contour (Side view)



Temp. contour (1m)

Summary

In case of ceiling concealed duct type, it preforms better distribution air flow and temperature than wall mounted type case.
If the discharge temperature is same, the air discharge speed of wall mounted(2.5m/s) is faster than the ceiling concealed duct type(1.4m/s). So the ceiling concealed duct type would be more pleasant to feel.

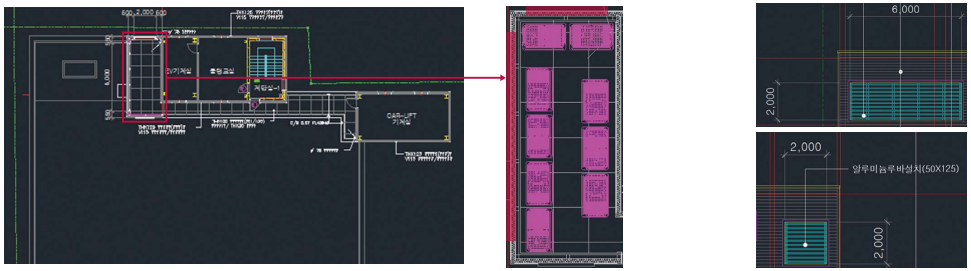
II-1. Outdoor Unit: Different Installation Heights

OFFICE

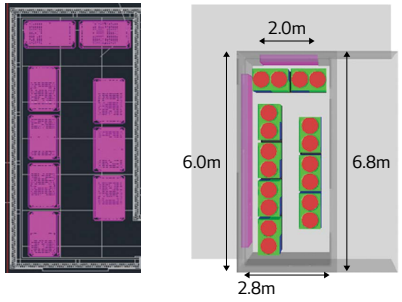
Project Scope

Outdoor units installation space was surrounded by the 4m height wall. There were the 2m height louver for air circulation in two walls of them. The first purpose of this CFD analysis simulation was checking normal operation of this site. But, with CFD analysis simulation, normal operation of this site was impossible. So LG proposed a solution. It is possible to air circulation under the outdoor units with lifting the installation position of outdoor units up.

CAD Drawing

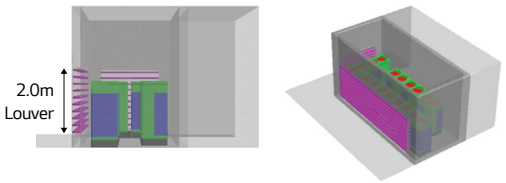


3D Modeling

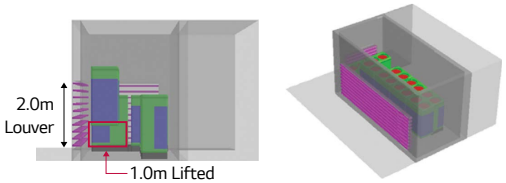


Top view

Case 1 – Installation without Additional Height

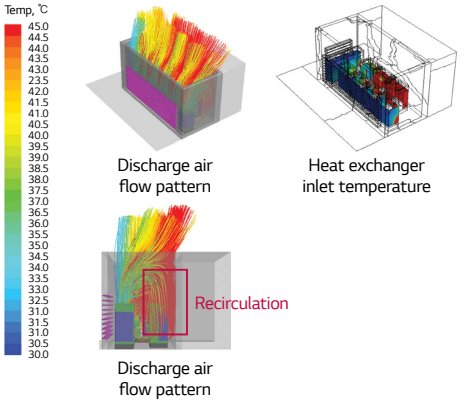


Case 2 – Installation with Lifted Outdoor (1m)



Case 1 Result – Installation without Additional Height

- Increase of heat exchanger inlet temperature occurs to the outdoor units located inside due to air flow interference
- Increase until 52℃ of the heat exchanger inlet temperature (17℃ higher than ambient temperature)



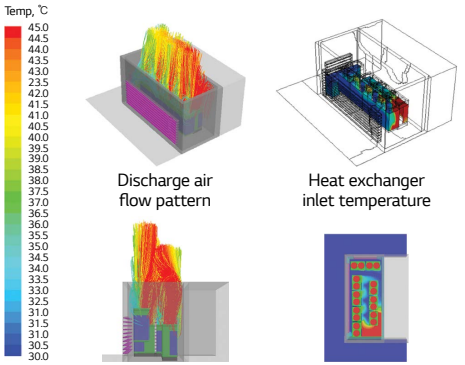
Discharge air flow pattern

Heat exchanger inlet temperature

Discharge air flow pattern

Case 2 Result – Installation with Lifted Outdoor (1m)

- By lifting outdoor units located outside up to 1m, ambient air is able to flow to the inside of outdoor units room
- Decrease of recirculation of discharge air, possible to normal operation
- The average heat exchanger inlet temperature is 42℃(7℃ higher than ambient temperature)



Discharge air flow pattern

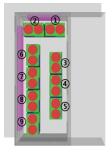
Heat exchanger inlet temperature

Discharge air flow pattern

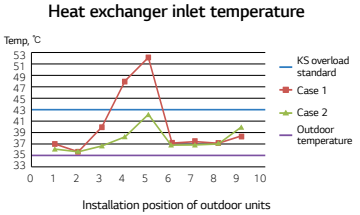
Temperature distribution of discharge air flow (1.3m height)

Summary

LG proposed a solution to the problem of this project like below. This enables ambient air to flow under the outdoor units (⑥~⑨) with lifting the installation position of outdoor units (⑥~⑨) up. As the result, the average heat exchanger inlet temperature decrease to 42℃.(10℃ lower than case 1)



Installation position of outdoor units



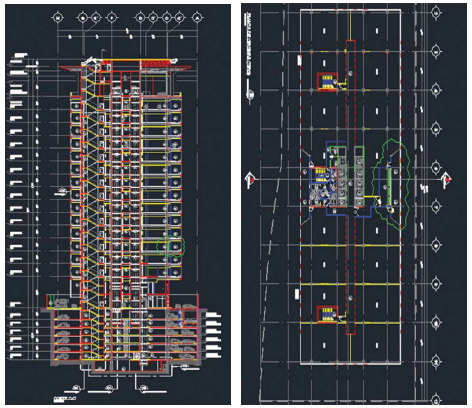
II-2. Outdoor Unit: Different Types

OFFICE

Project Scope

In Mexico, the outdoor unit was installed in the indoor space of office building. The original proposed model, LG Multi V S, a side-discharge model, could not operate because of recirculation of outdoor unit's discharge air. LG conducted the CFD analysis simulation to solve the problem and proposed the solution to the client.

CAD Drawing

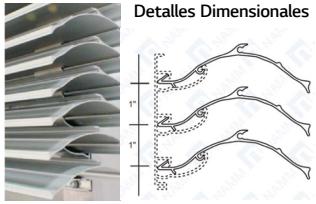


Elevation view

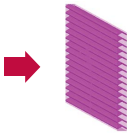
Plan view

3D Modeling

Ambient temperature 31℃

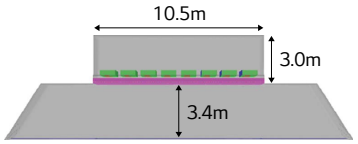


HLAO Louver

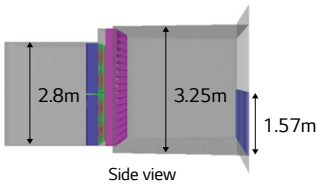


3D modeling

Case 1 – Multi V S 16ea



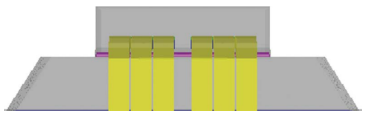
Top view



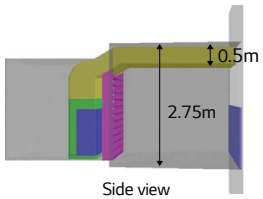
Side view

Case 2 – Multi V IV

To have a higher space, reduce duct size and raise installation height compared to drawing



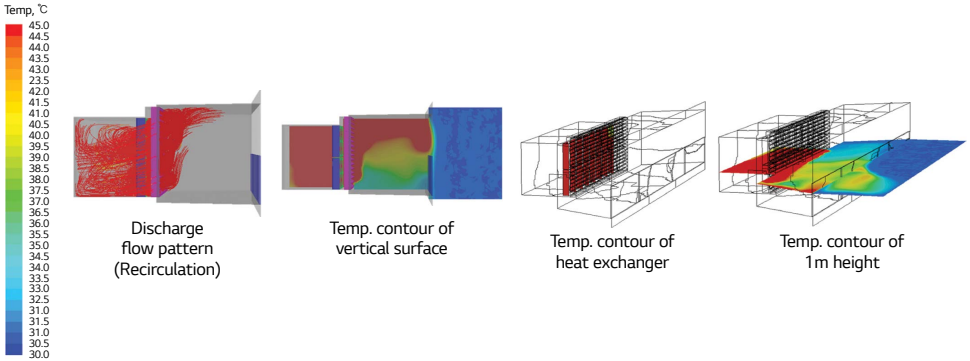
Top view



Side view

Case 1 Result – Multi V S

- Unit : Multi V S
- It's NOT possible to operate outdoor units due to hot air recirculation.
- Outdoor units will shut down due to high suction temperature.



Discharge flow pattern (Recirculation)

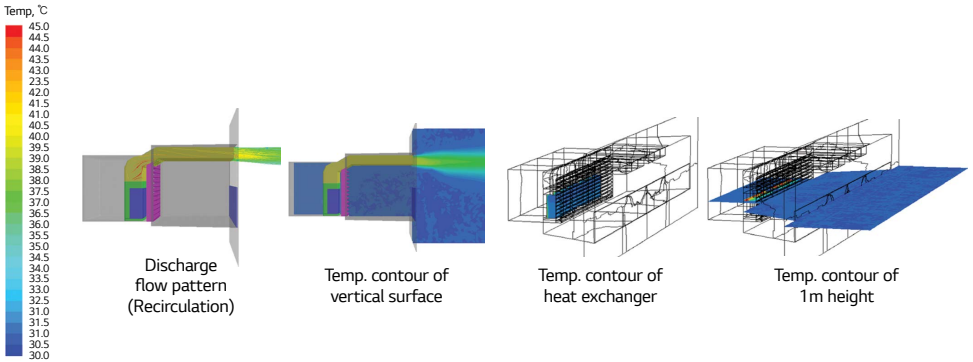
Temp. contour of vertical surface

Temp. contour of heat exchanger

Temp. contour of 1m height

Case 2 Result – Multi V IV

- Unit : Multi V IV
- It's possible to operate outdoor units normally.
- 0.5m height duct enables the residents to make use of the extra balcony space.



Discharge flow pattern (Recirculation)

Temp. contour of vertical surface

Temp. contour of heat exchanger

Temp. contour of 1m height

Summary

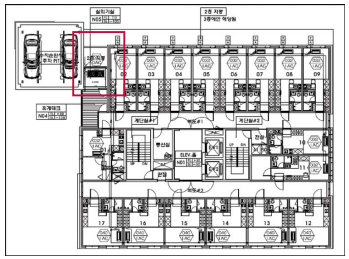
Case 1, the Multi V S, a side discharge model could not operate because of discharge air recirculation of outdoor unit in the outdoor unit room(balcony). So LG proposed the solution to install Multi V IV, a top-discharge model. By adopting a 0.5m height duct to Multi V IV discharge fan, it could eliminate the discharge air from outdoor unit. Through selecting the Multi V IV, It became possible to operate in normal condition and let the residents use the extra outdoor unit room(balcony).

II-3. Outdoor Unit: Different Louver Installations

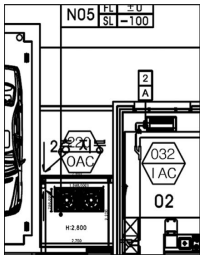
RESIDENTIAL

Project Scope

This project is a residential building in Korea. The outdoor unit installation space is in the building, its facade is blocked with z-louver which has bended tip. So it might be normal operation impossible. With CFD analysis simulation, LG checked the operation and offered a solution to solve the problem.

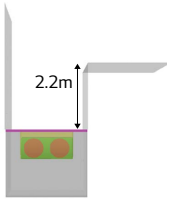


Plane view of cooling and heating for 3rd ~ 13th floor

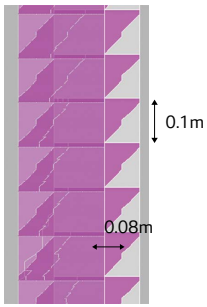


Outdoor unit installation space

3D Modeling

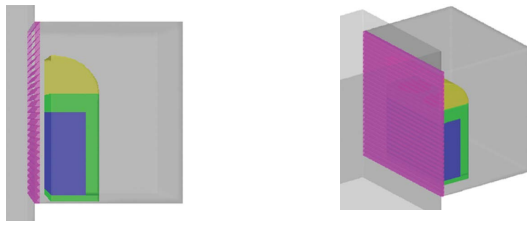


Top view

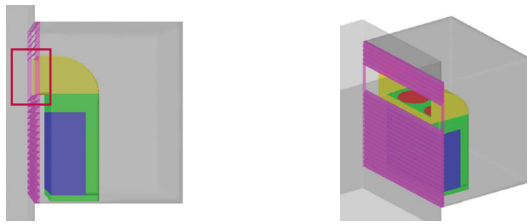


Z-louver modeling

Case 1 – Conventional Design

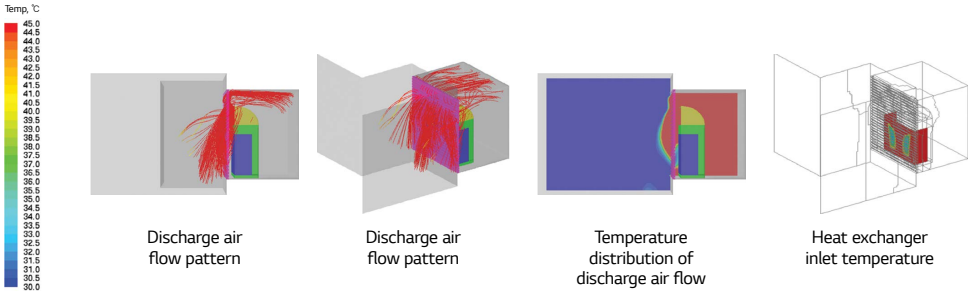


Case 2 – Air-guide Extension and Removal of Z-Louver



Case 1 Result – Conventional Design

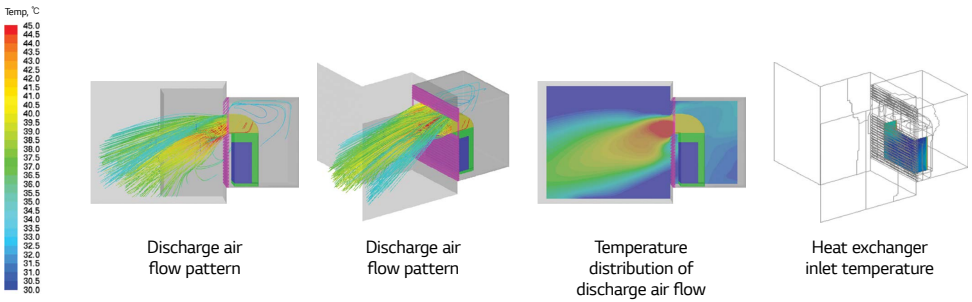
- Recirculation of discharge air occurs in the outdoor unit room due to air flow interference of z-louver
- Room temperature goes over 60℃ in outdoor unit room, making normal operation impossible
- About 84% of discharge air flow volume in comparison with normal operation condition



Discharge air flow pattern | Discharge air flow pattern | Temperature distribution of discharge air flow | Heat exchanger inlet temperature

Case 2 Result– Air-guide Extension and Removal of Z-Louver

- In case of z-louver removal at discharge area, no recirculation of discharge air occurred in outdoor unit room.
- Also, heat exchanger inlet temperature(36.3℃) is in the normal condition.



Discharge air flow pattern | Discharge air flow pattern | Temperature distribution of discharge air flow | Heat exchanger inlet temperature

Summary

The issue in this project was the fact that in conventional installation, normal operation would be impossible. The reasons are abnormal outdoor unit room temperature and low discharge air flow volume due to air flow interference of z-louver. So LG proposed the solution like below: 1. Extend air guide to the outside wall, and 2. remove z-louver of discharge area. As a result recirculation of discharge air does not occur, and heat exchanger inlet temperature became normal(36.3℃).