

# Make Your MEP Design Better using Autodesk CFD Simulation

Dr. Munirajulu. M

Chief Engineering Manager, MEP-CFD

L&T Construction, Larsen & Toubro Limited, Chennai, India





## About the speaker

### Dr. Munirajulu. M

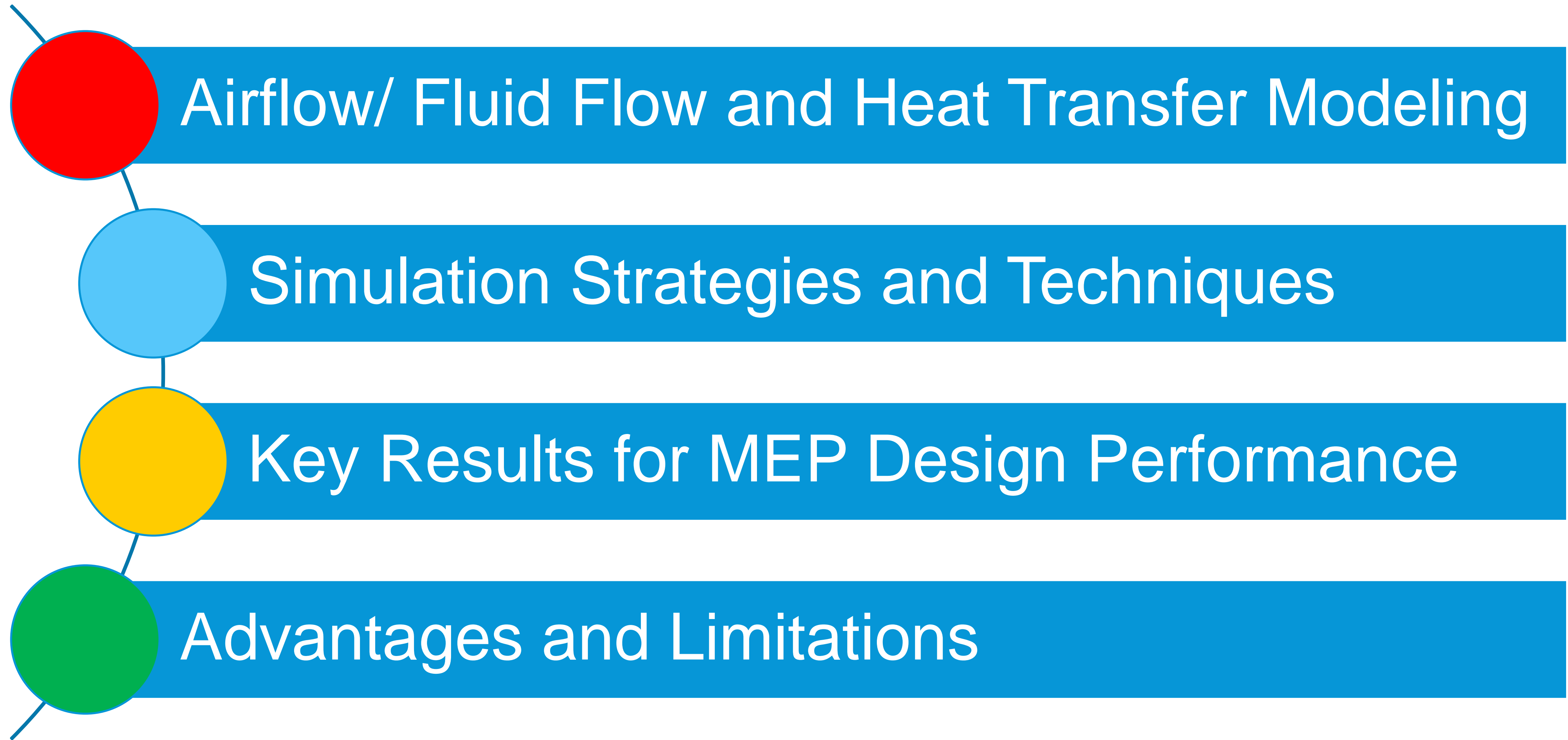
22+ years industry experience in CFD analysis

Speaker at AU 2017, Las Vegas

Using Autodesk CFD Simulation for MEP design

Focus areas: Data Center Cooling, Basement Car  
Park Ventilation, DG room ventilation, Smoke  
simulation in buildings

# Today's Focus Points





# Airflow/ Fluid Flow and Heat Transfer Modeling



# Airflow and heat transfer modeling stages

## STAGE A

### CAD MODEL AND MATERIALS

- Keep it simple
- Extend inlets and outlets

## STAGE B

### BOUNDARY CONDITIONS

- Flow rate and pressure
- Heat generation, temperature

## STAGE C

### MESHING

- 4-5 elements on inlet and outlet extensions
- 4-5 elements uniform meshing on internal fans, resistance materials.

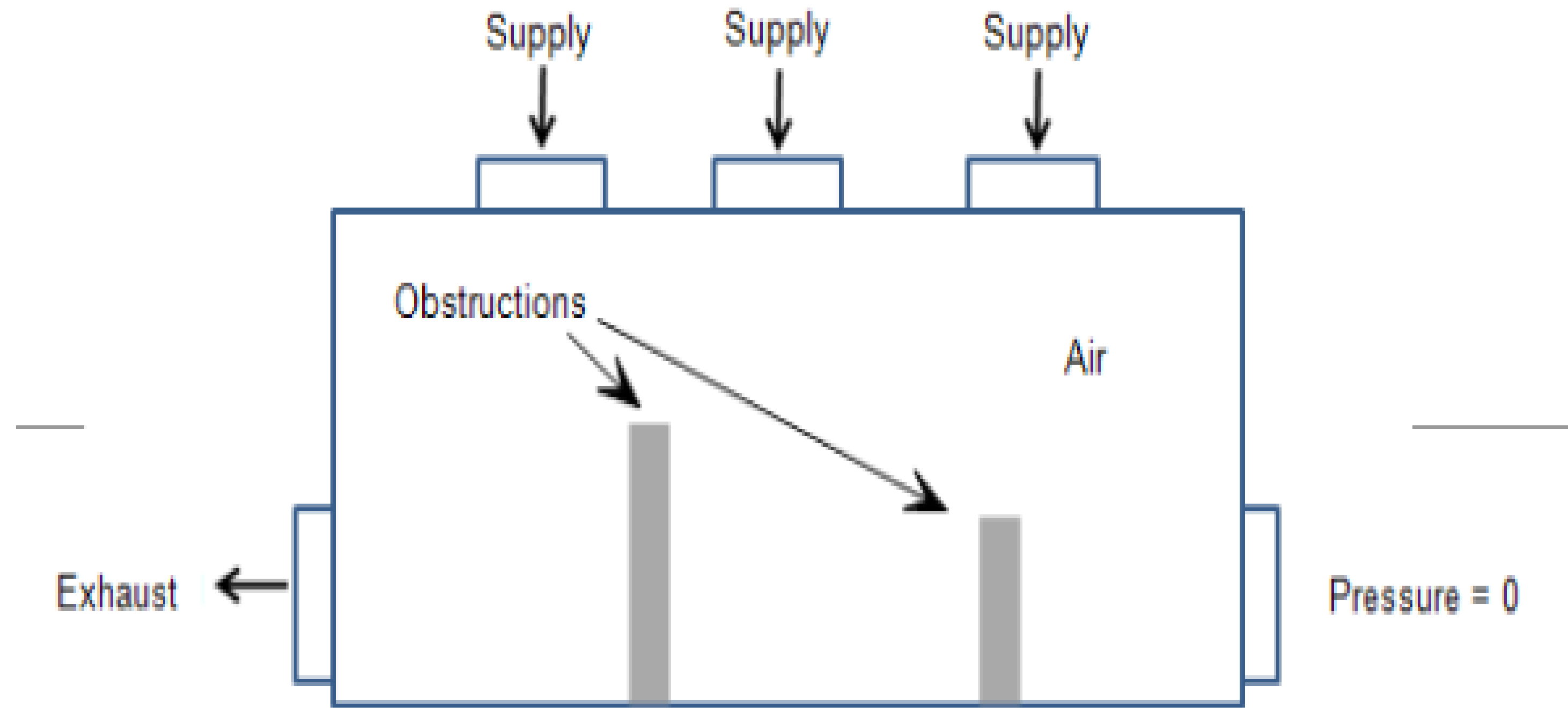
## STAGE D

### SOLVING AND RESULTS

- Physics
- Steady / Transient solution
- Coupled and de-coupled solution
- Results: Velocity, Temperature, Smoke Visibility

# Airflow/ Fluid flow modeling

- Create air region
- Include internal obstructions.
- Include vents, diffusers, fans etc. for supply.
- Include registers, vents, fans etc. for exhaust



<https://knowledge.autodesk.com/search?search=mechanical%20ventilation&p=SCDSE&sort=score>

**Tips:** Extending Inlet and Outlets to Help Avoid Divergence, Improve Mass Balance and Aid Accuracy

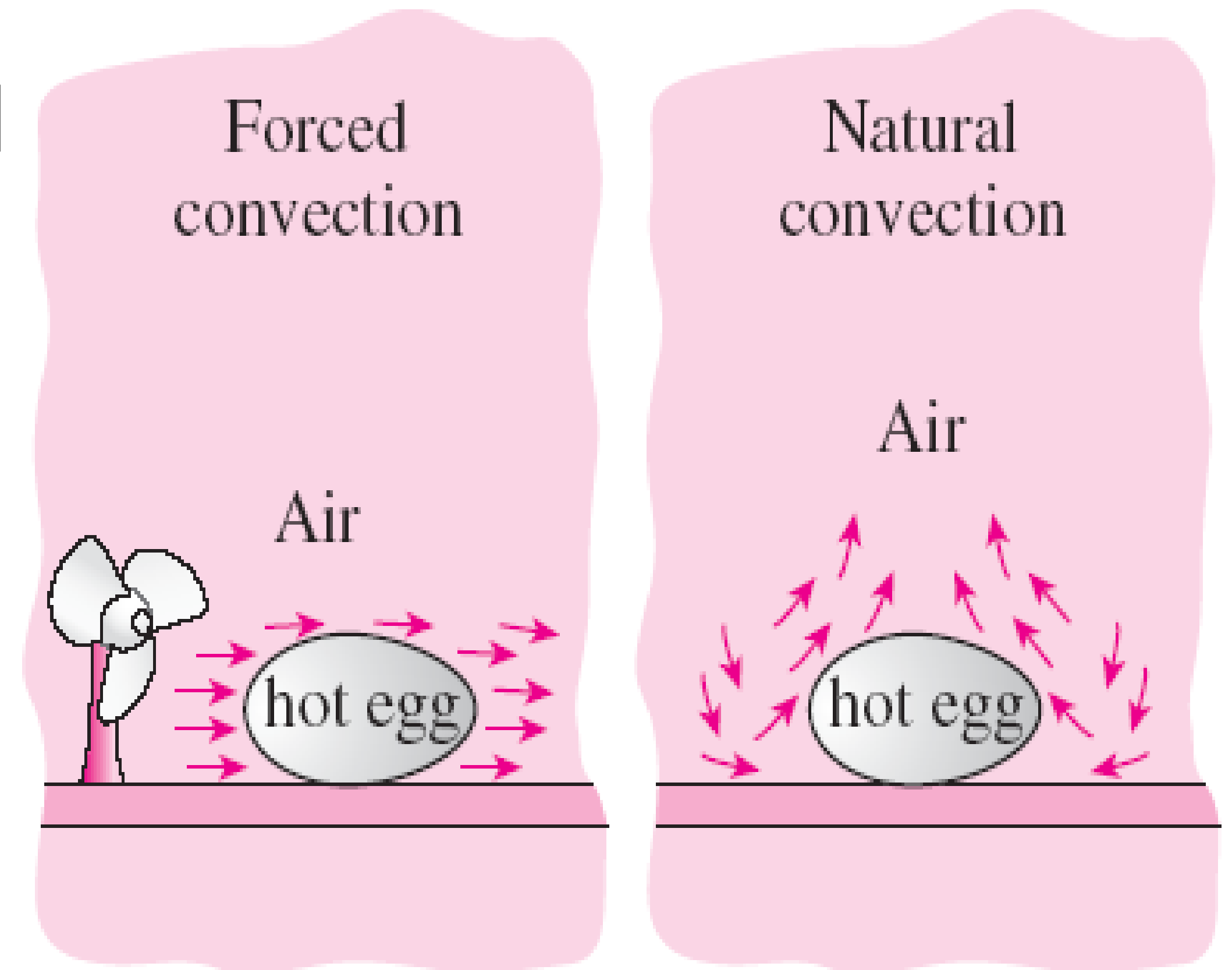
# Heat Transfer modeling

## Forced Convection

- Flow and heat transfer - decoupled
- Solve flow first and then thermal distribution.

## Natural Convection

- Flow and heat transfer - coupled
- Buoyancy-driven flow



<http://www.mhhe.com/engcs/mech/cengel/notes/ConvectionHeatTransfer.html>



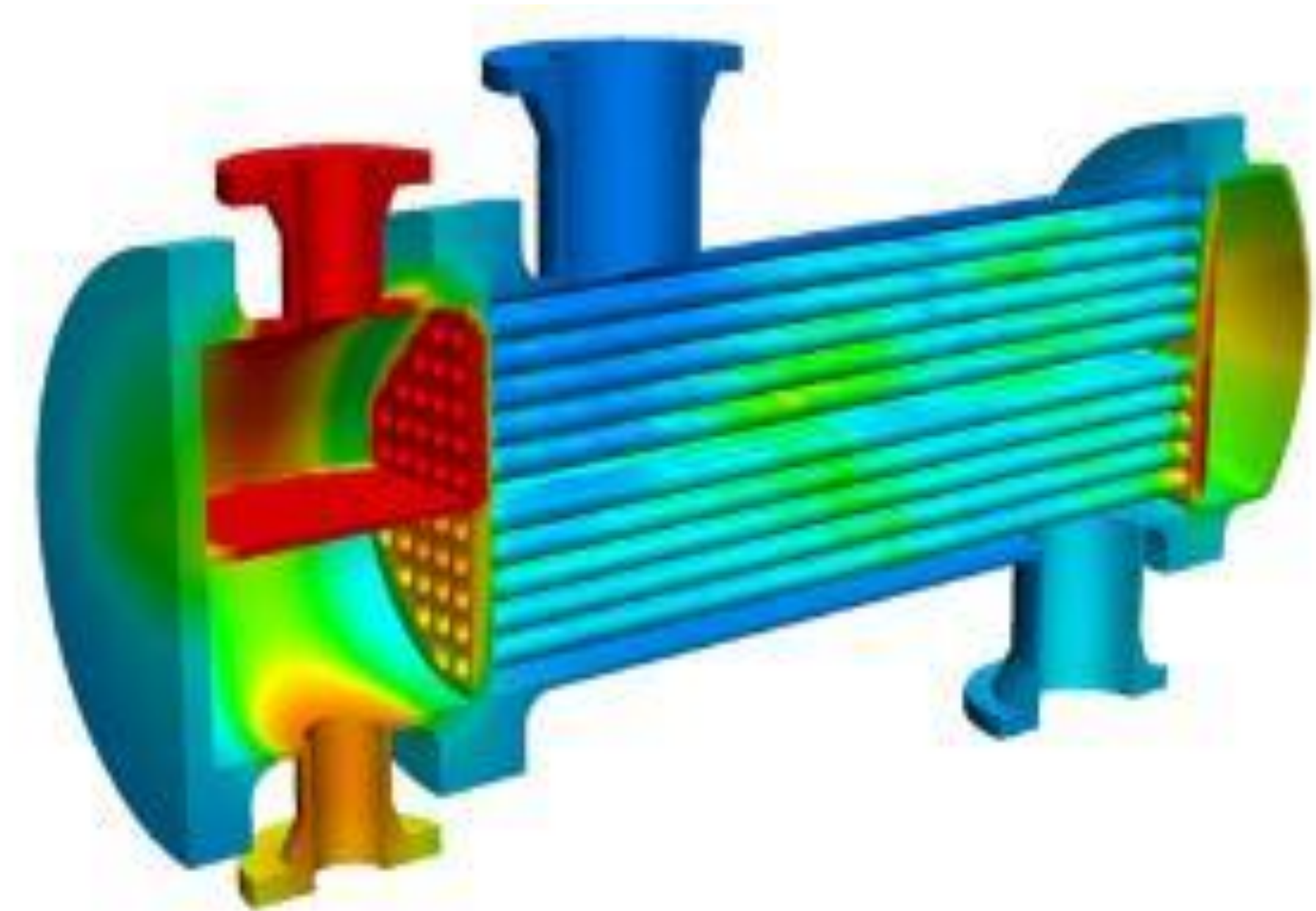
# Heat Transfer modeling



Source: 🔥 Lighting of the Olympic cauldron  
#PyeongChang2018 #Olympics" Twitter.com

## Mixed convection

- Natural and forced convection.
- Appreciable buoyancy effects



Source: IMAGINiT Technologies blog

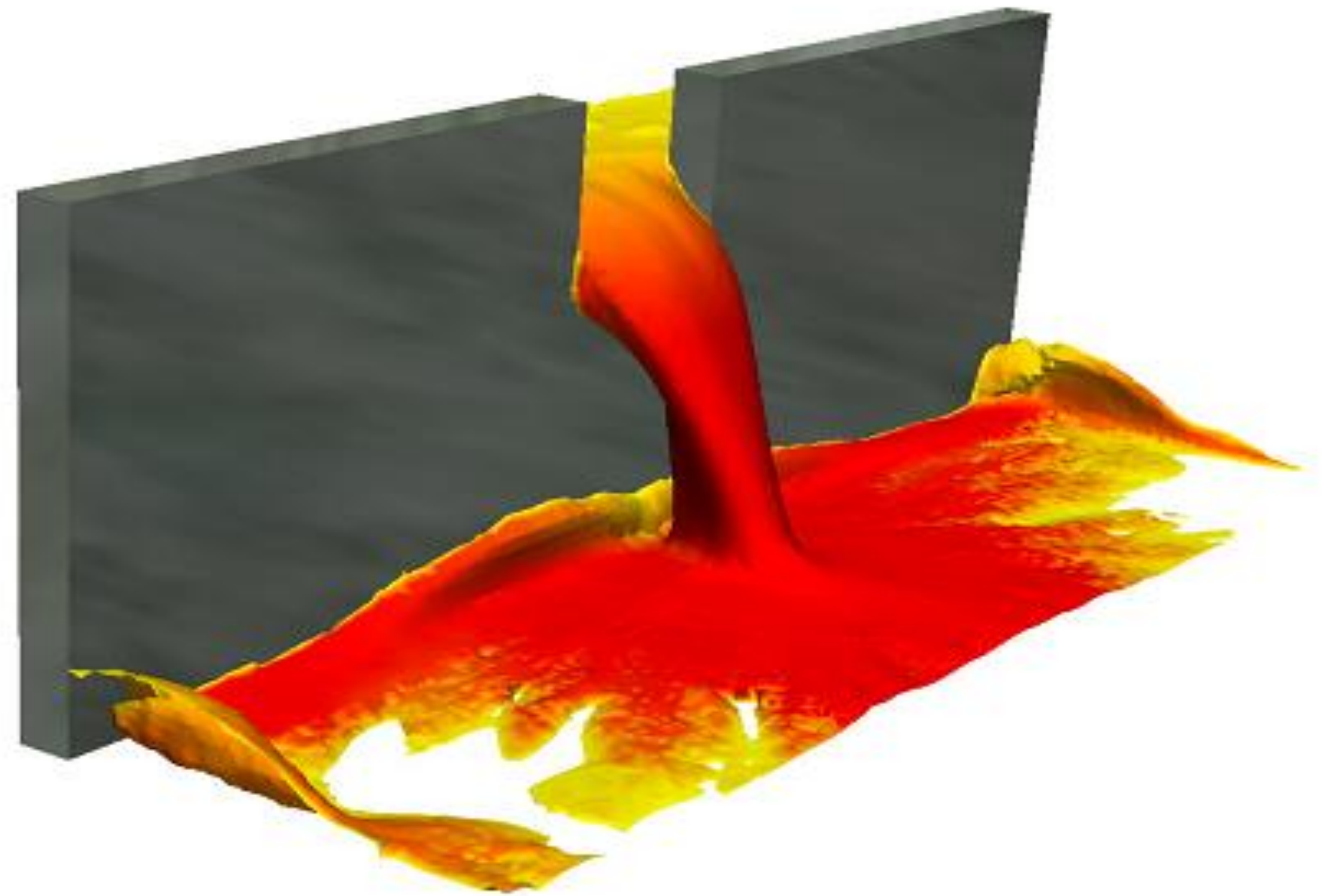
## Conjugate heat transfer

- Solid material conduction
- Fluid convection



# Free Surface modeling

- Flow phenomena such as waves, sloshing, and spilling.
- Rain water flow on air port roofs / water drains



Source:  
AUTODESK KNOWLEDGE NETWORK

# Simulation Strategies and Techniques

## DG Room Ventilation analysis





# Why CFD simulation for DG room?

## Issue:

- DG sets tripping at more than 50% rated load.
- room temperature rising sharply.

## Goal of CFD simulation:

- air flow and temperature distribution
- recommend solution

## Design performance criteria:

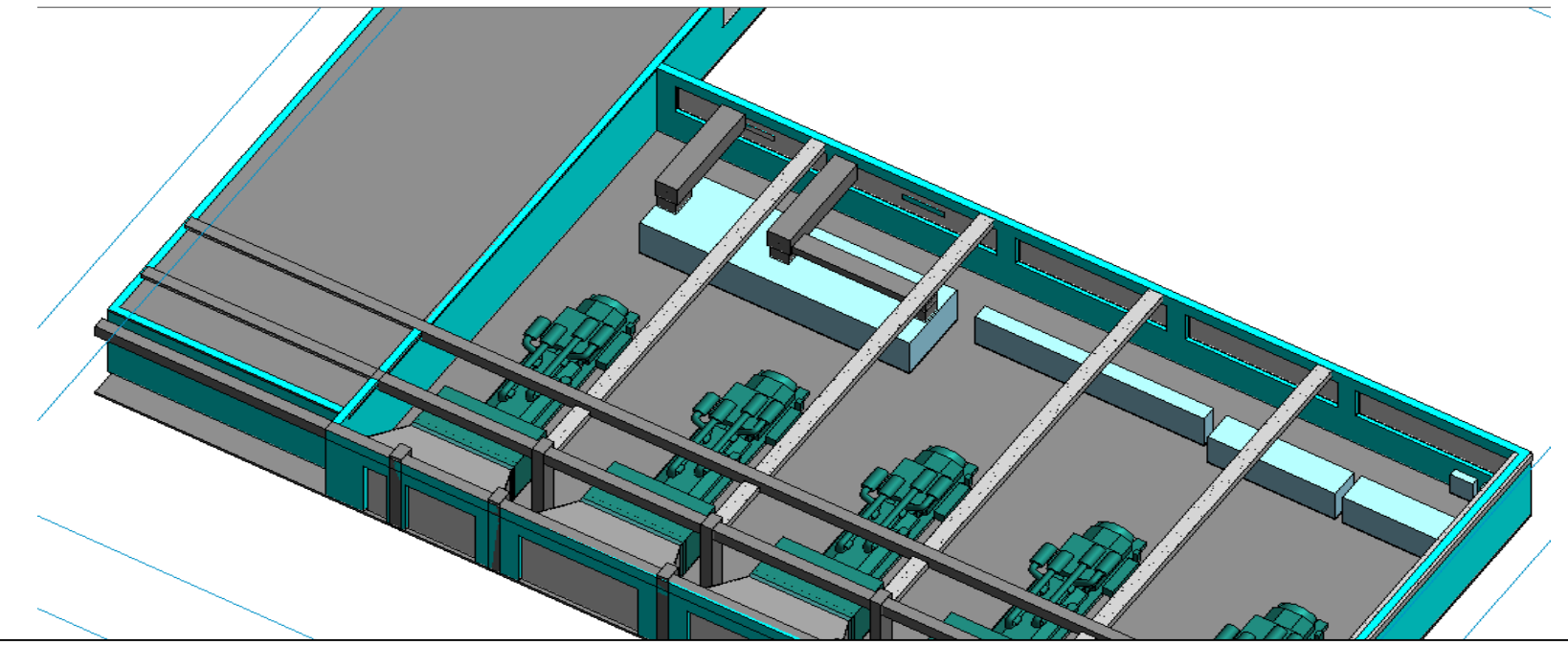
- air temperature  $< 50^{\circ}\text{C}$  around engine and  $45^{\circ}\text{C}$  around alternator
- bottom to top air flow



# CAD Model

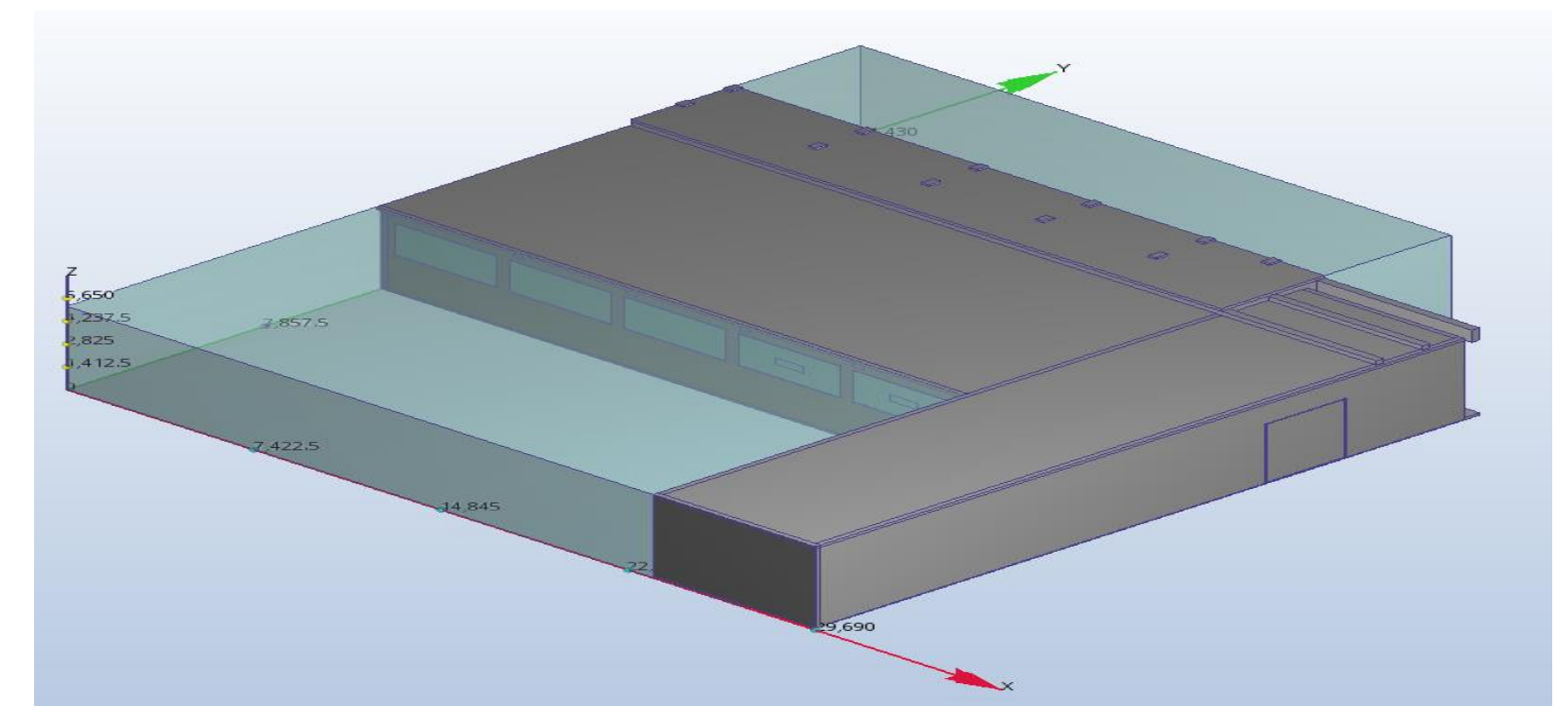
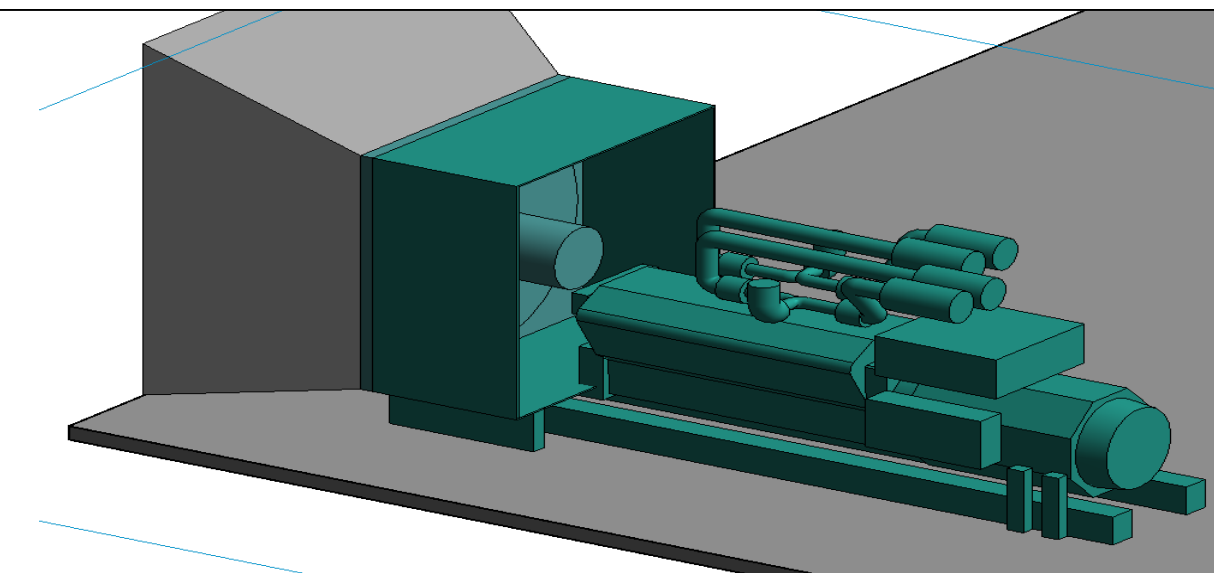
## Create CAD model in Revit.

- Simplify geometrical details
- Add inlet and outlet extensions



DG rating = 2000 KVA (5 nos.)

Room size = 23.25m L X 13.675m W X 5 m H

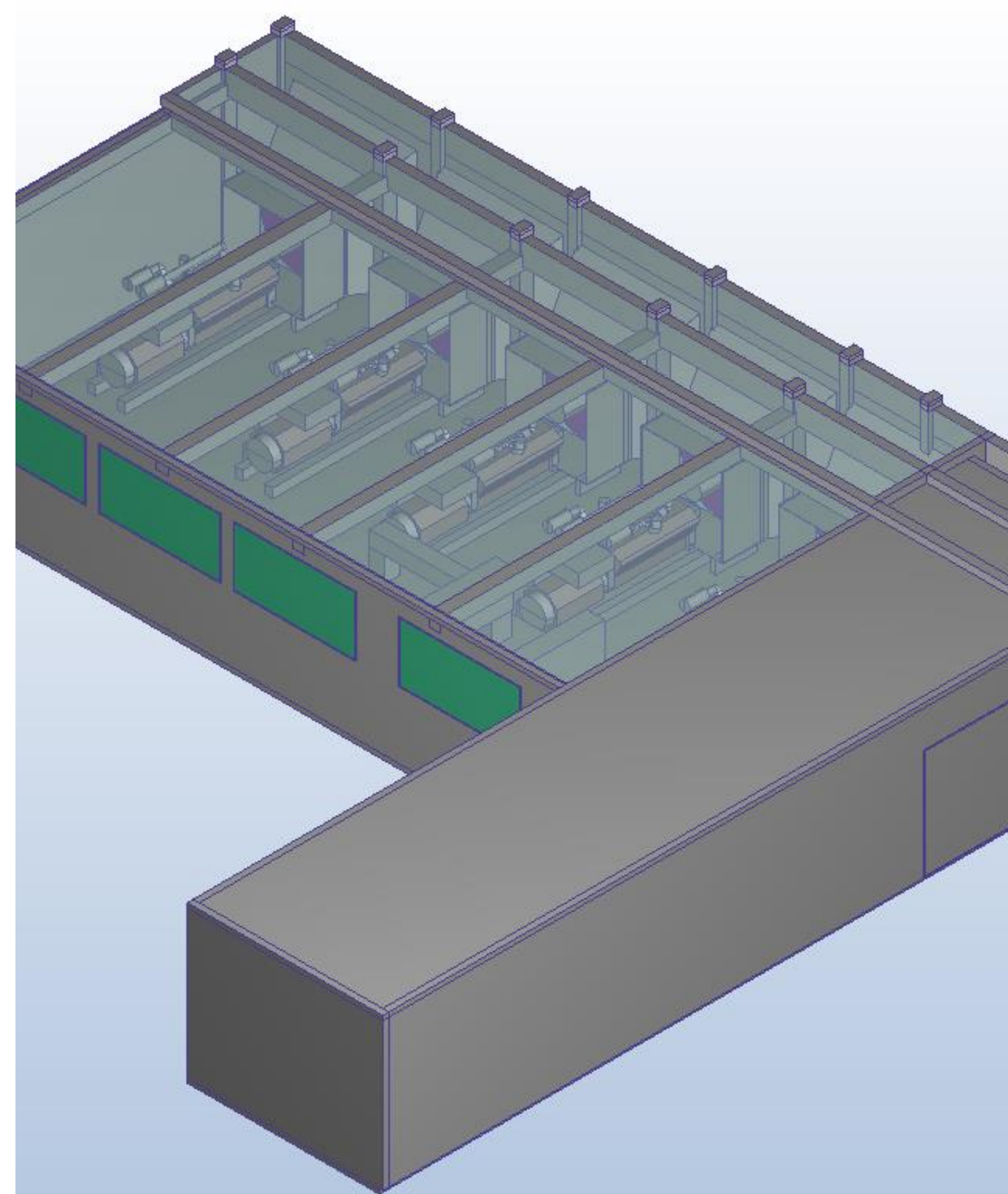
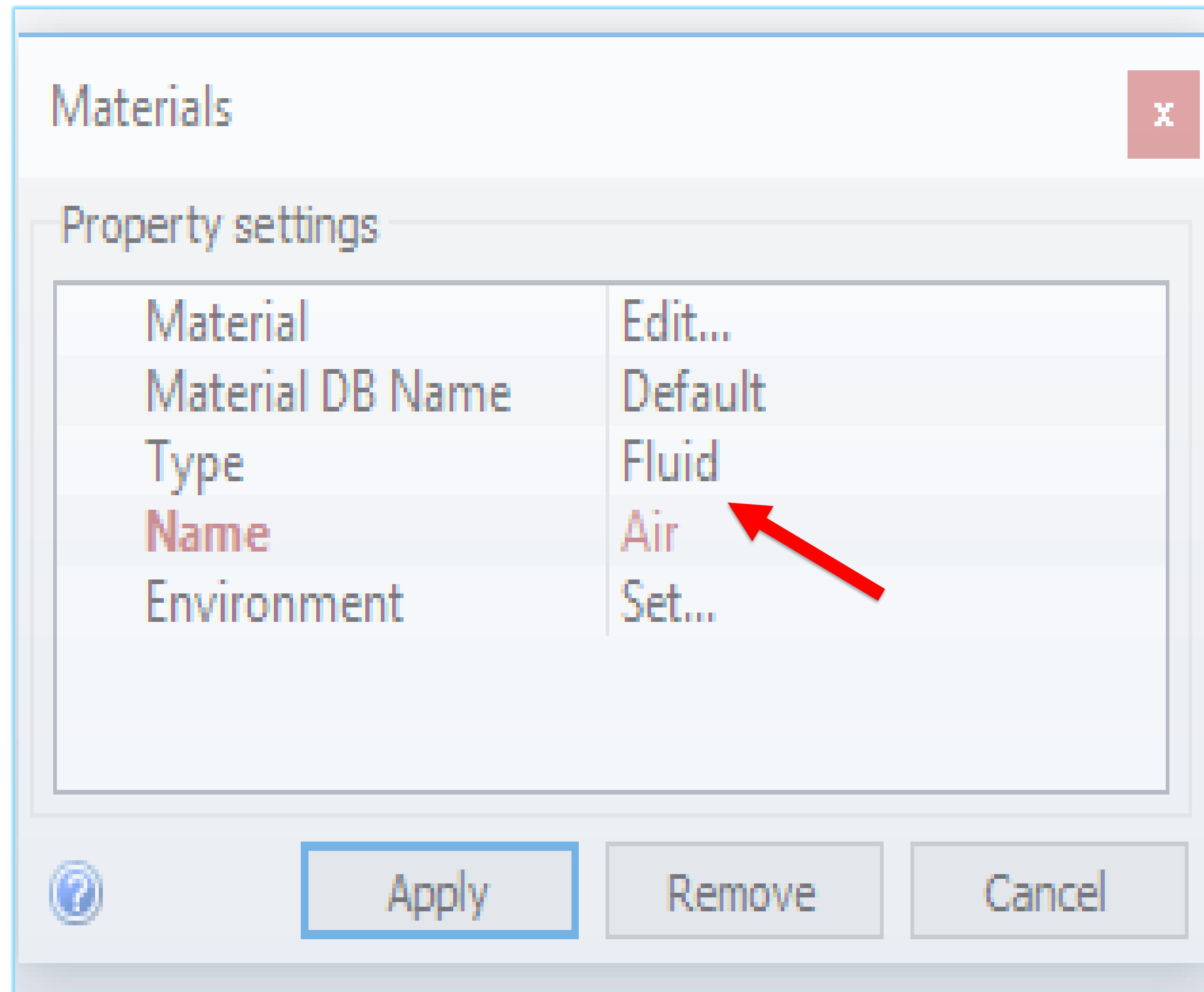


# Materials



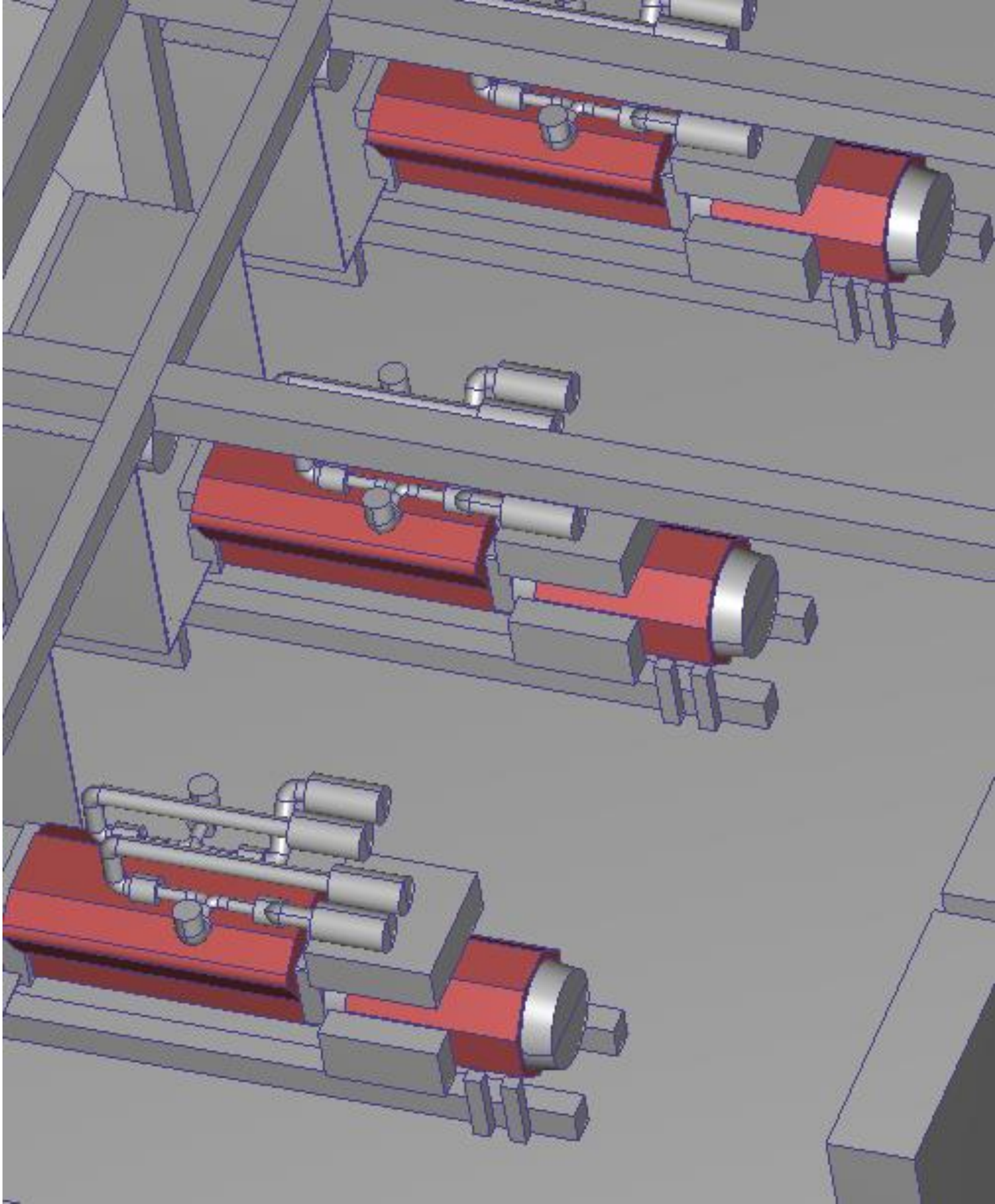
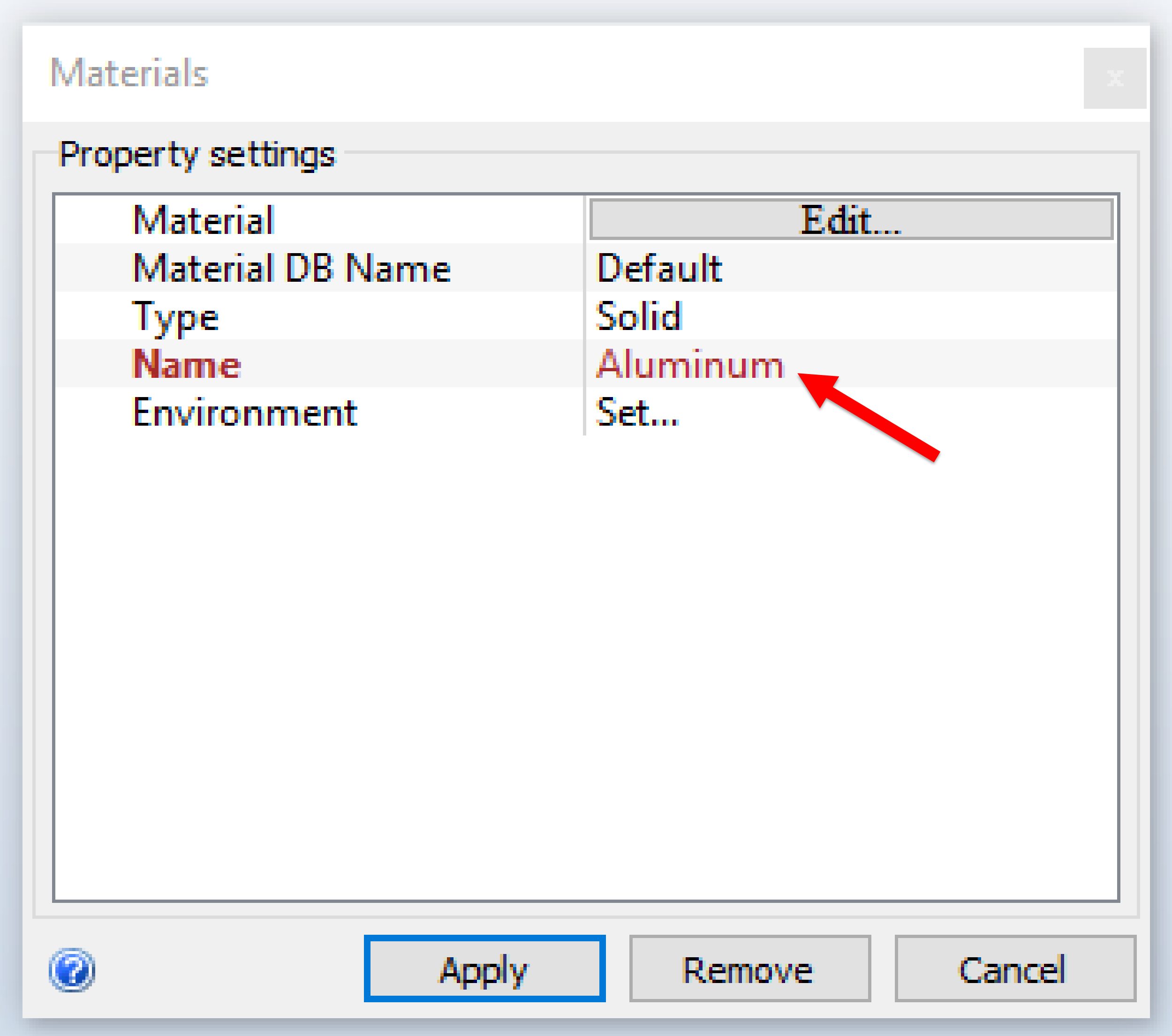


# Air volume and assign air material



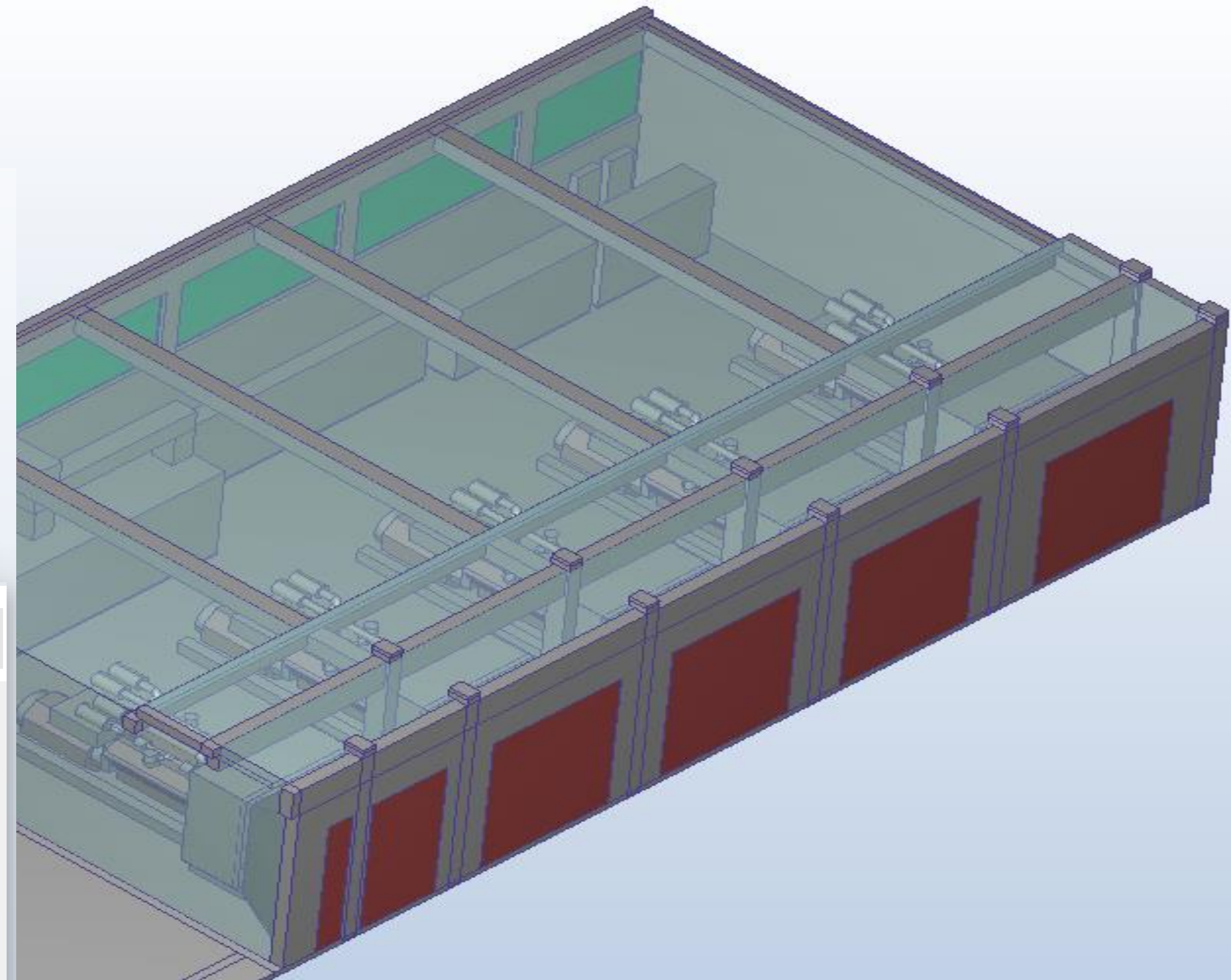
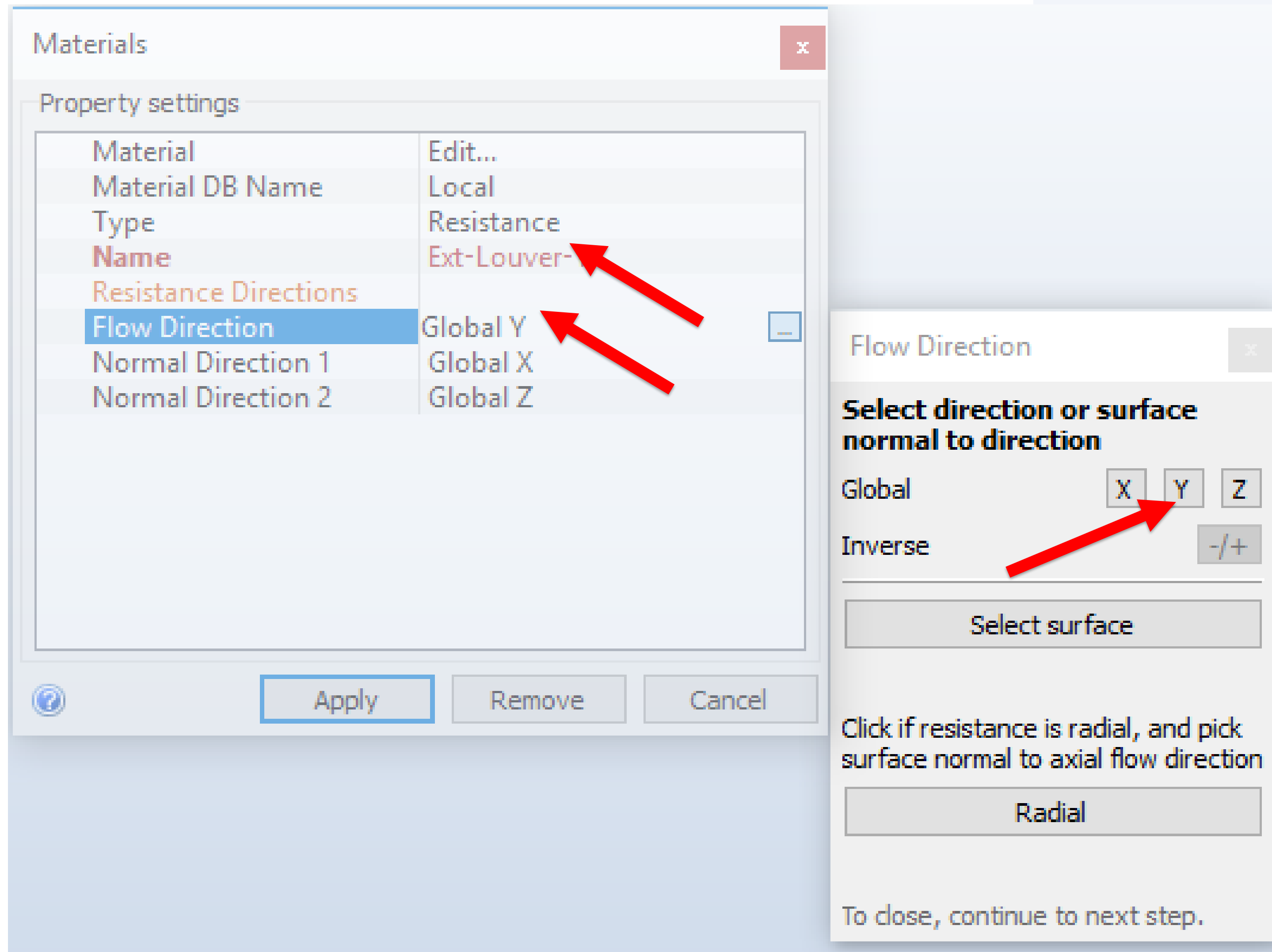


# Aluminum material assigned for engine and alternator





# “Resistance” material for ventilation louvers (inlet and outlet)



Free area ratio of 0.55 in “Through-Flow” direction. In other directions, “0” free area ratio assigned to make flow in one direction.

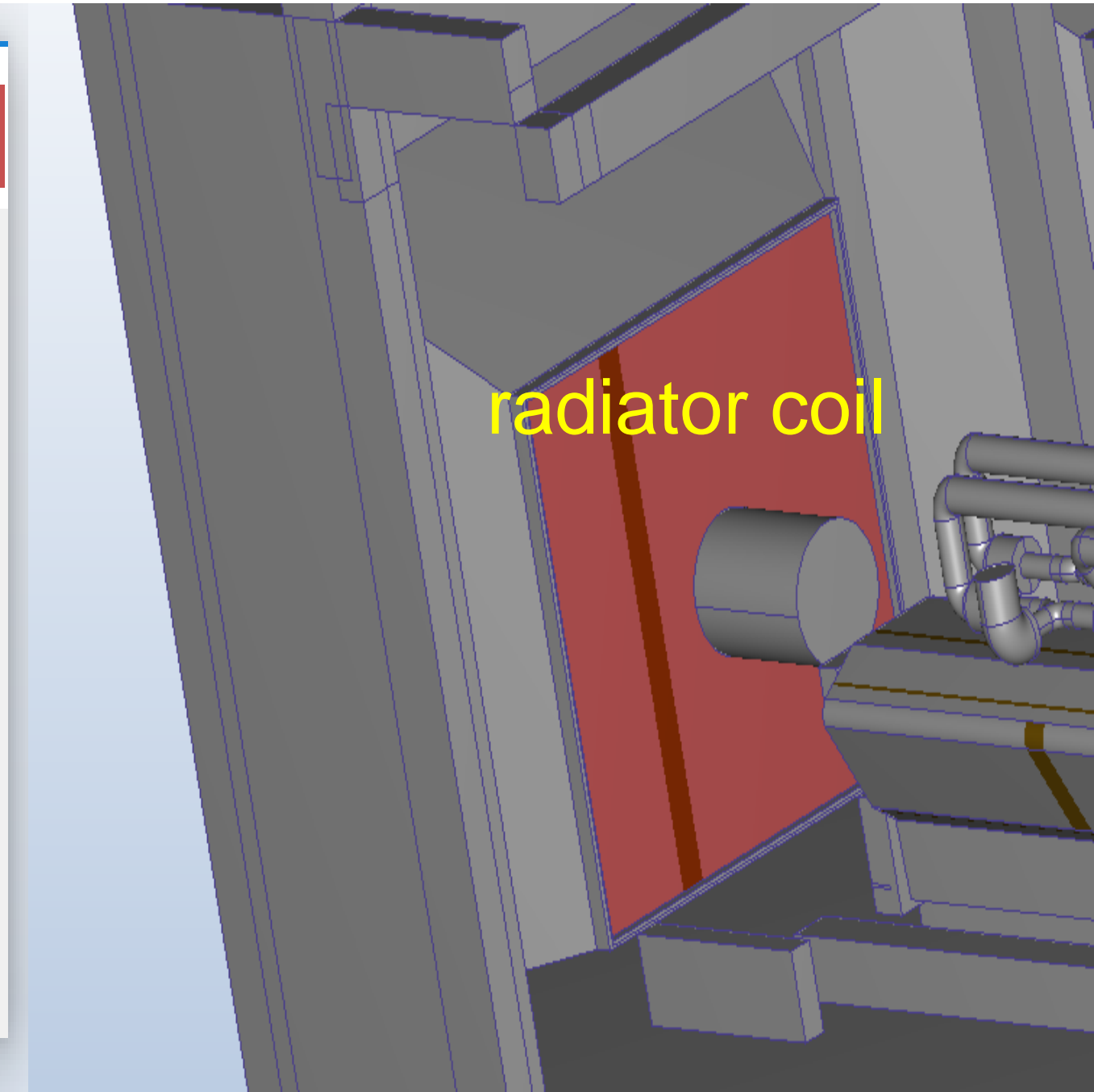
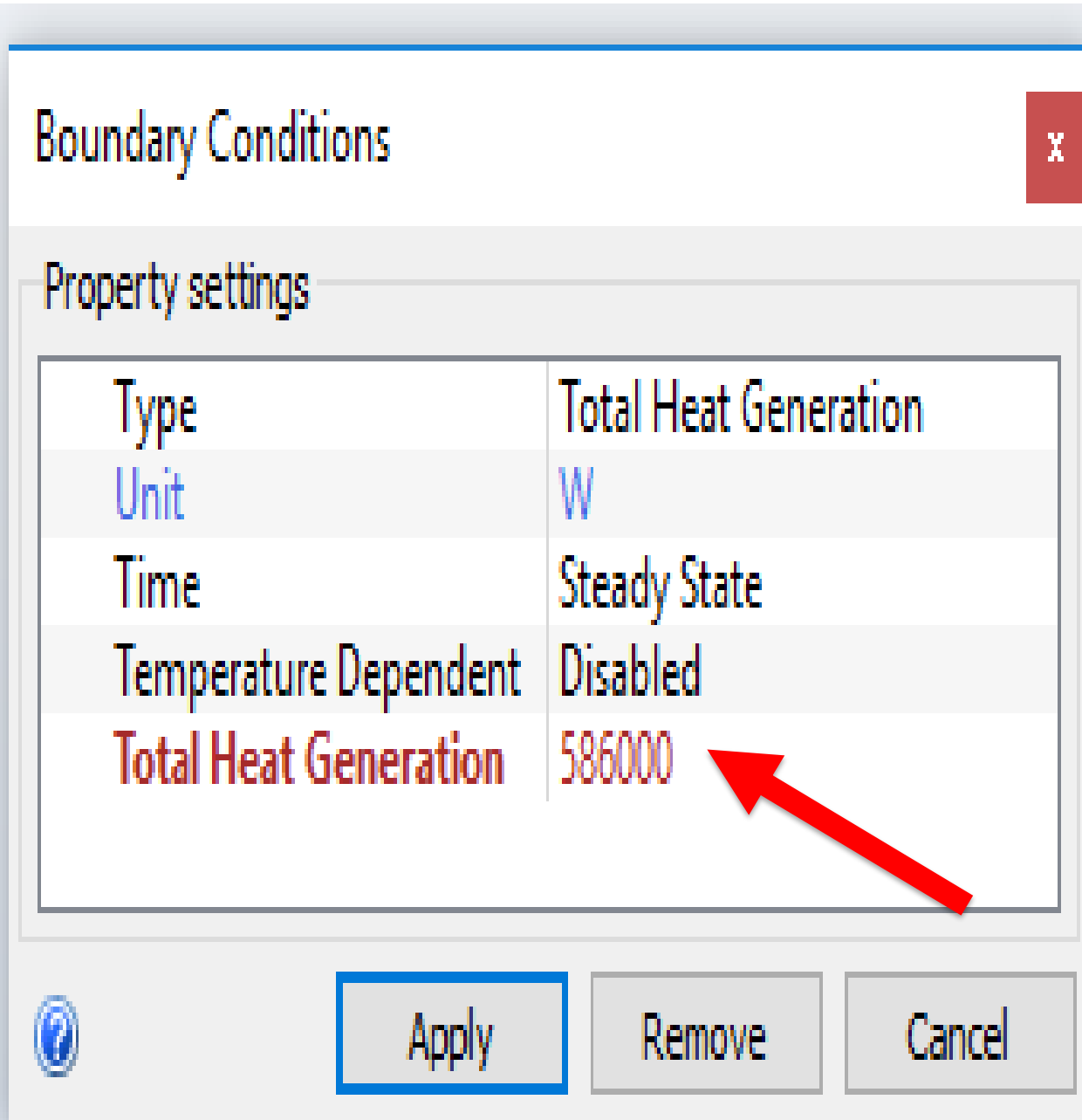
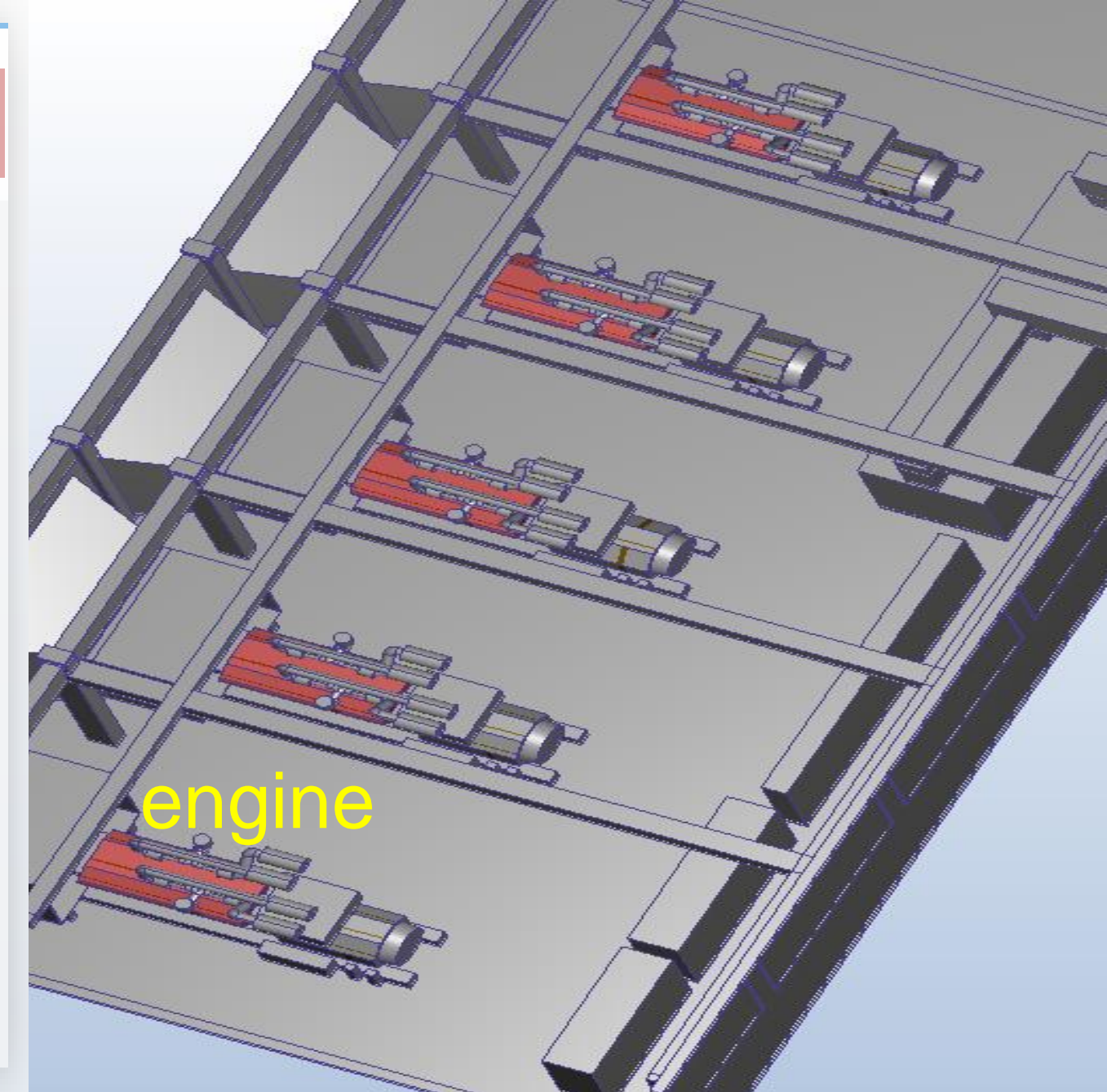
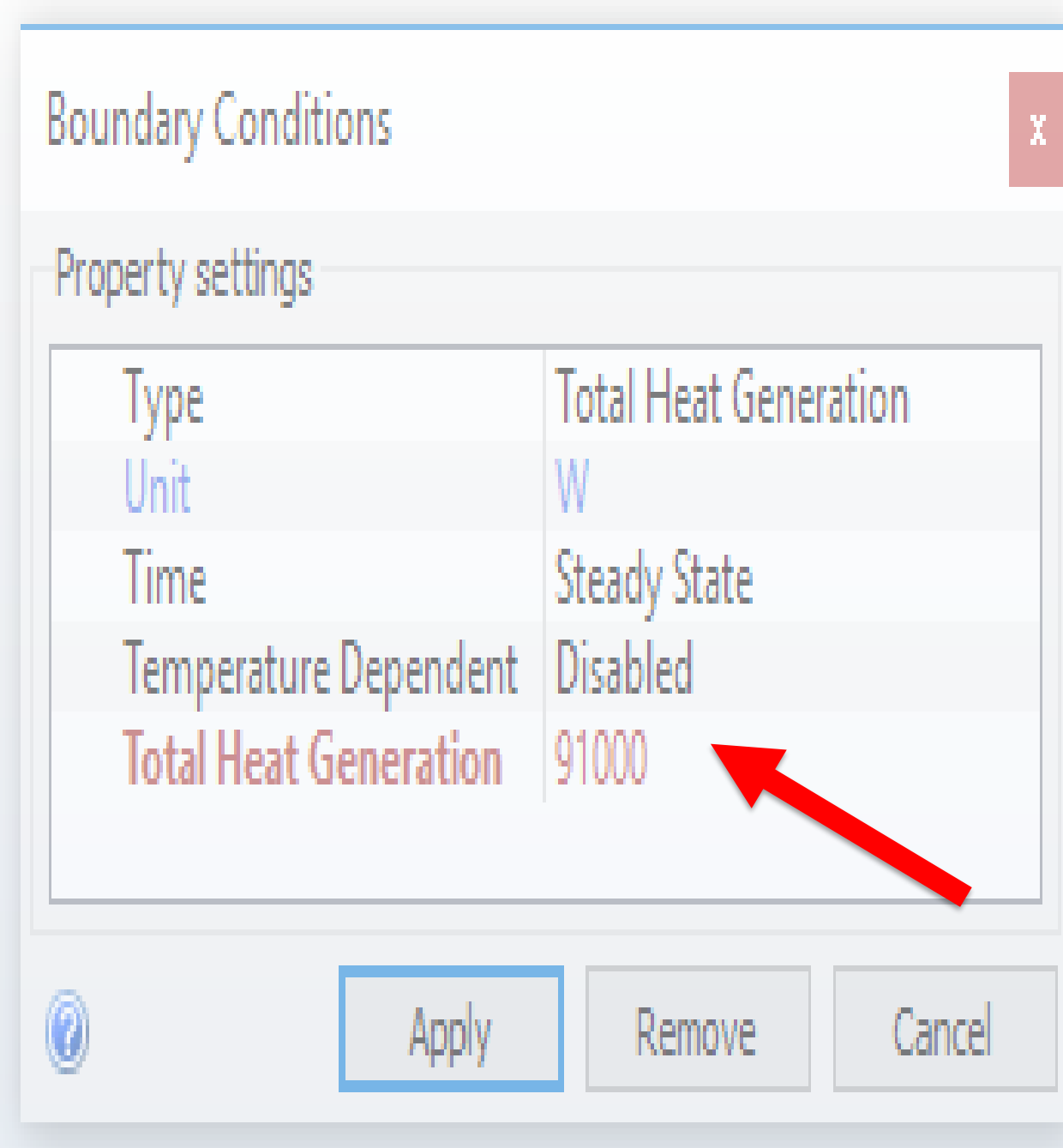
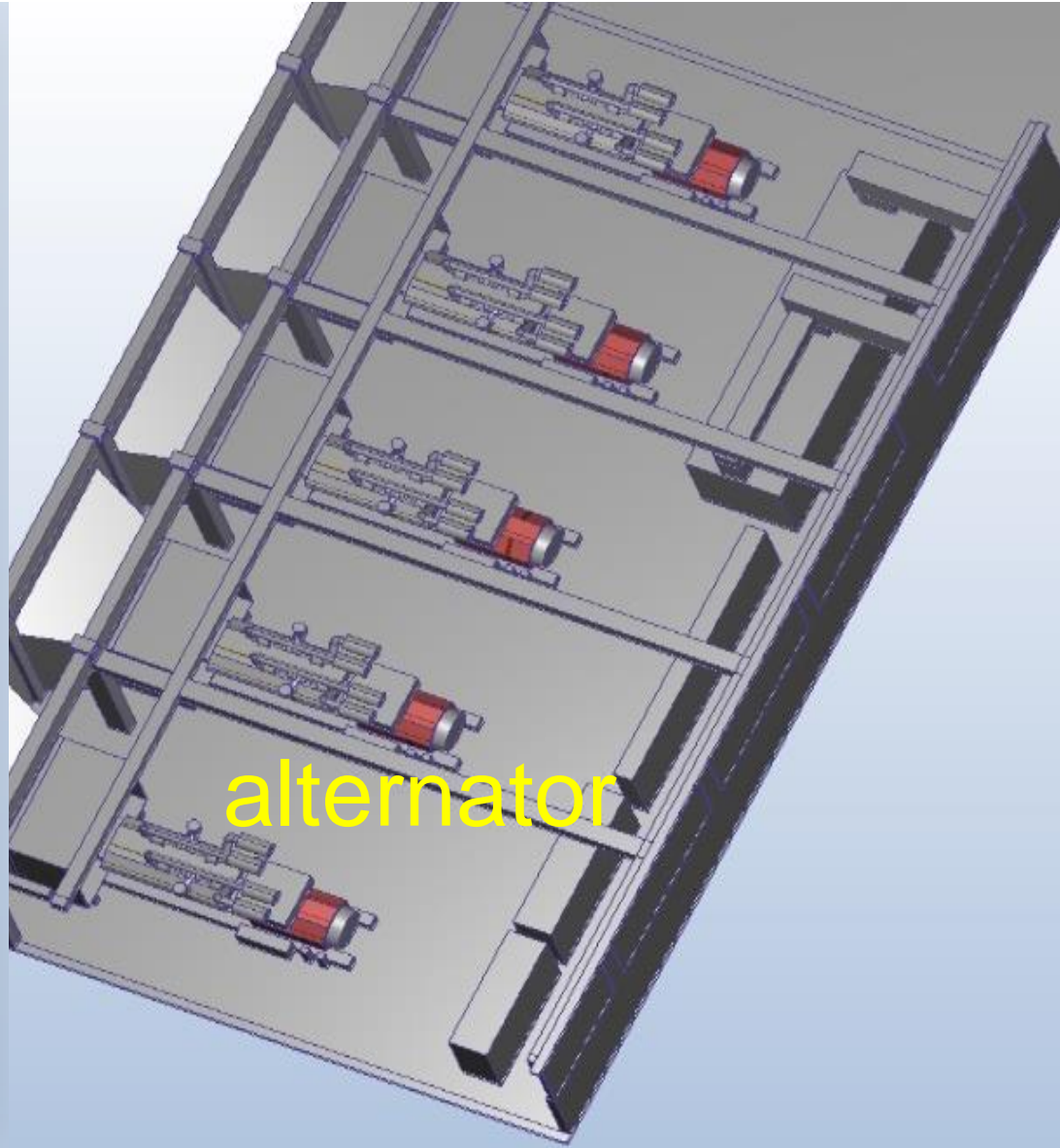
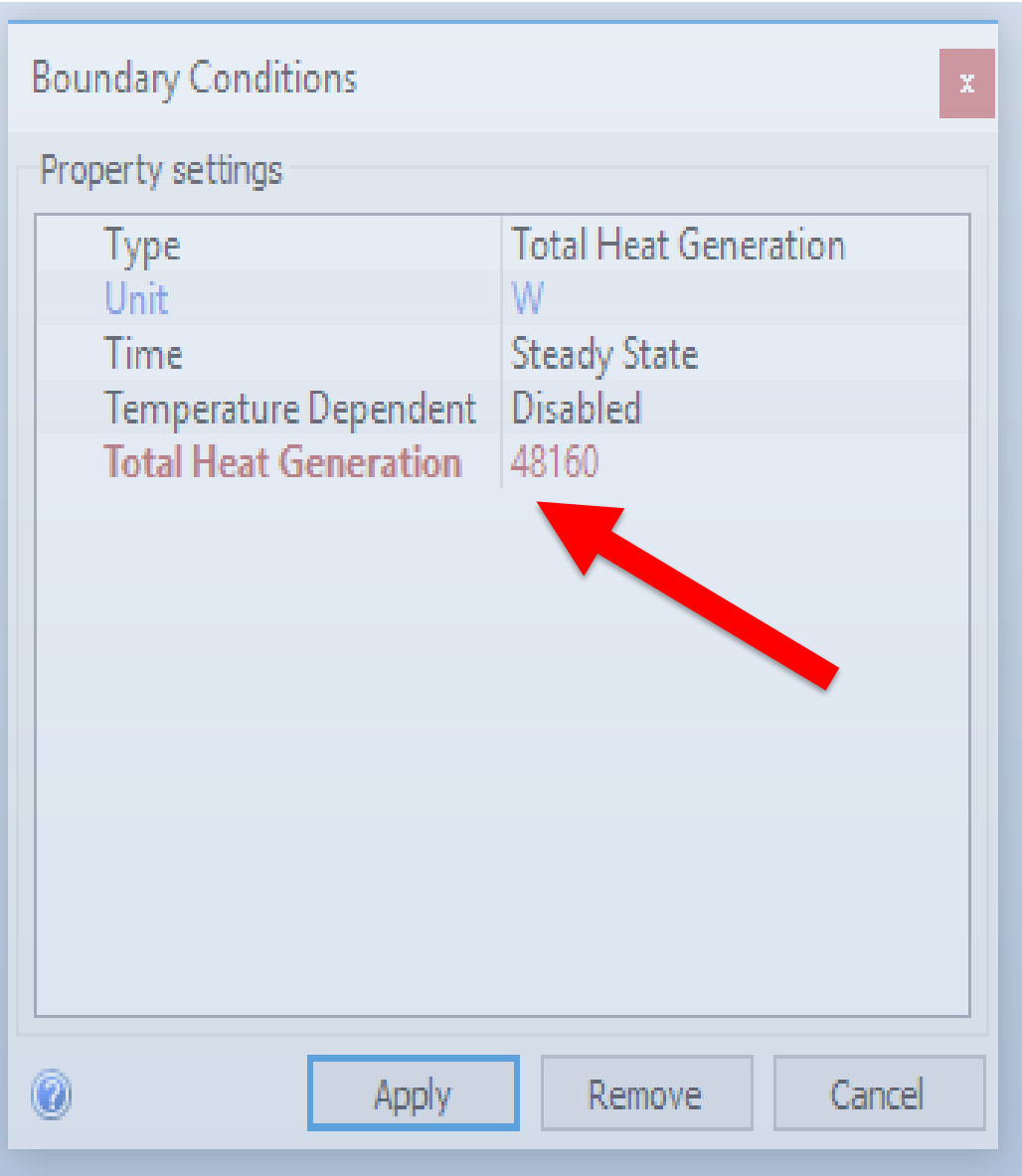


# Boundary Conditions

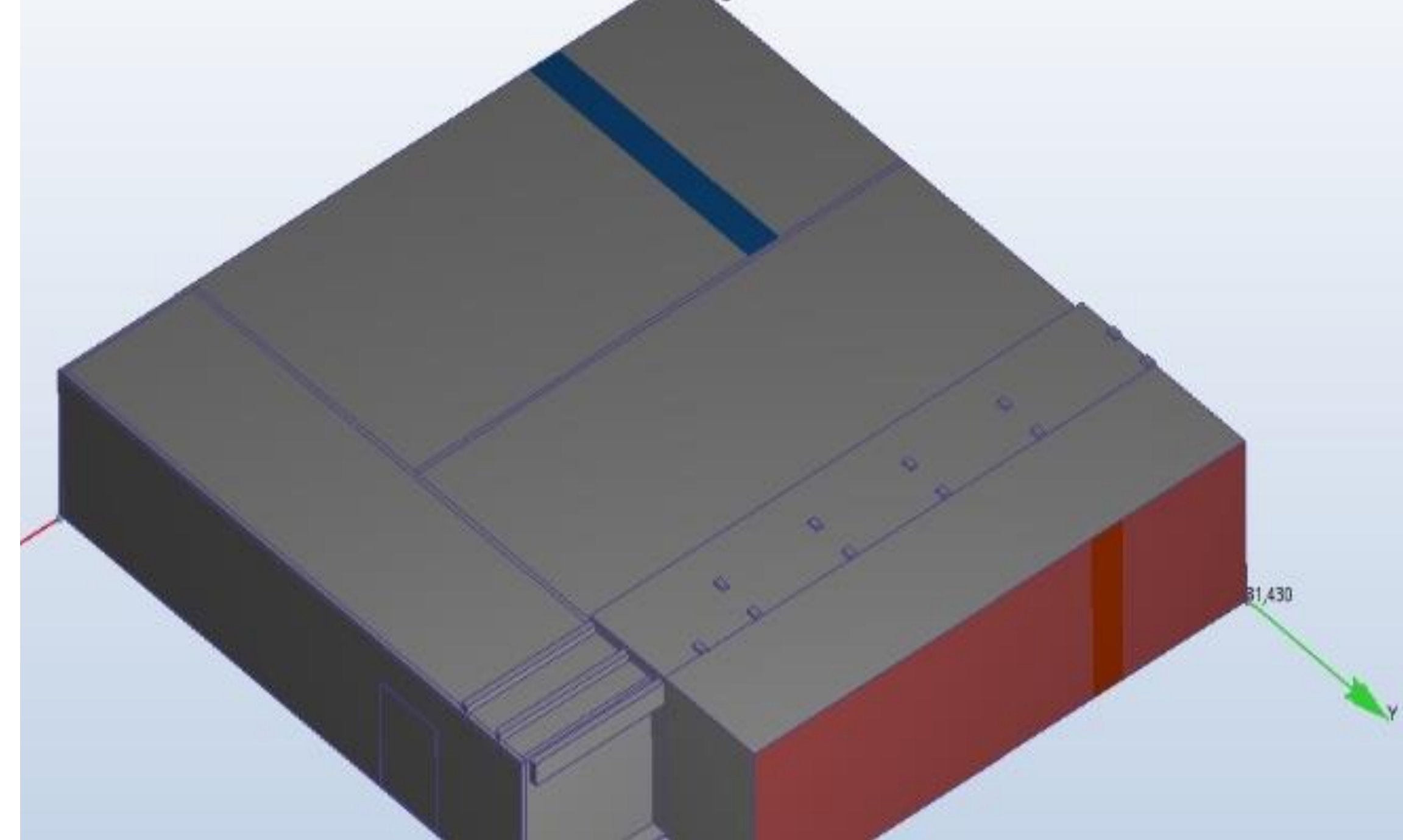
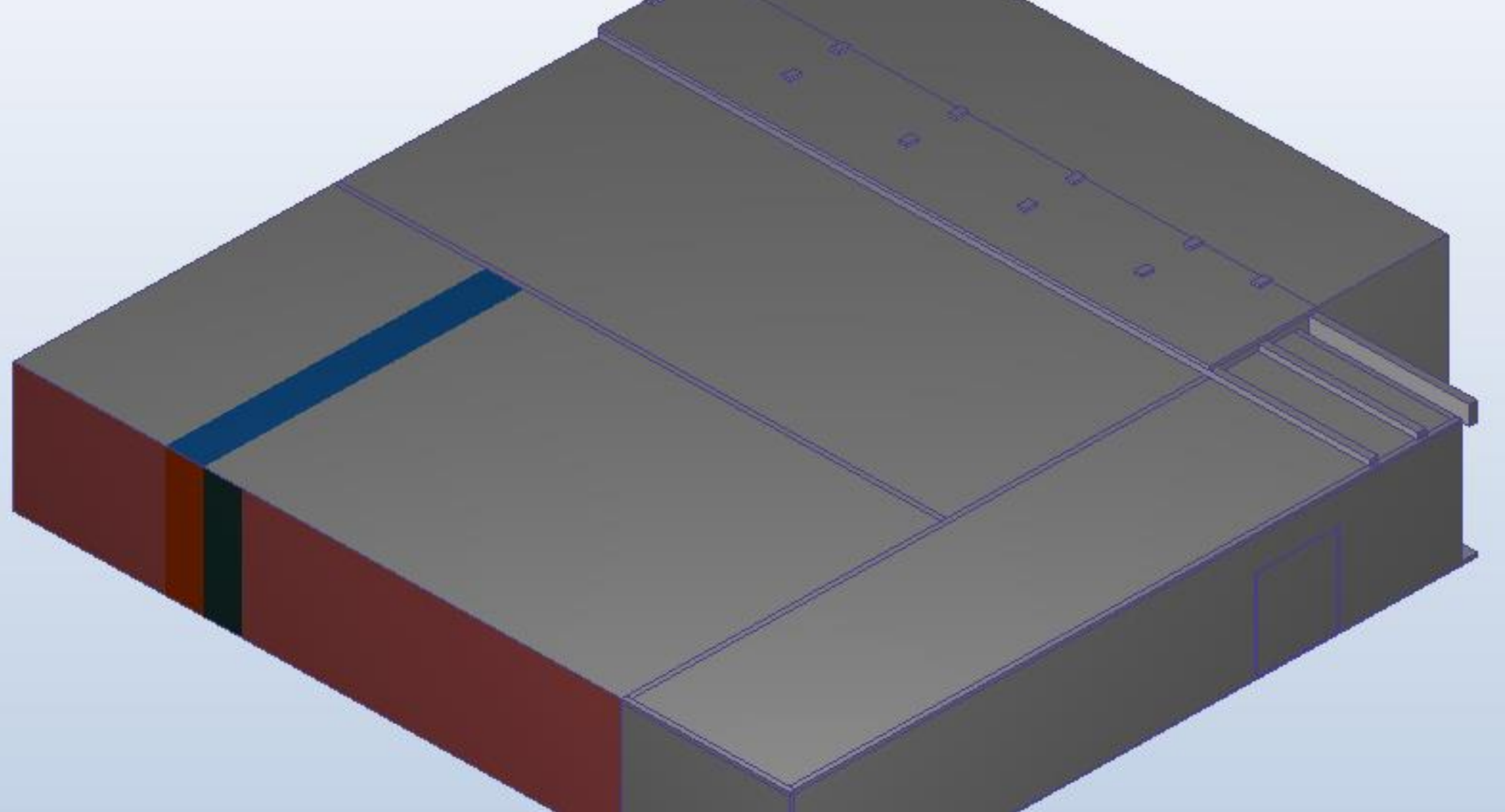




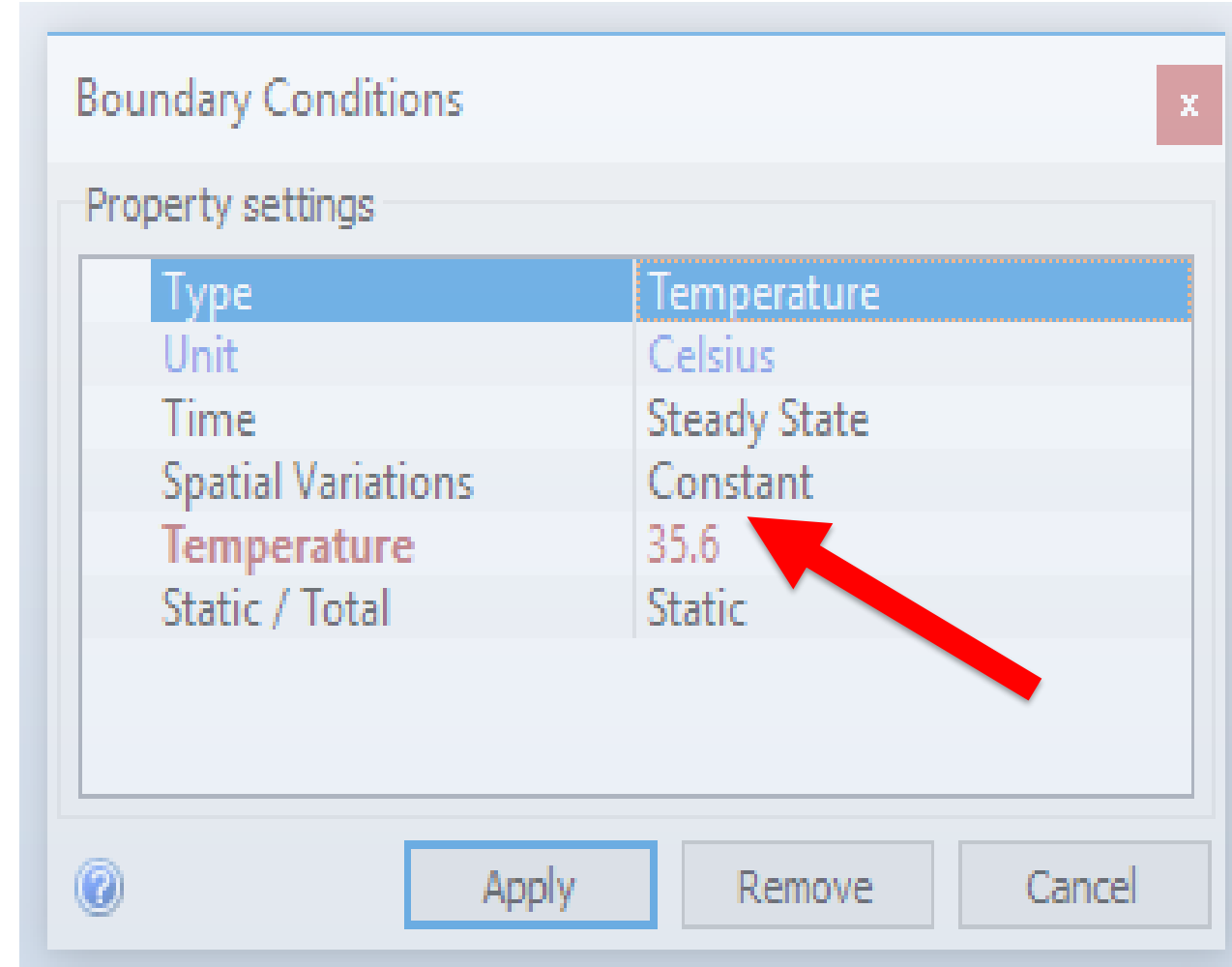
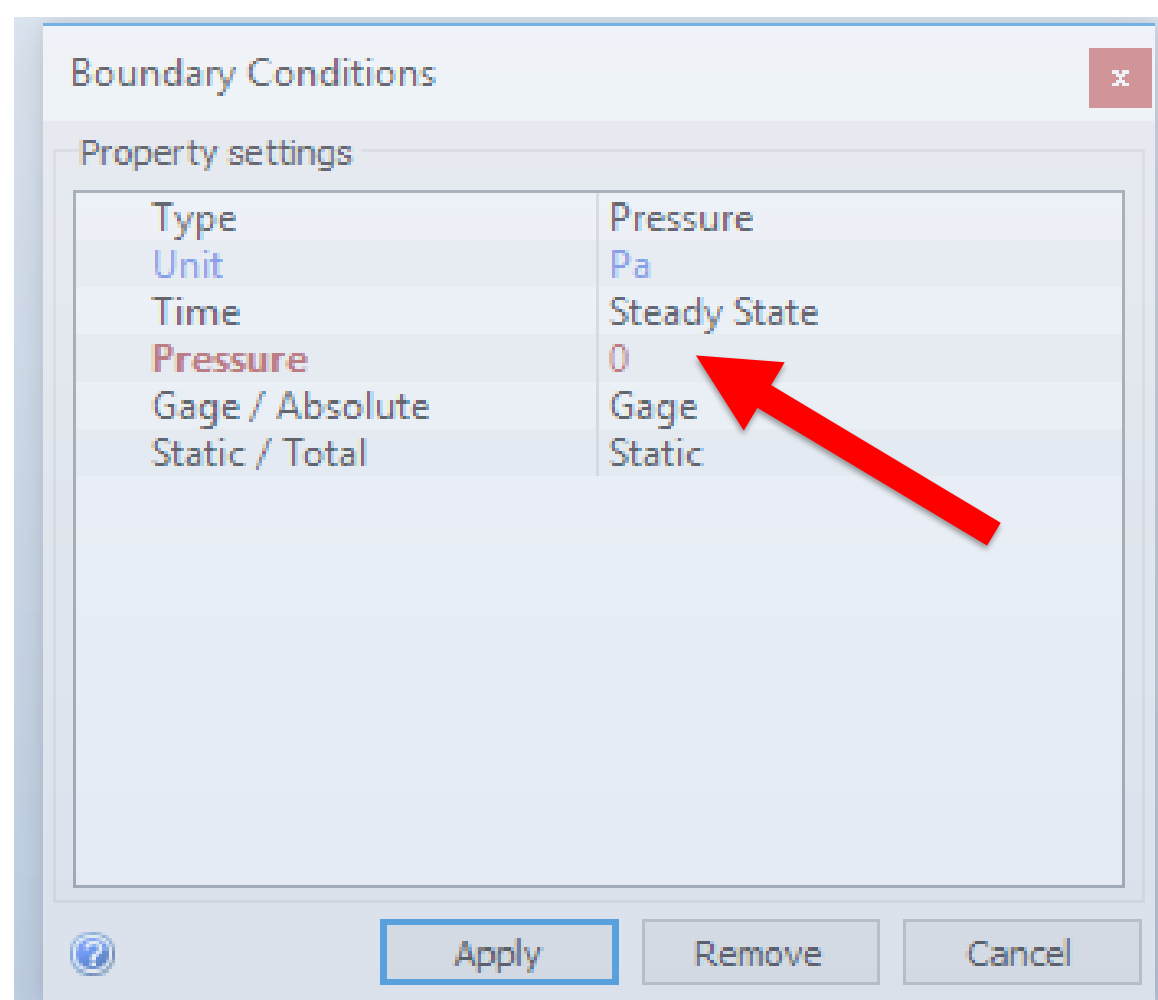
# Total heat generation BC applied to engine, alternator and radiator coil



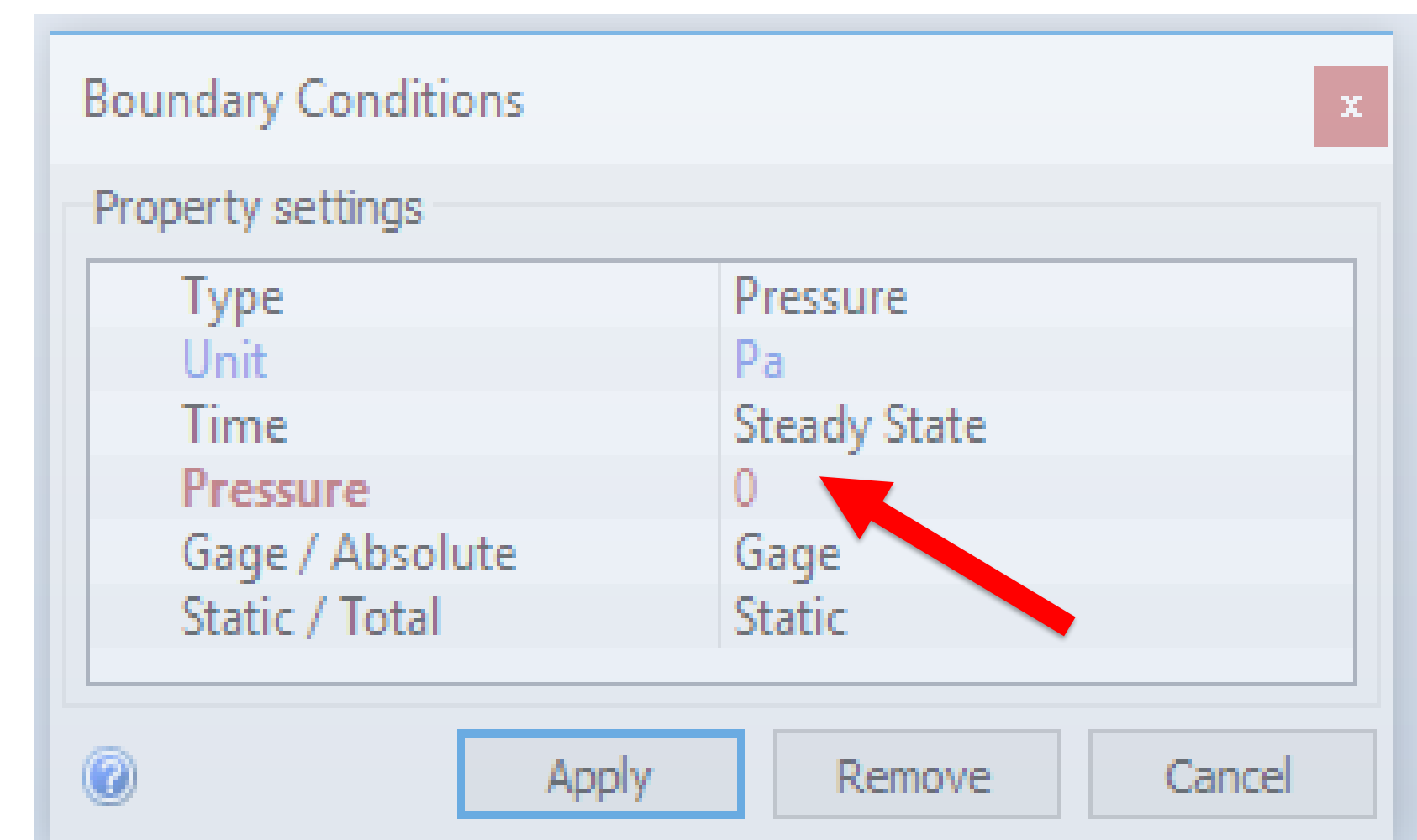




## Inlet BC



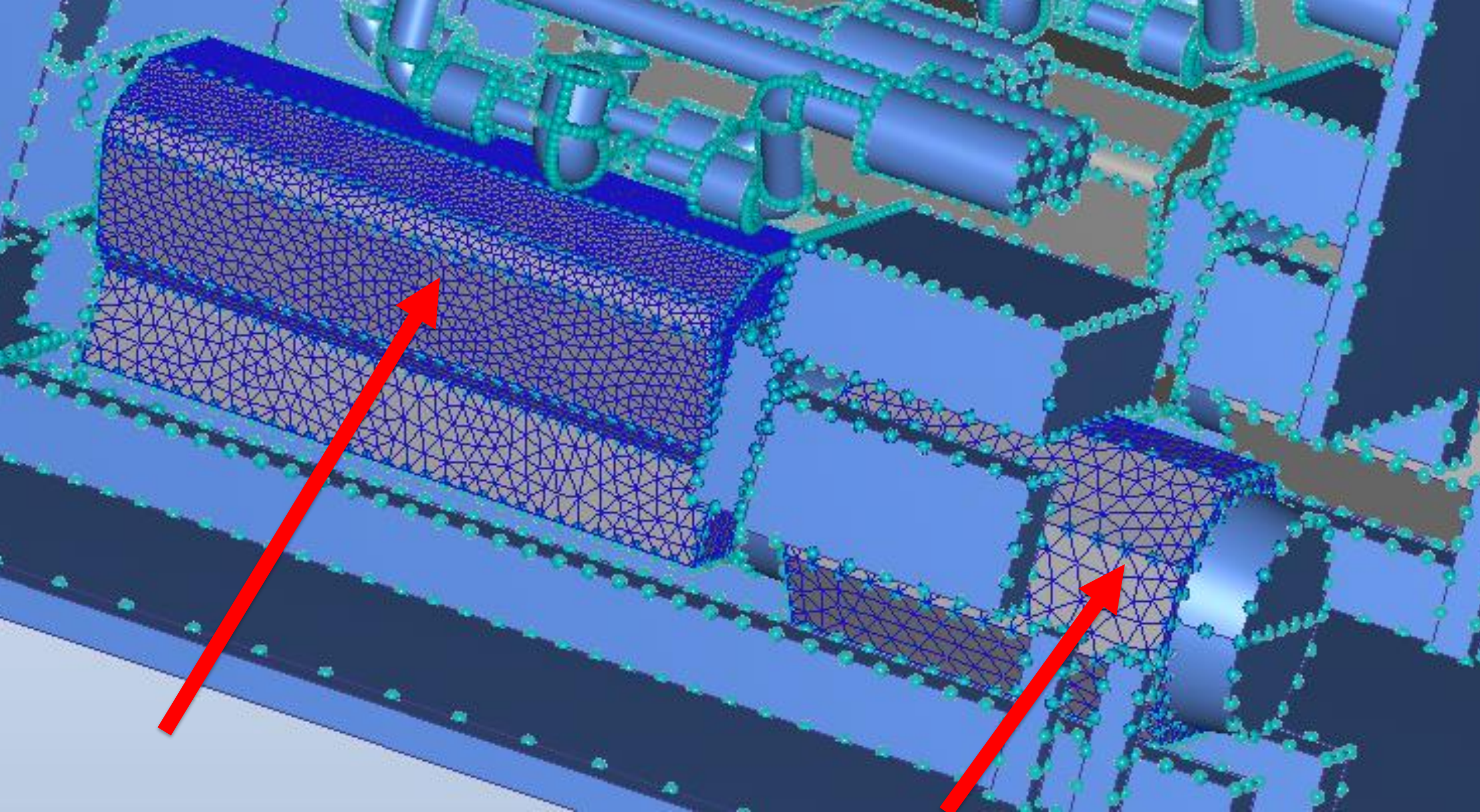
## Outlet BC



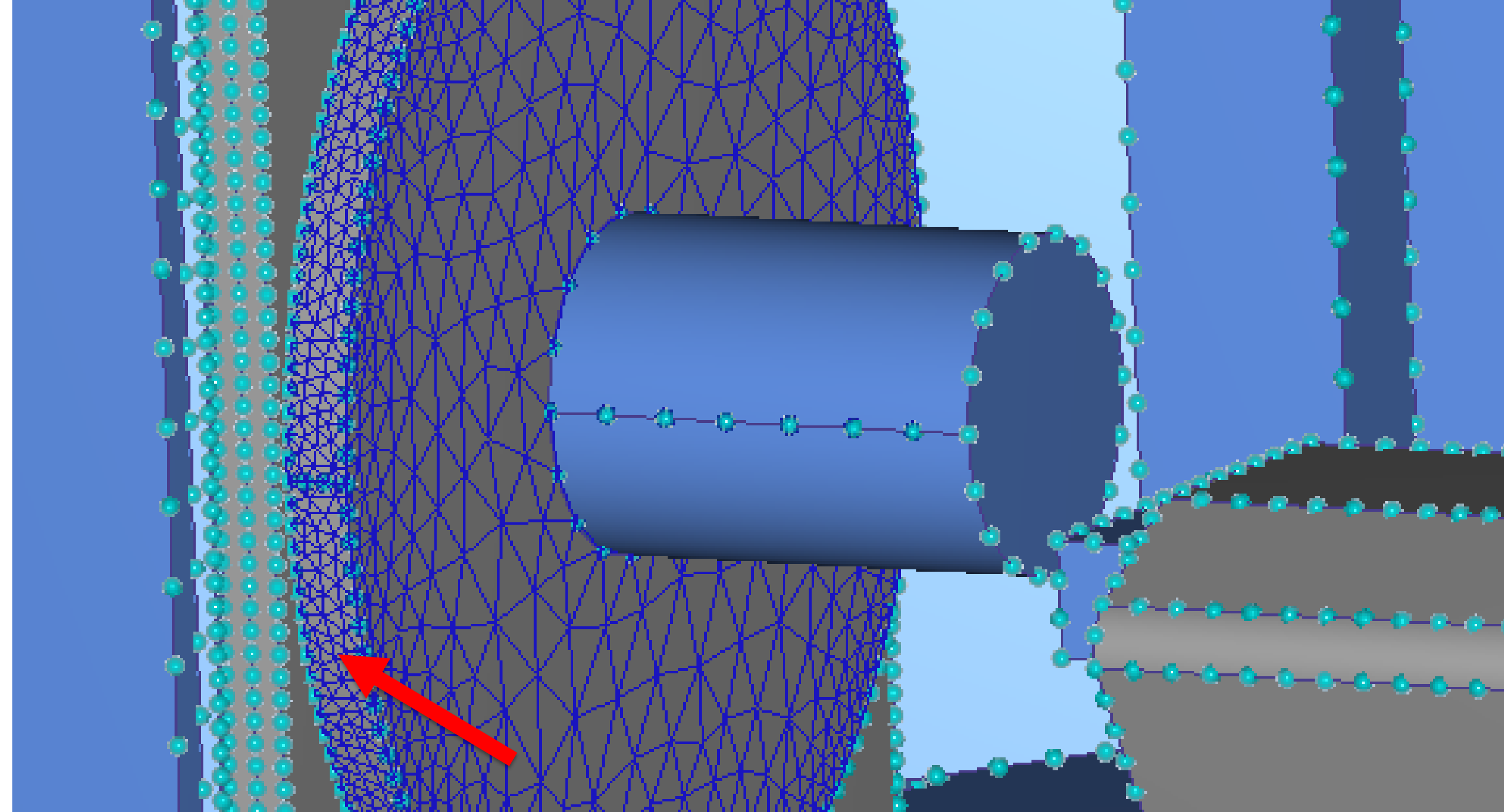
# Meshing





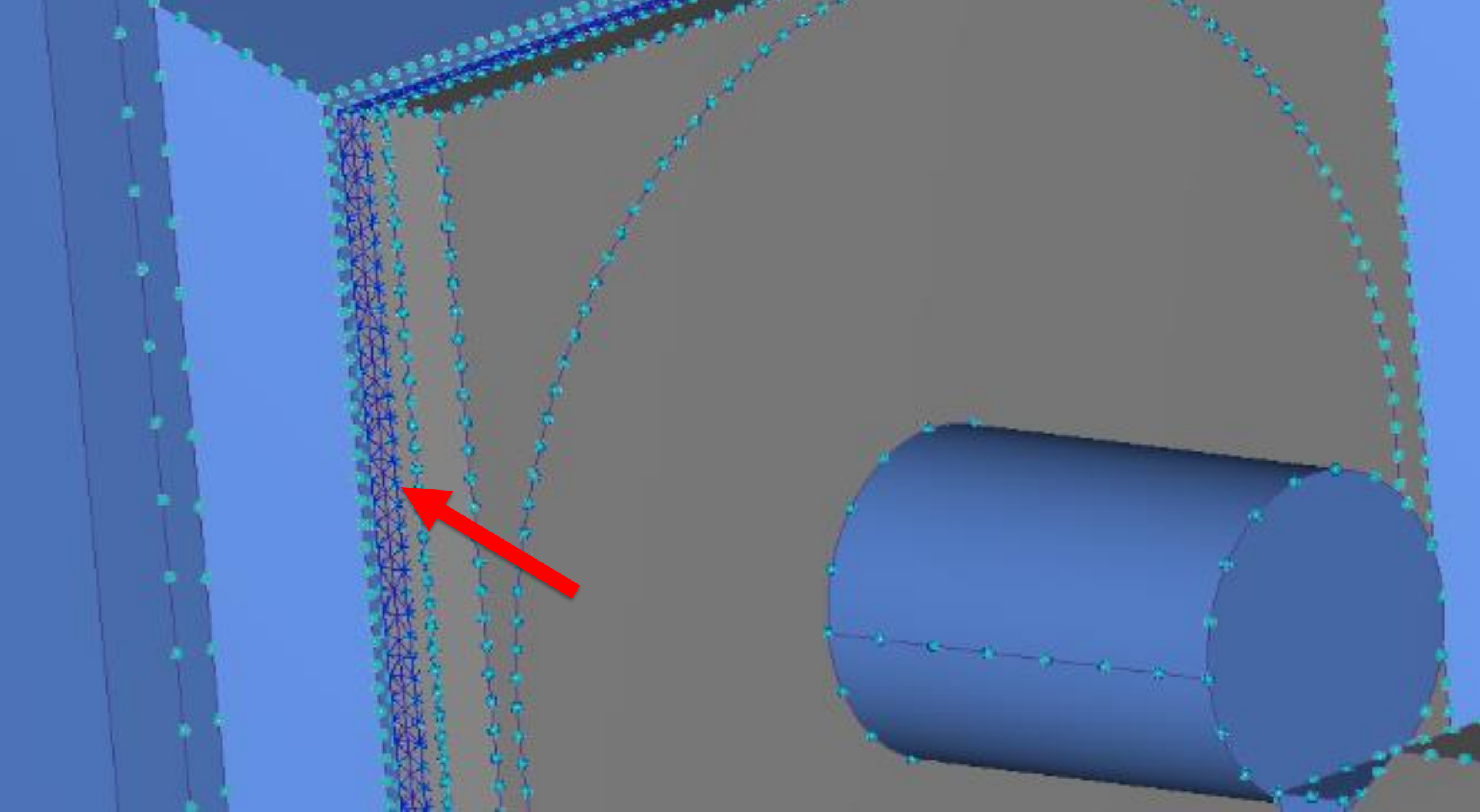


Fine mesh on engine and alternator to capture volumetric heat loss

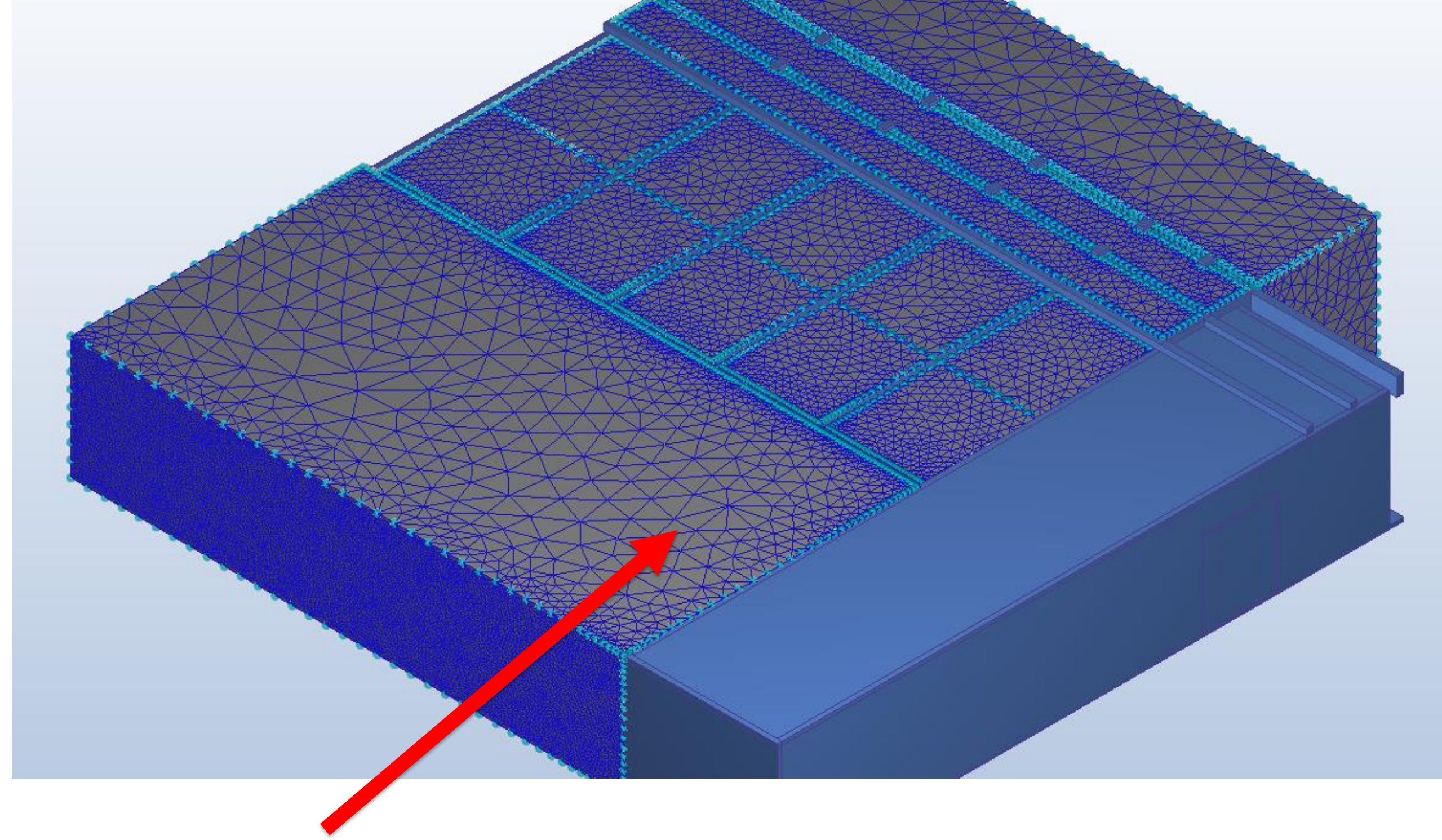


Uniform mesh to capture internal fan flow effects on radiator fan.





Uniform mesh to capture  
flow through radiator coil



Overall mesh for the DG  
room CFD model

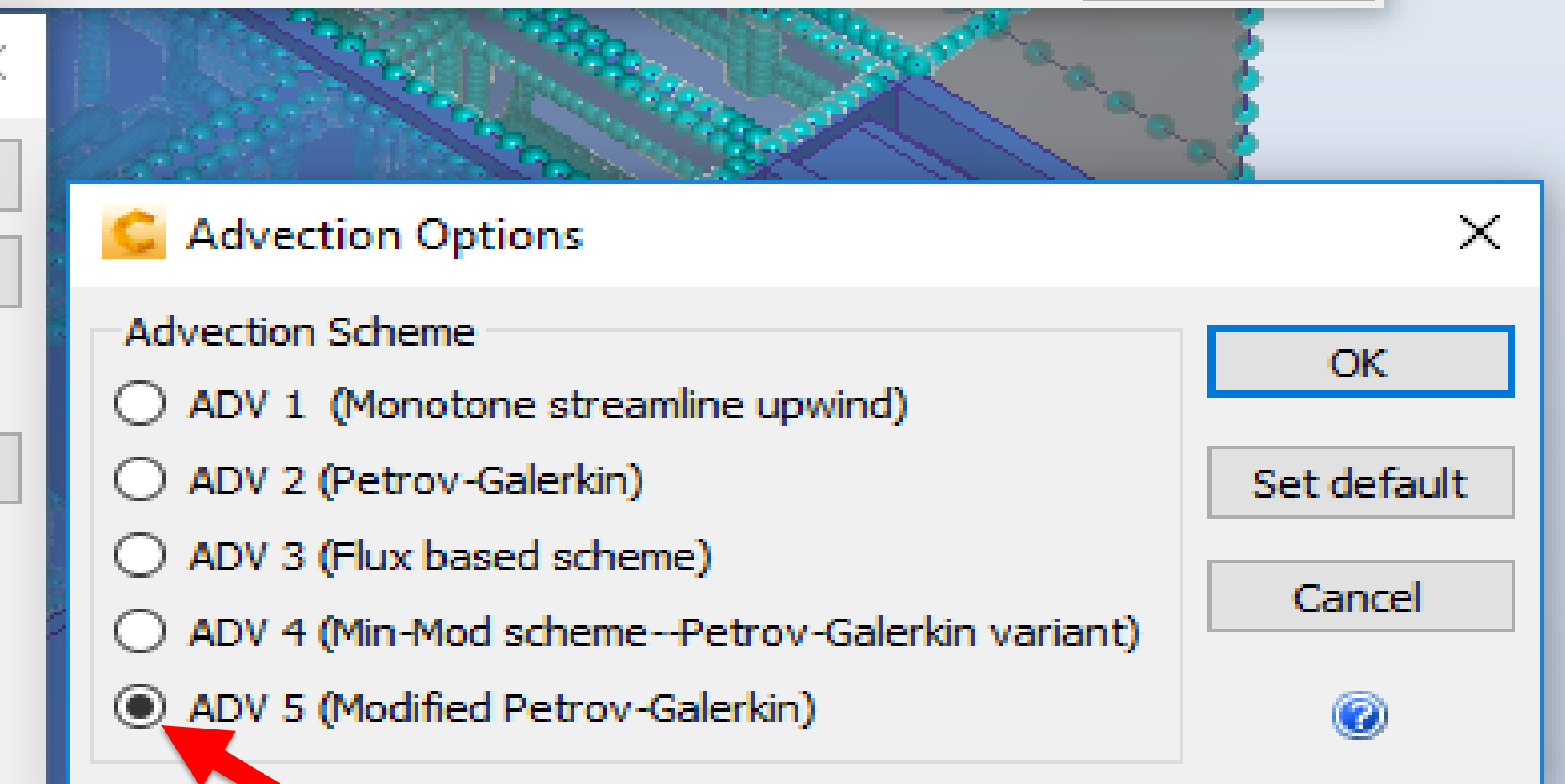
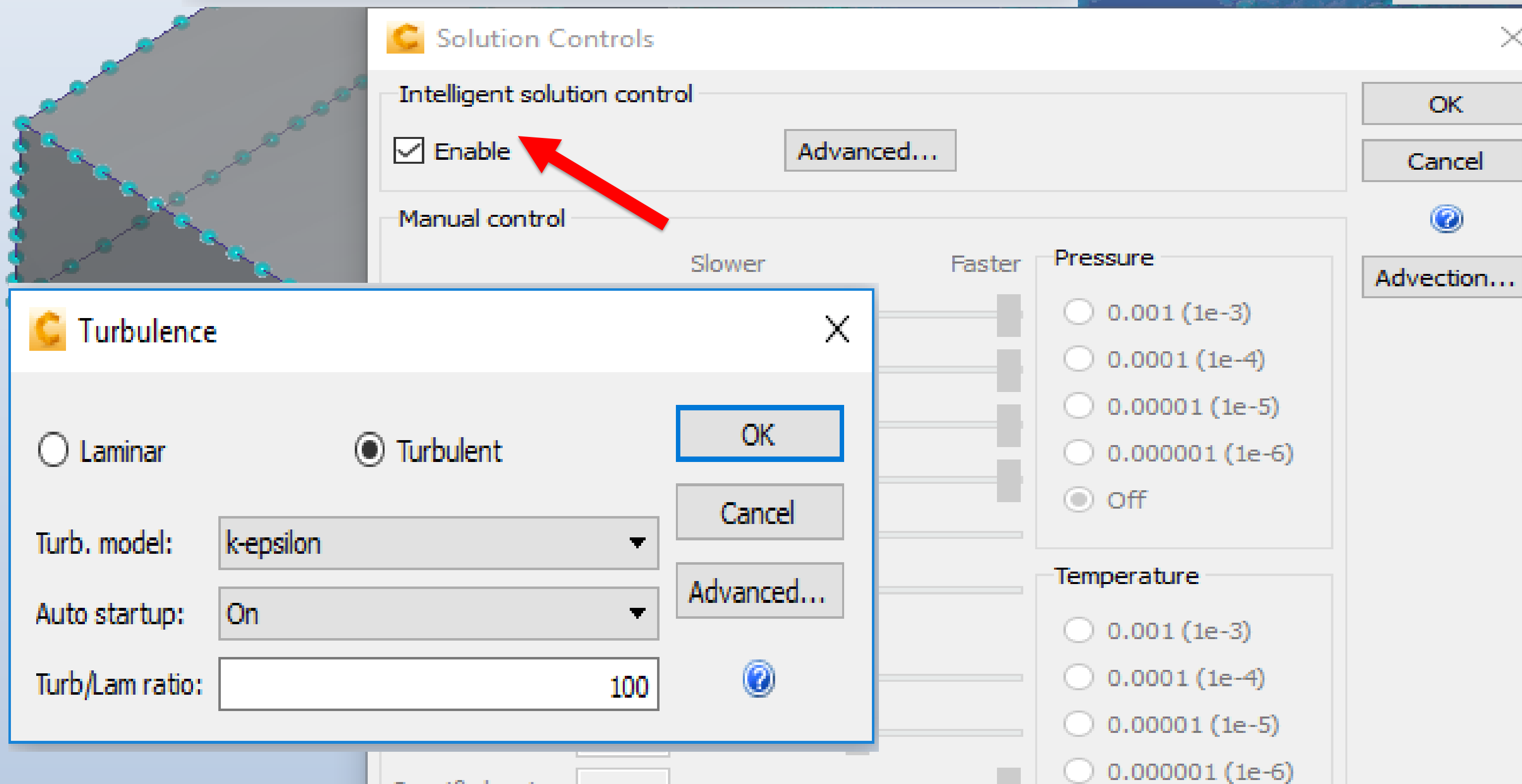
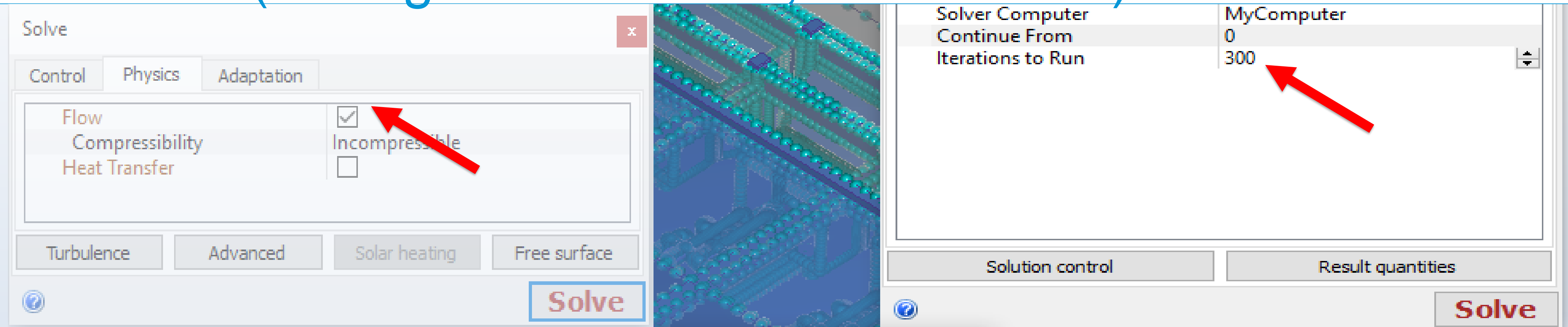


# Solver Settings





# Solver settings for air flow analysis (Intelligent control-on, Advection 5)



# Solver settings for heat transfer analysis (Intelligent control-off, Advection 5)

The image displays three screenshots of a software interface for solving heat transfer problems, showing various settings and controls.

**Solve Window (Top Left):** This window has tabs for Control, Physics, and Adaptation. The Control tab is active, showing the following settings:

Parameter	Value
Solution Mode	Steady State
Save Intervals	
Solver Computer	MyComputer
Continue From	s215
Iterations to Run	300

Buttons at the bottom include "Solution control", "Result quantities", and a large red "Solve" button.

**Solution Controls Window (Top Middle):** This window has tabs for Intelligent solution control and Manual control. The Intelligent solution control tab is active, showing the following settings:

Parameter	Value
Intelligent solution control	<input type="checkbox"/> Enable
Advanced...	

Buttons at the bottom include "OK", "Cancel", and "Advection...".

**Advection Options Window (Bottom Left):** This window has a tab for Advection Scheme. The following settings are shown:

Advection Scheme	Value
ADV 1 (Monotone streamline upwind)	<input type="radio"/>
ADV 2 (Petrov-Galerkin)	<input type="radio"/>
ADV 3 (Flux based scheme)	<input type="radio"/>
ADV 4 (Min-Mod scheme--Petrov-Galerkin variant)	<input type="radio"/>
ADV 5 (Modified Petrov-Galerkin)	<input checked="" type="radio"/>

Buttons at the bottom include "OK", "Set default", "Cancel", and a question mark icon.

**Solve Window (Top Right):** This window has tabs for Control, Physics, and Adaptation. The Control tab is active, showing the following settings:

Parameter	Value
Flow	<input type="checkbox"/>
Compressibility	Incompressible
Heat Transfer	<input checked="" type="checkbox"/>
Auto Forced Convection	<input type="checkbox"/>
Gravity Method	Earth
Gravity Direction	0,0,0
Radiation	<input type="checkbox"/>

Buttons at the bottom include "Turbulence", "Advanced", "Solar heating", "Free surface", and a large red "Solve" button.

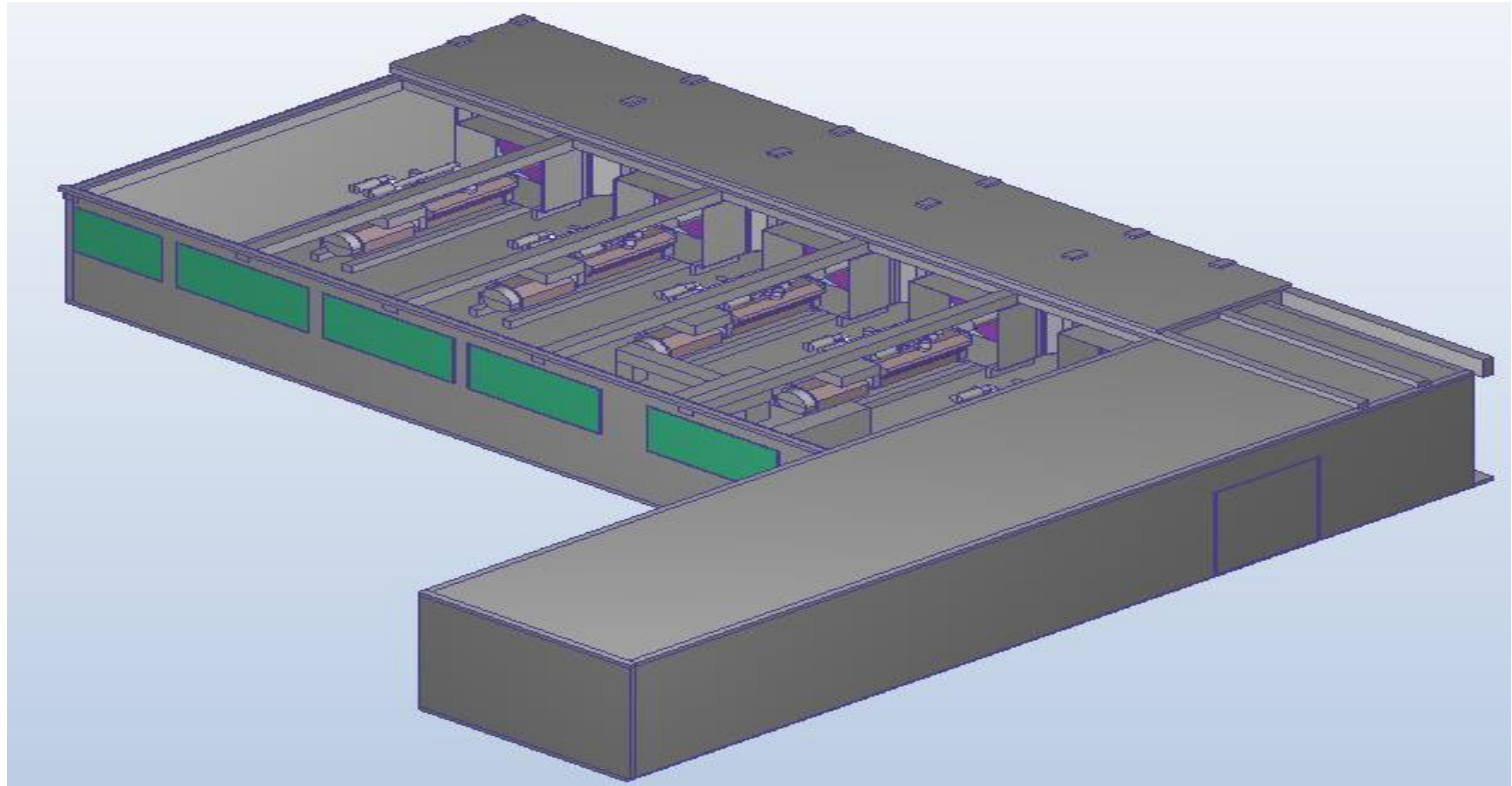


# Key Results for Design Performance

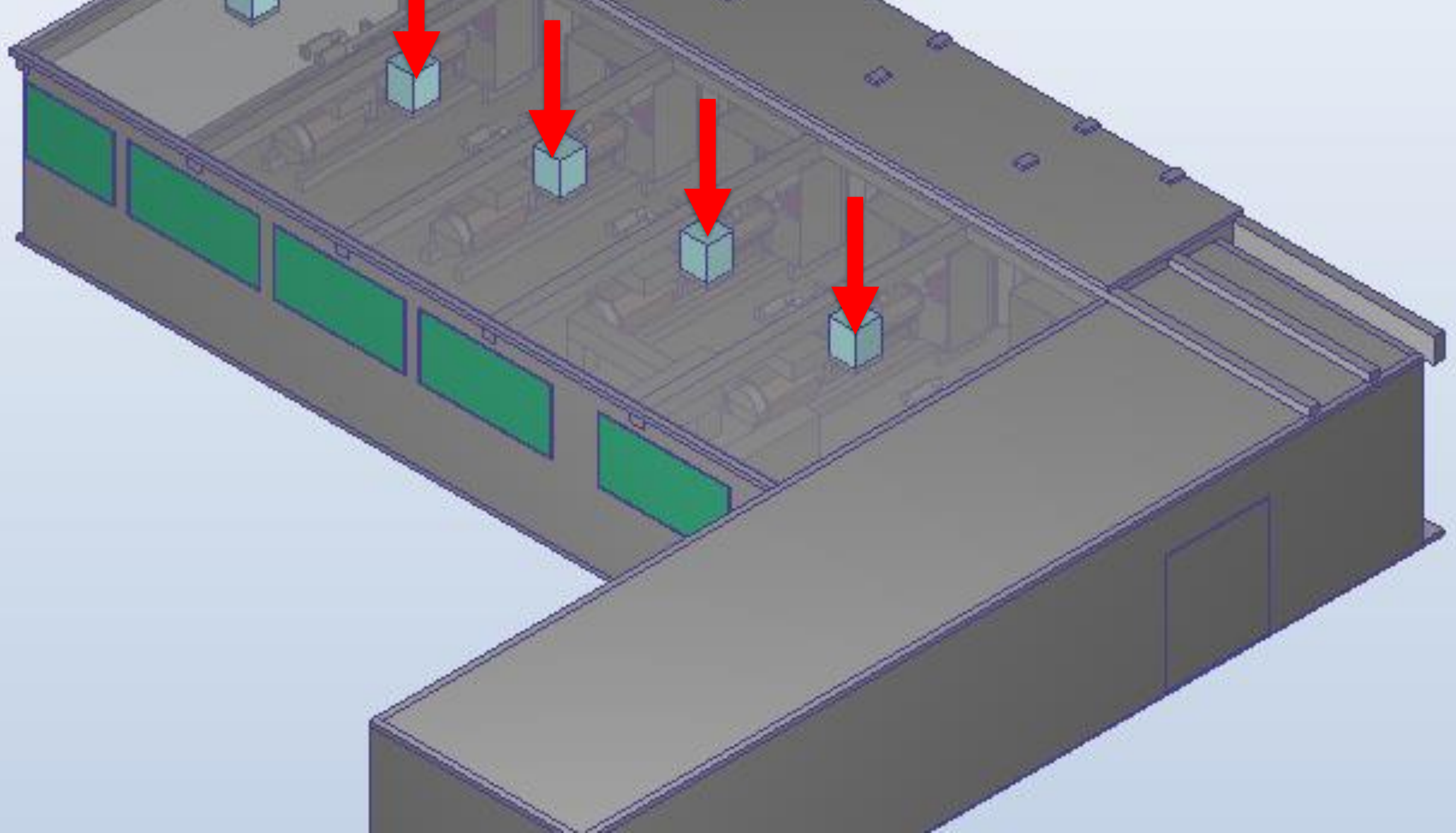




As designed layout  
This is the original proposed design for the room.

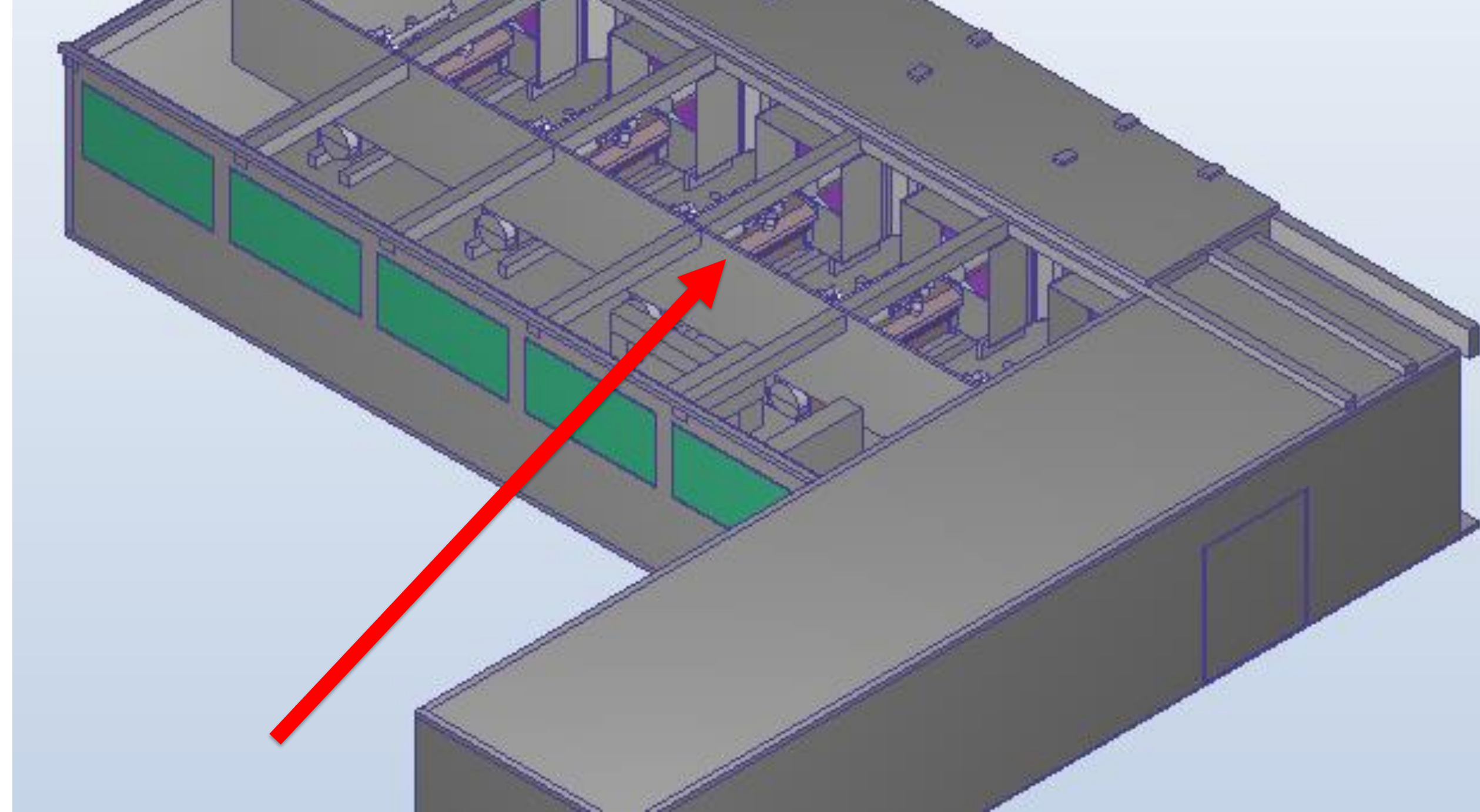






## Design option 1

In this option additional fans were considered for cooling the DG directly by adding air circulation



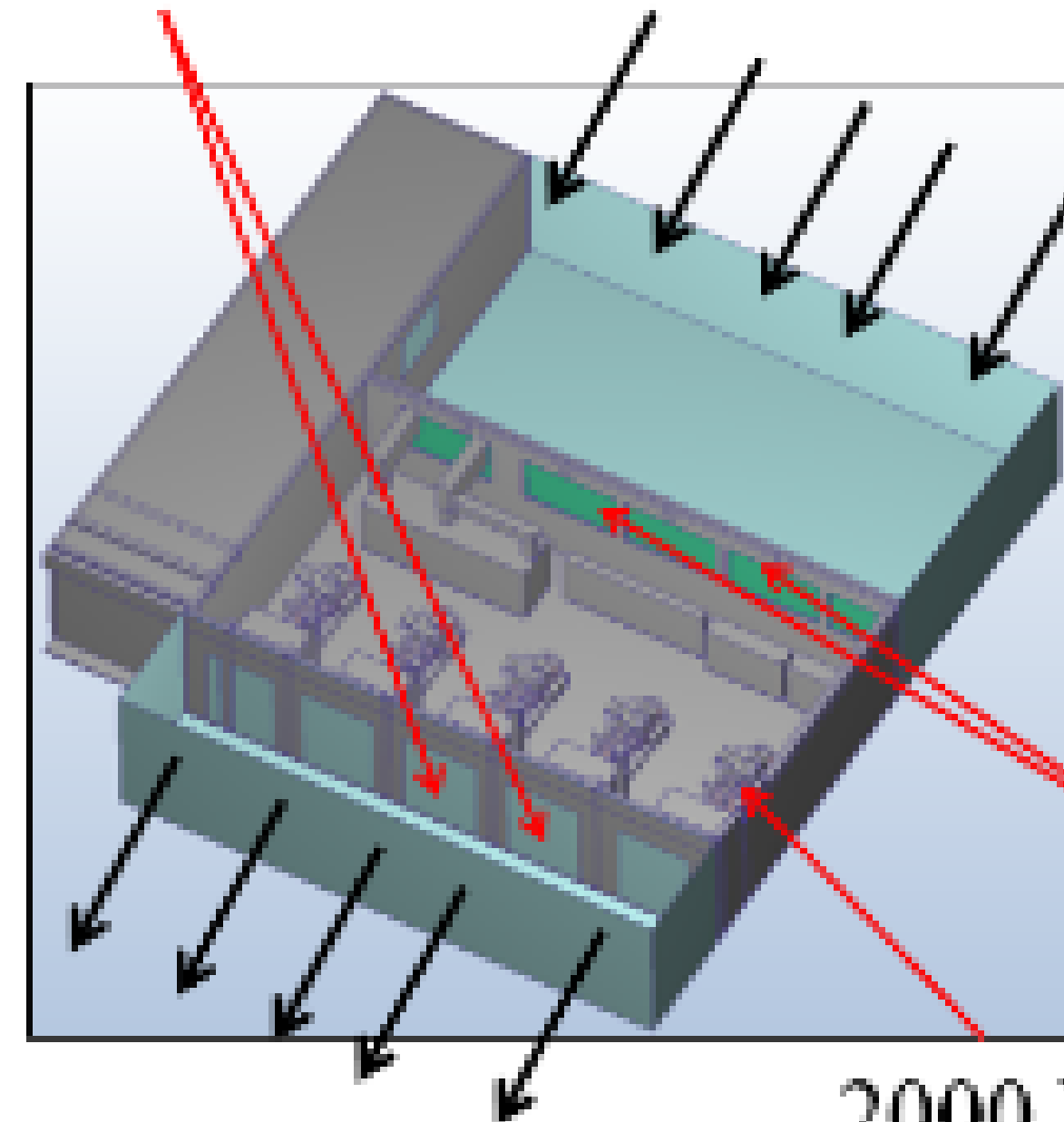
## Design option 2

In this option an additional wall is considered containing the heat and airflow closer to the DG

# CFD analysis results for as-was layout

## As-was lay out

Air exhaust louvers  
Free area ratio = 0.55



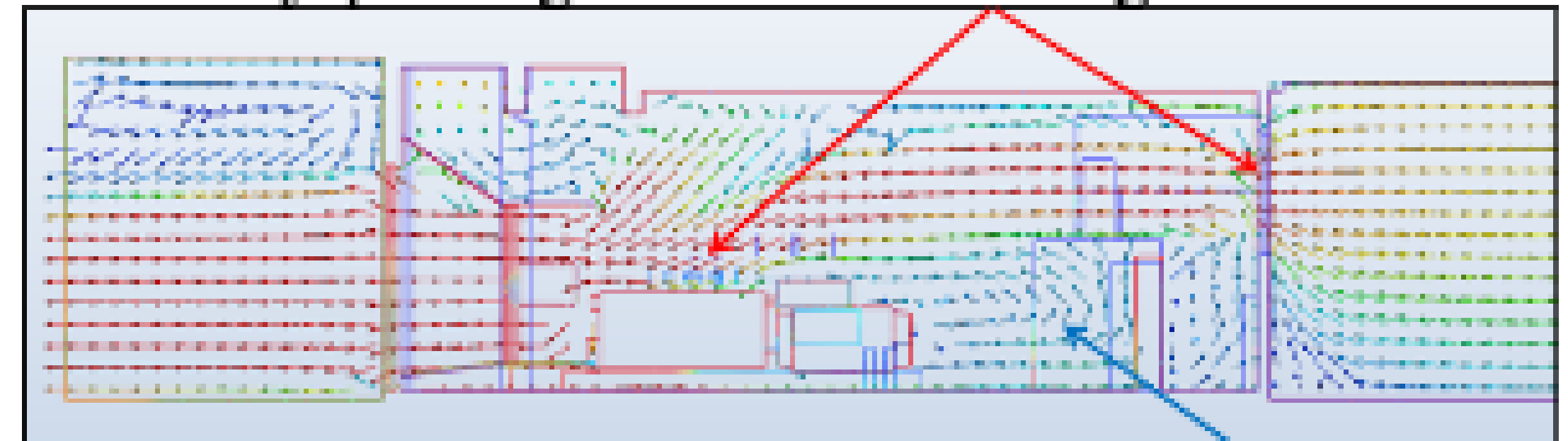
Fresh Air  
intake

Air entry louvers  
Free area  
ratio = 0.55

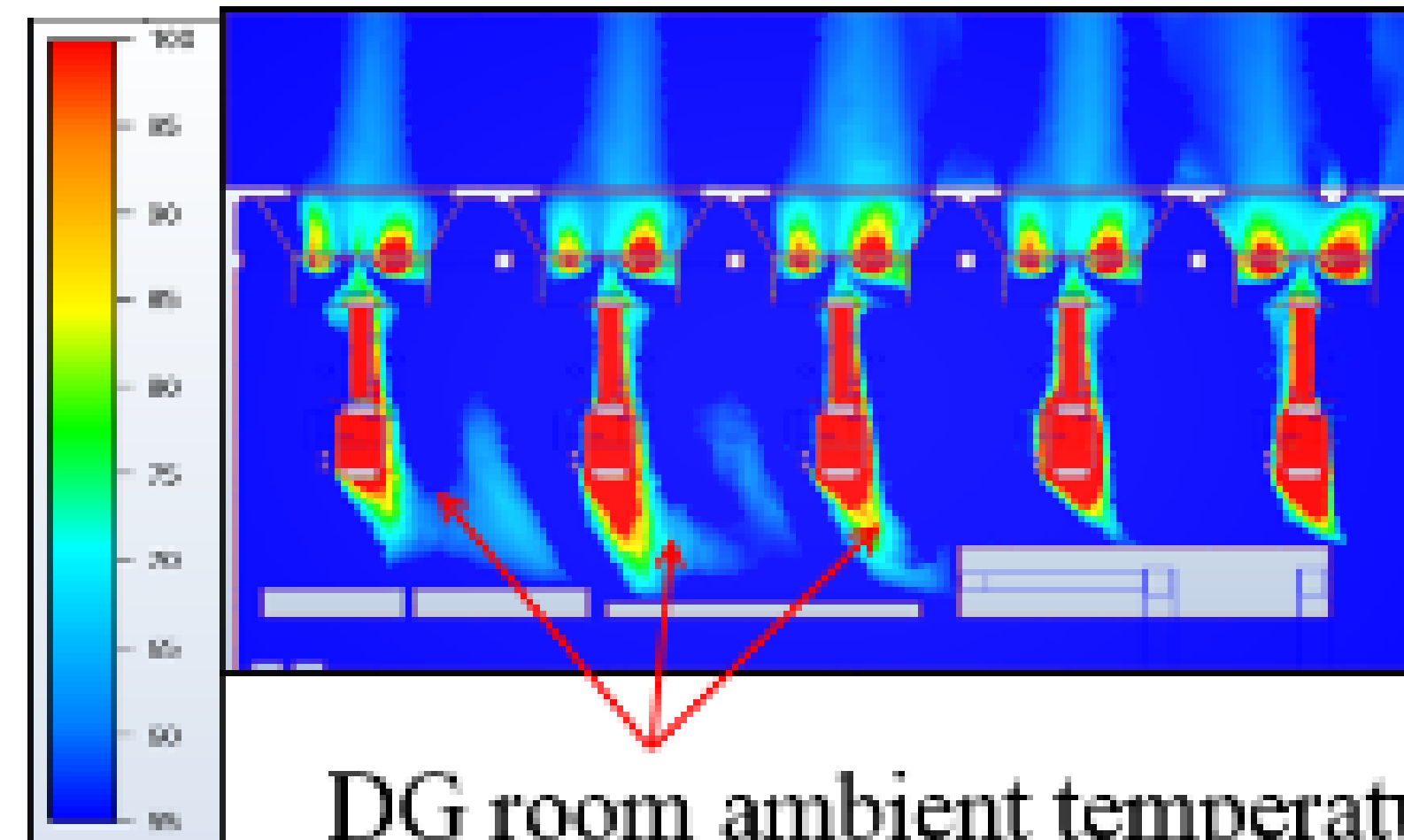
Air Exhaust  
from DG  
room

2000 KVA  
DG Engines

Top inlet for fresh air entry results in air flow  
by-passing alternator and engine



Hot air  
recirculation

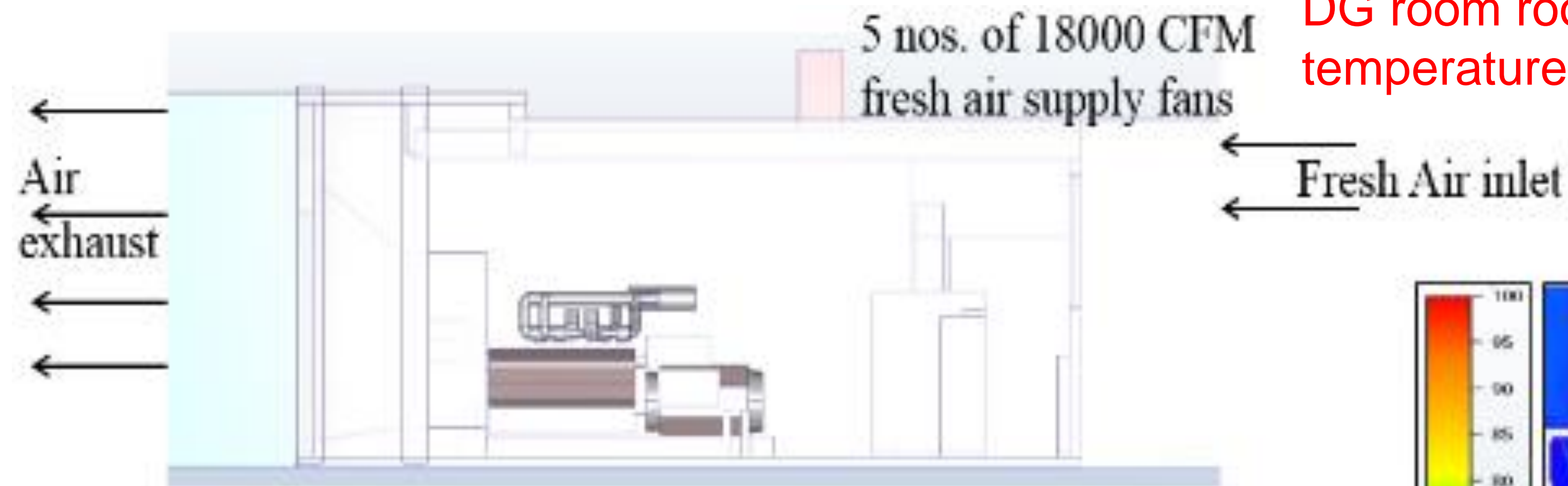


DG room ambient temperature  
above 55° C in these areas (NOT  
ACCEPTABLE)

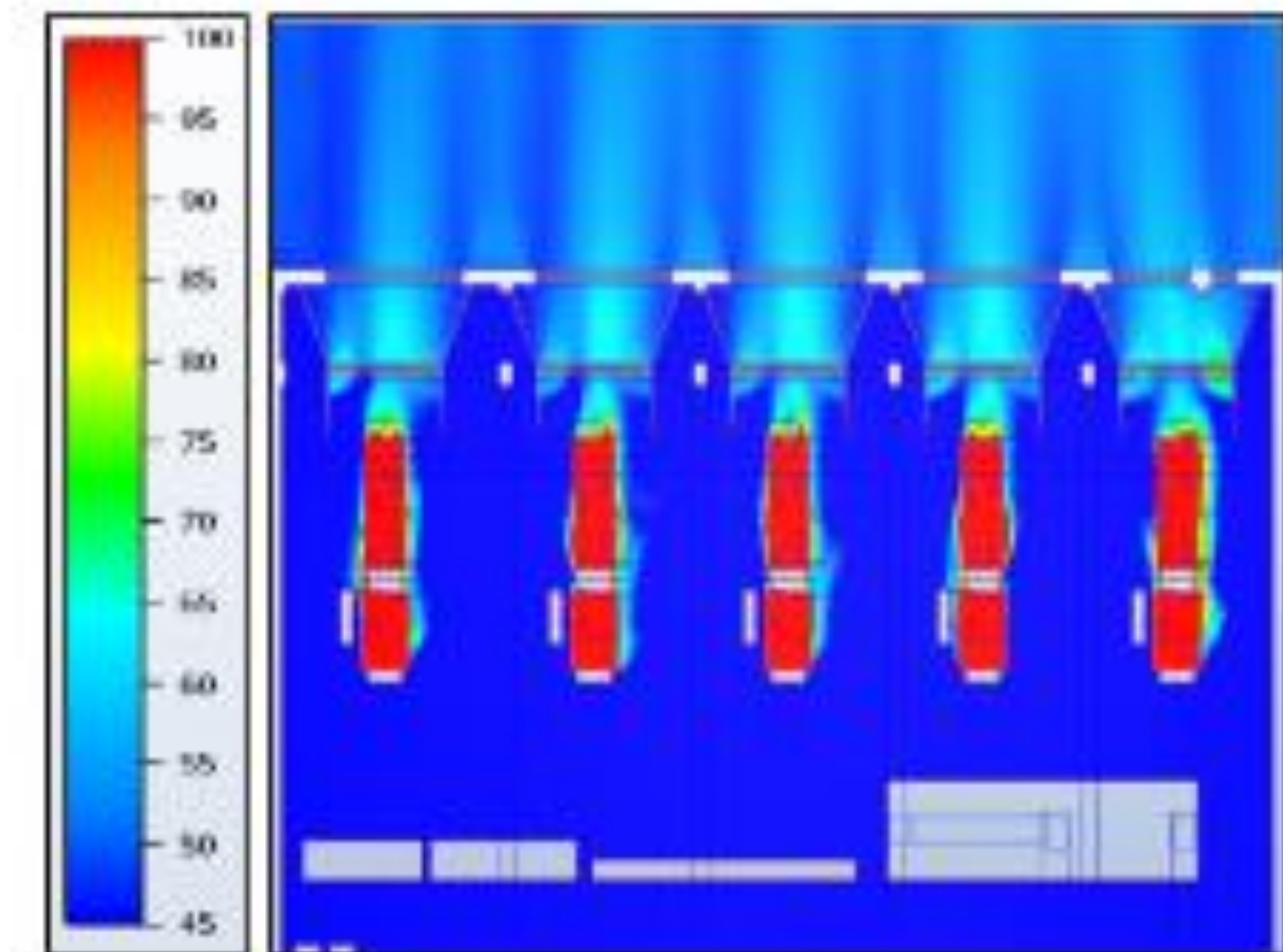
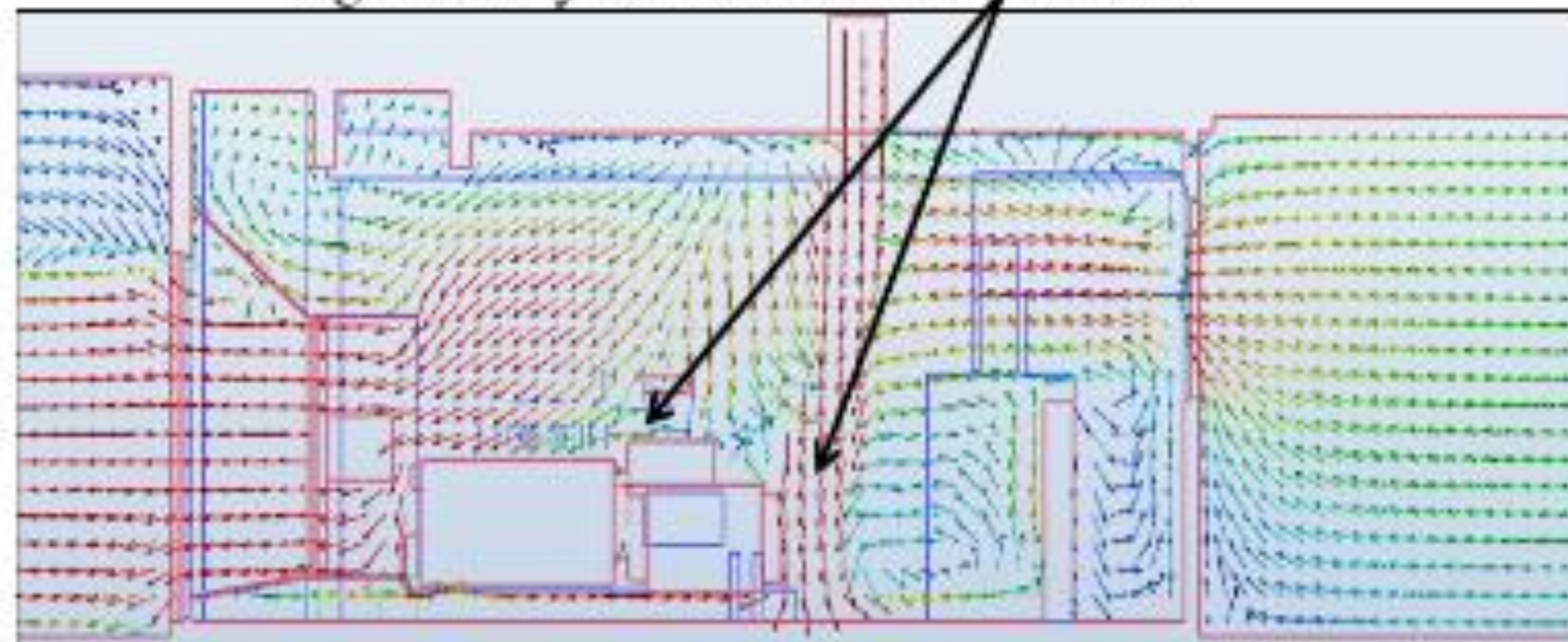


# Design option-1:CFD analysis results for roof supply fans

Mechanical ventilation through supply fans at the DG room roof (expensive), though room ambient temperature was brought down to 45° C



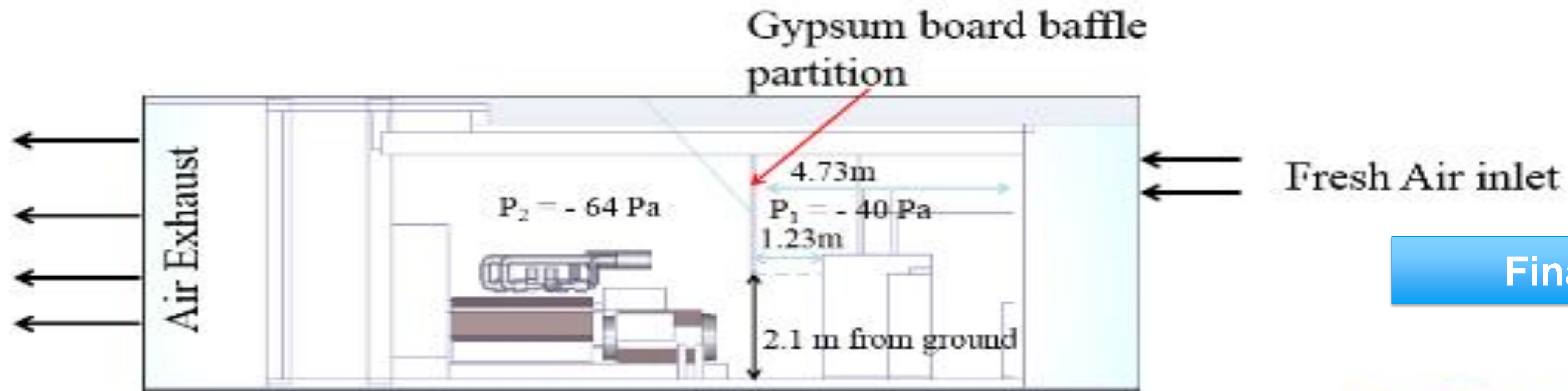
Air flow to the engine and alternator improved significantly due to fan flow from roof



DG room ambient temperature below 45 Deg C

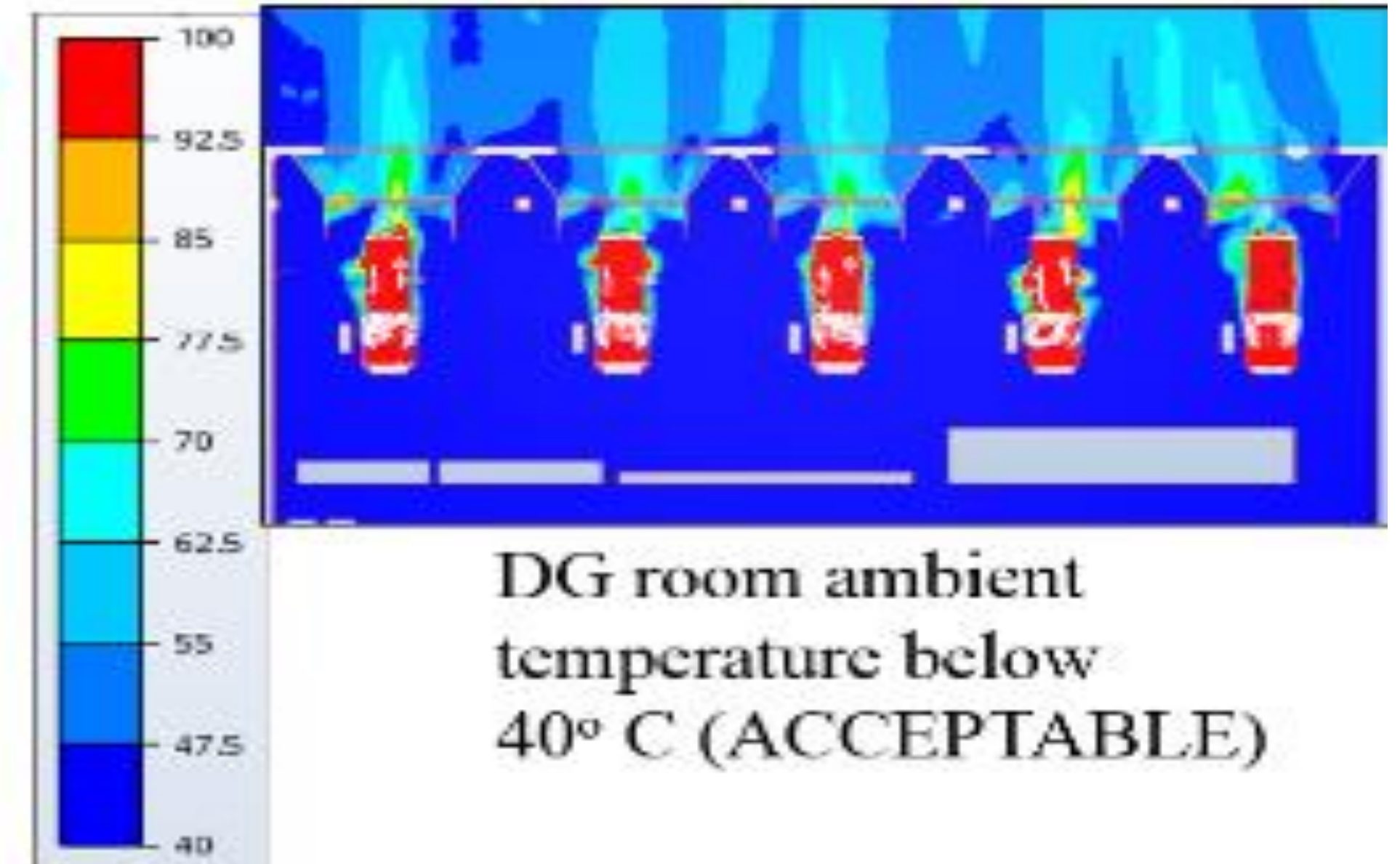
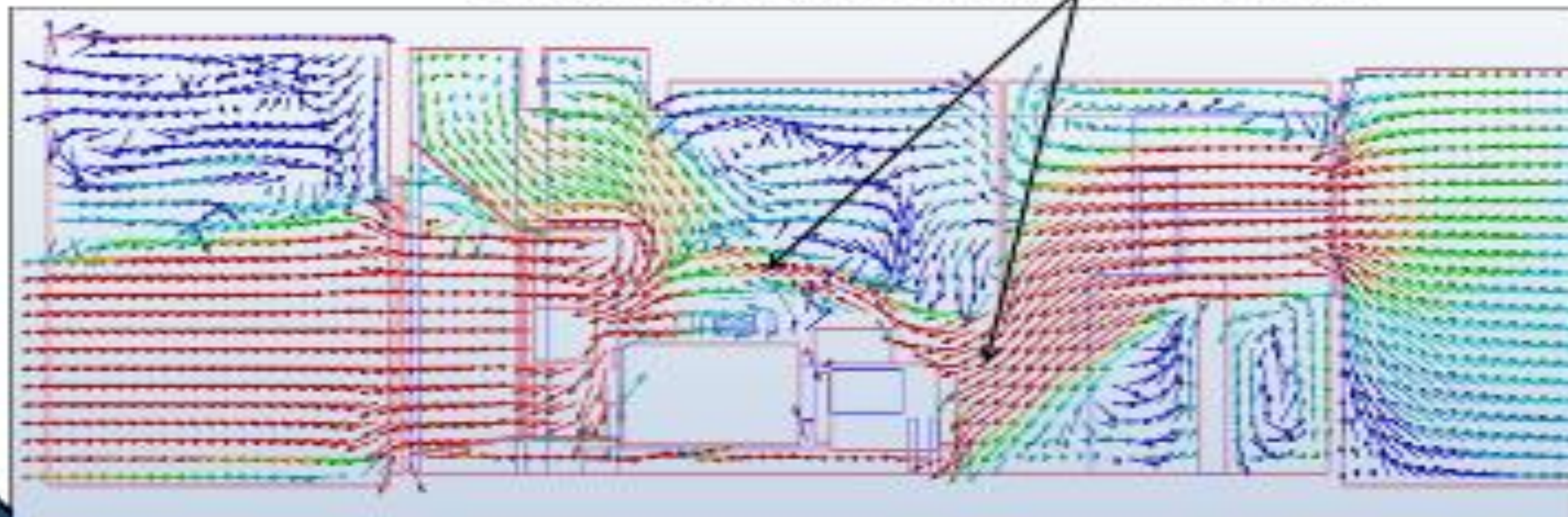


# Design option-2: CFD analysis results with baffle partition



**Final Design**

Air flow to the engine and alternator improved due to baffle partition placed inside DG room before alternator





# Outcome- DG room

## Design Option 2 results in:

- DG room ambient temperature below 40<sup>0</sup> C
- Bottom to top air flow over alternator and engine surfaces
- Continued availability of back up power after  
Implemented at site
- Avoidance of warranty and performance issues.

# Simulation Strategies and Techniques

## Smoke Visibility and Extraction





# How does CFD simulation of smoke visibility help?

## Need:

- Will traditional design approach based on ACH work?
- Smoke exhaust capacity based on fire size, not by ACH

## Goal of CFD simulation:

- Smoke visibility
- Smoke temperature
- Air/smoke flow fields

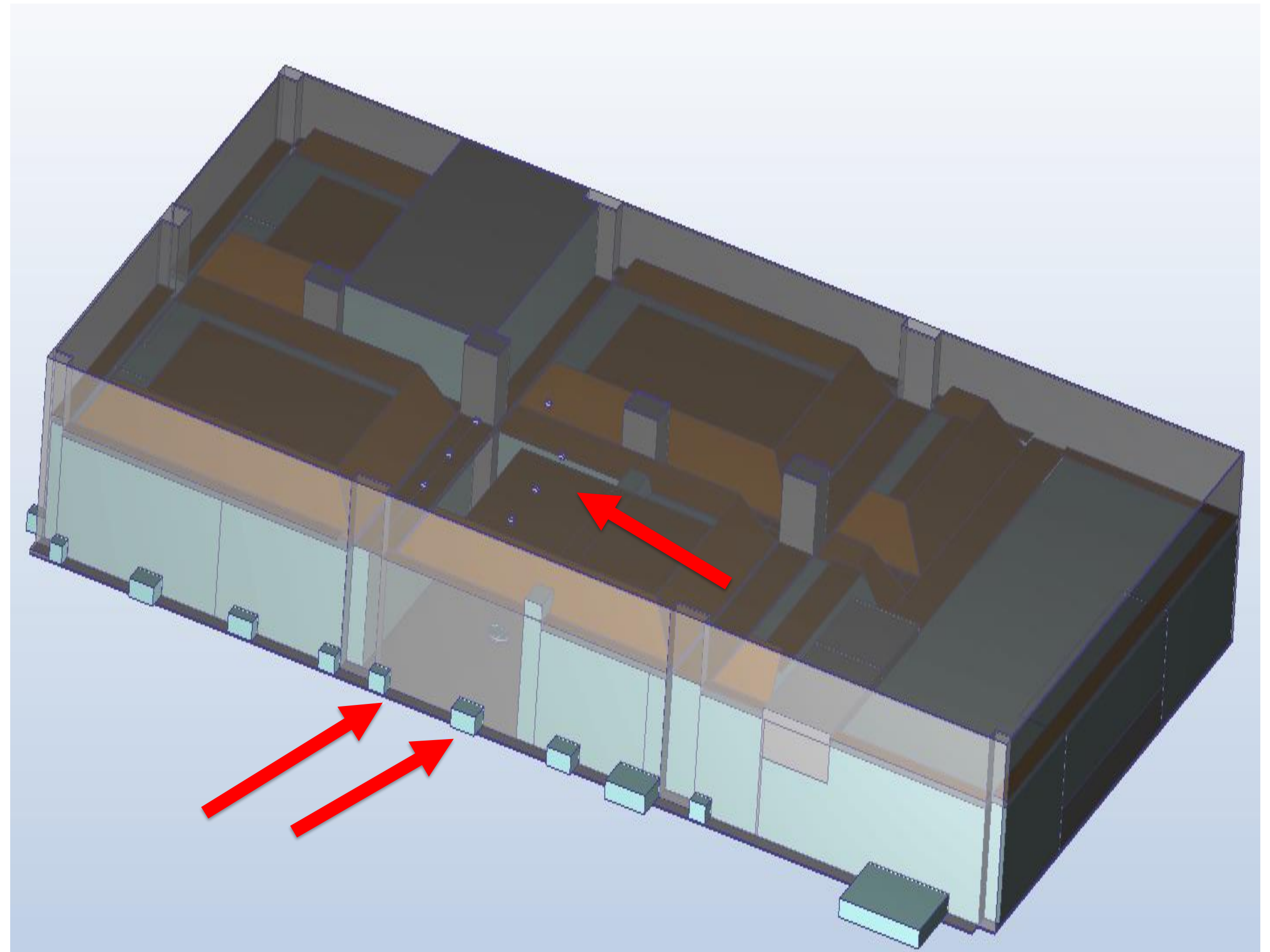
## Design performance criteria (BS PD 7974-6:2004 Annex G, Table G.1 and G.3):

- Smoke visibility of minimum 10 m
- Smoke temperature limit of 60<sup>0</sup> C

## CAD Model

Create CAD model in Revit.

- Simplify geometrical details
- Add inlet and outlet extensions
- Create fire part



Exhibition hall size = 245L X160W X22.4H

No. of smoke zones = 10

Zone size for CFD = 80m X 40 m

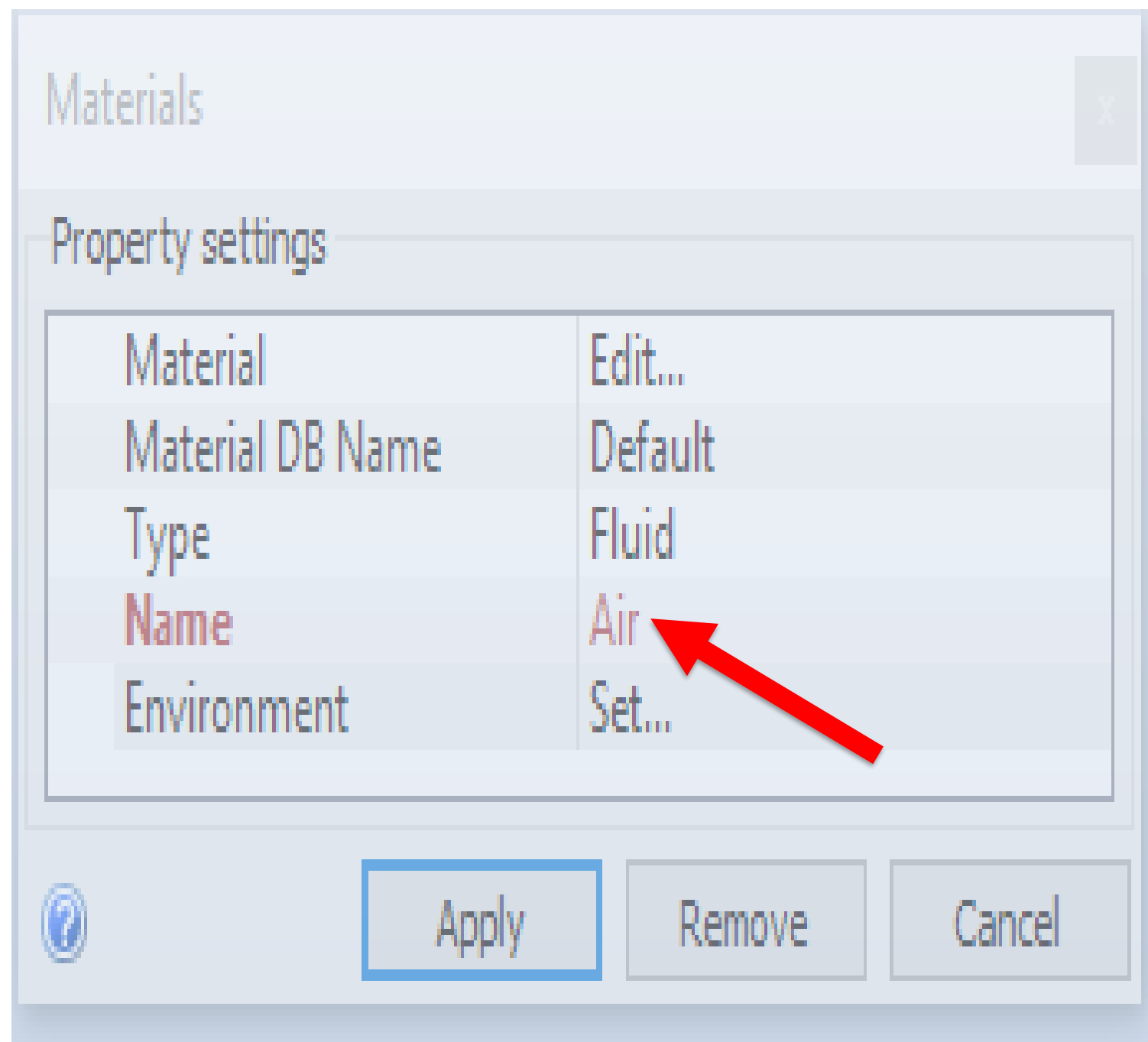
Fire mode ventilation design= 12 ACH



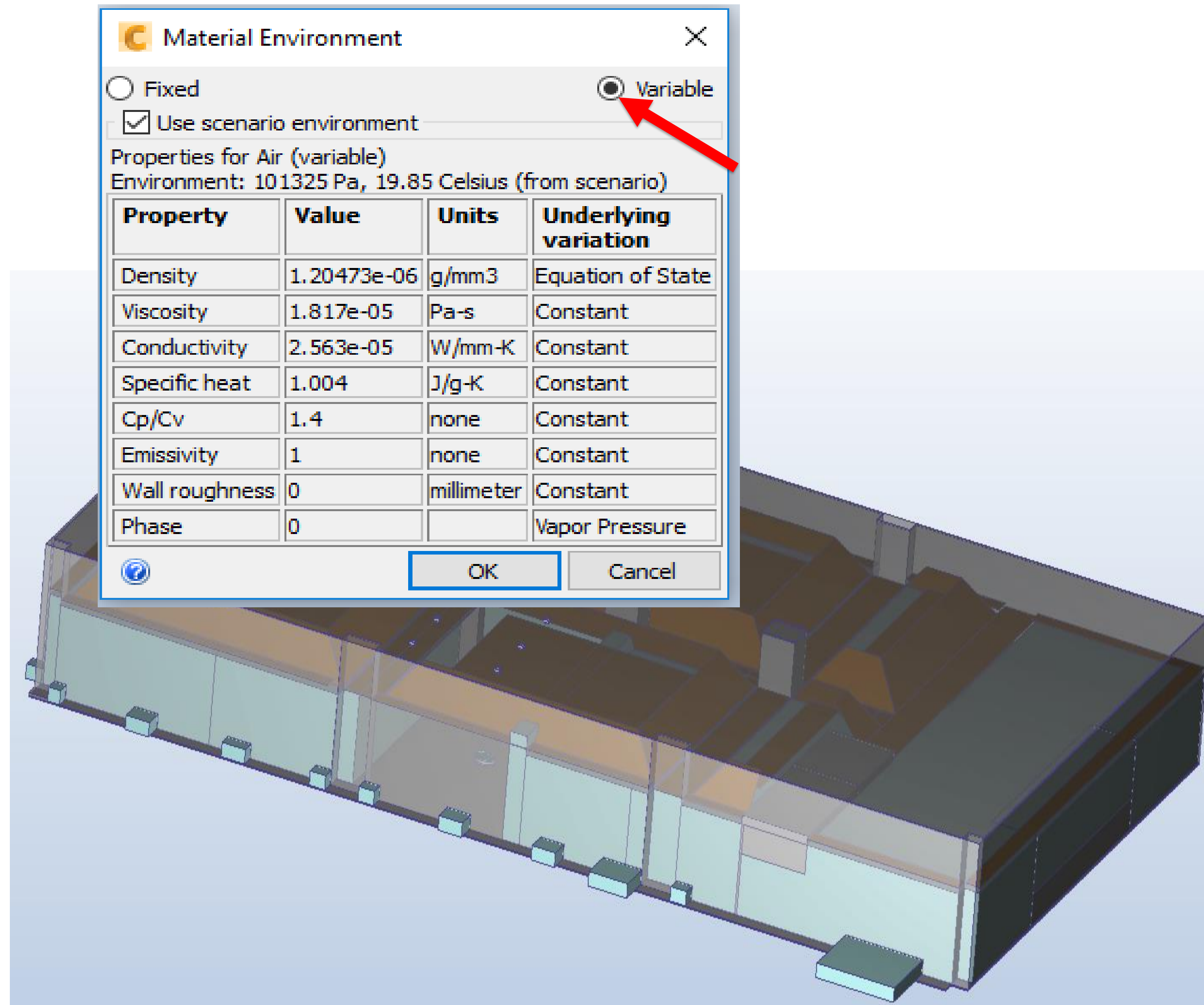
# Materials



# Air domain material



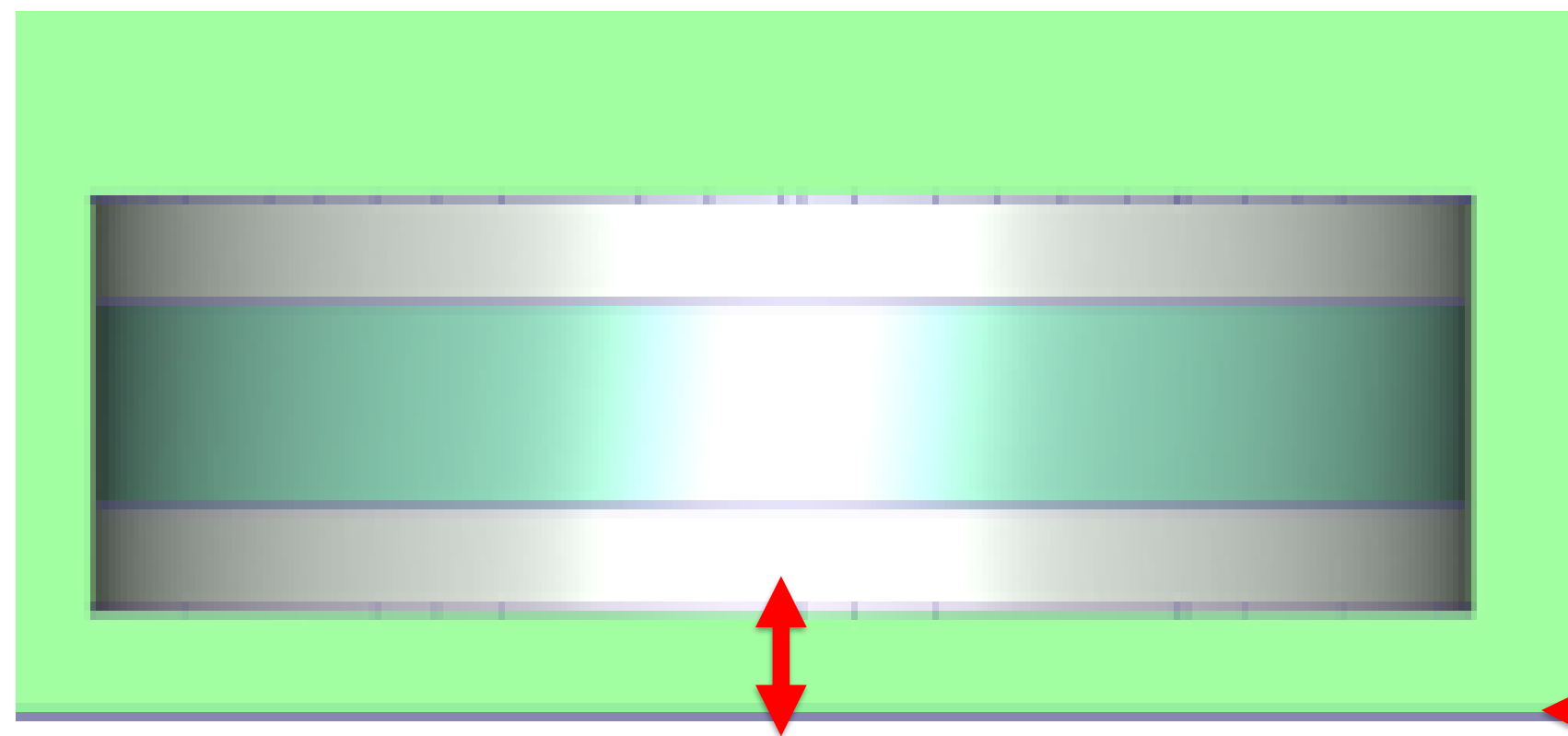
- All air volumes set to 'variable'



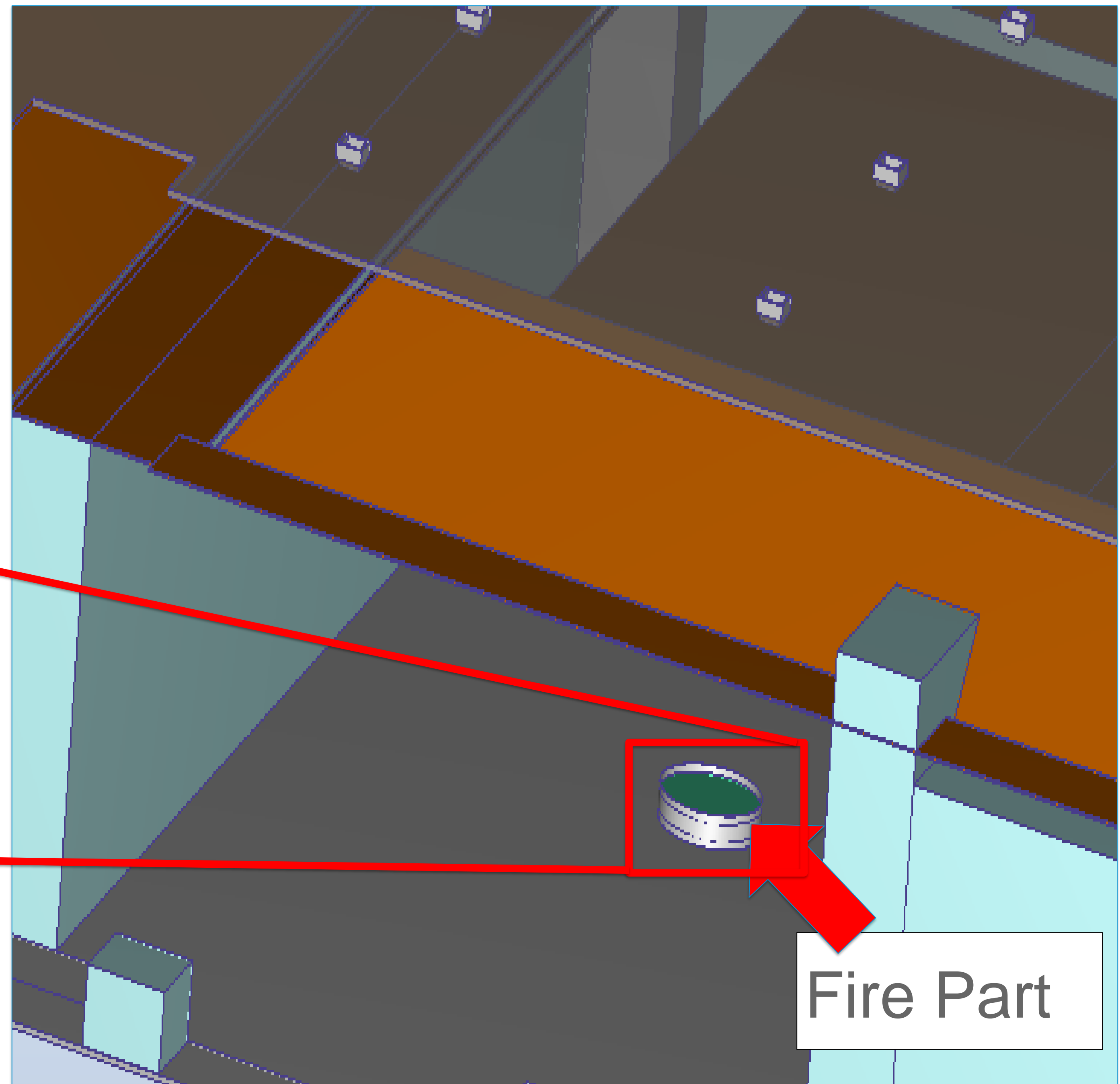


## Fire Part material

- Resistance with 0.85 FAR
- Conductivity of 200 W/m-K
- Solid ring suppressed
- Flow direction set to vertical



gap underneath the fire for  
cool air to be drawn in



# Boundary Conditions

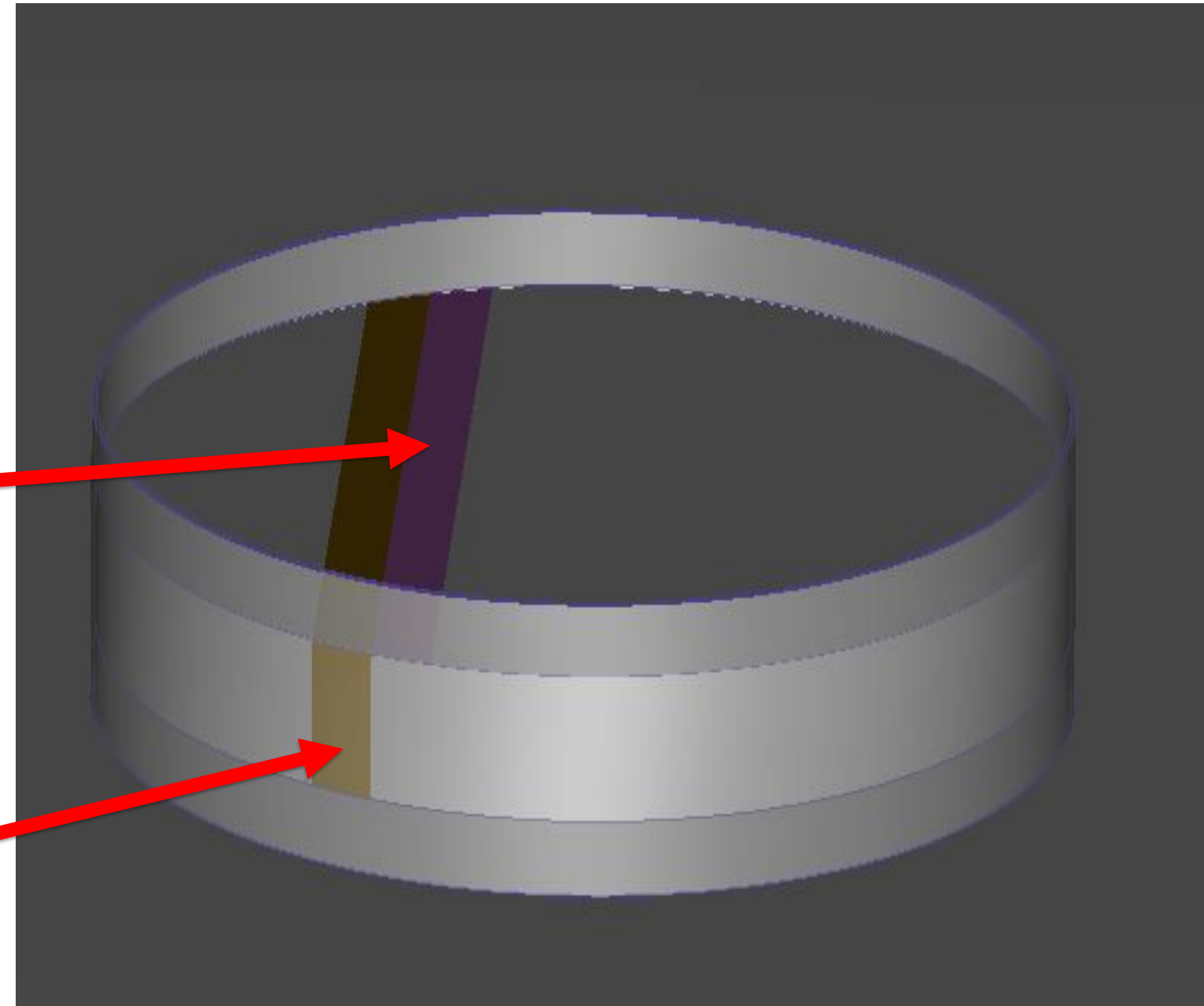


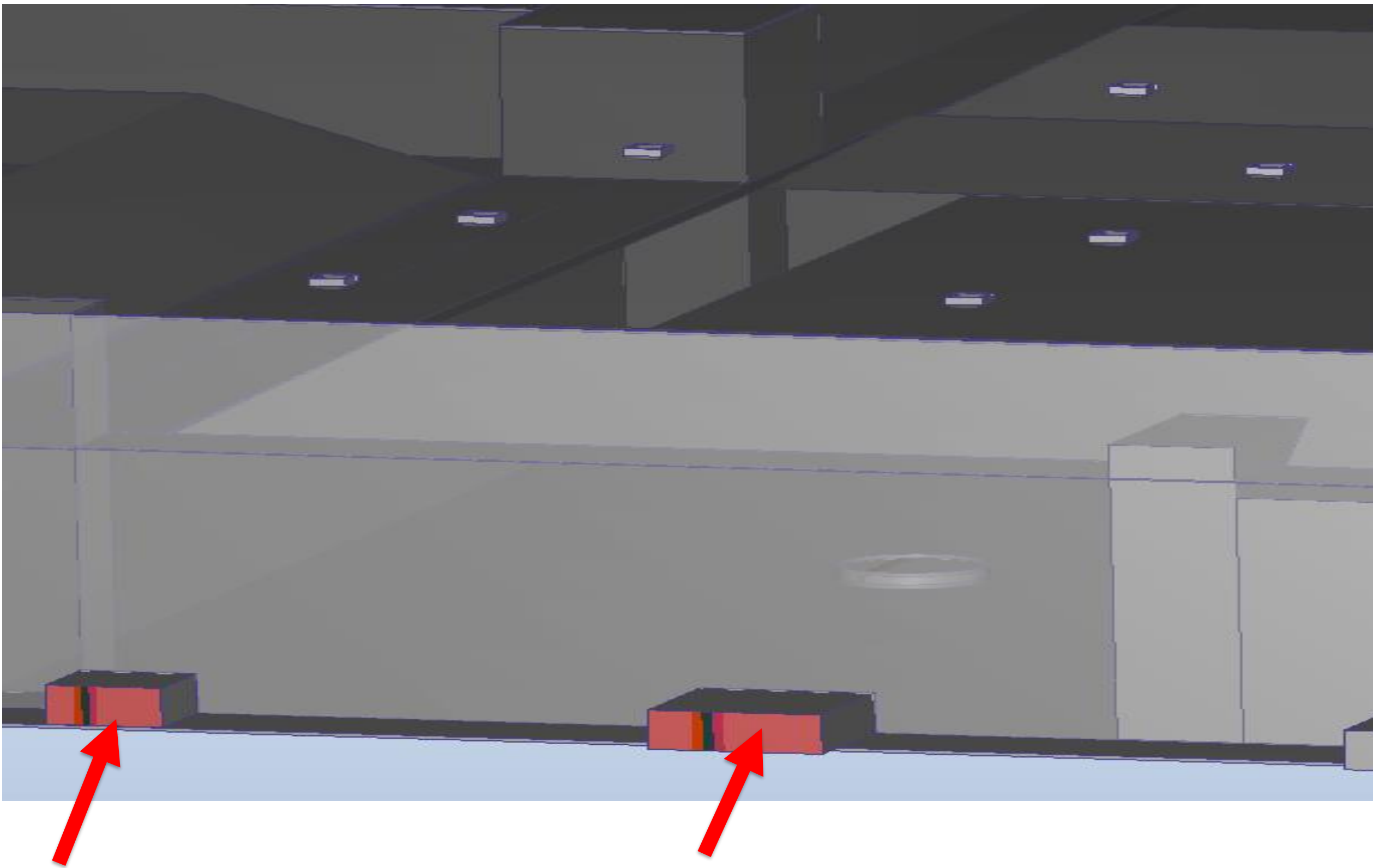


# Heat source and scalar BCs

## Fire part

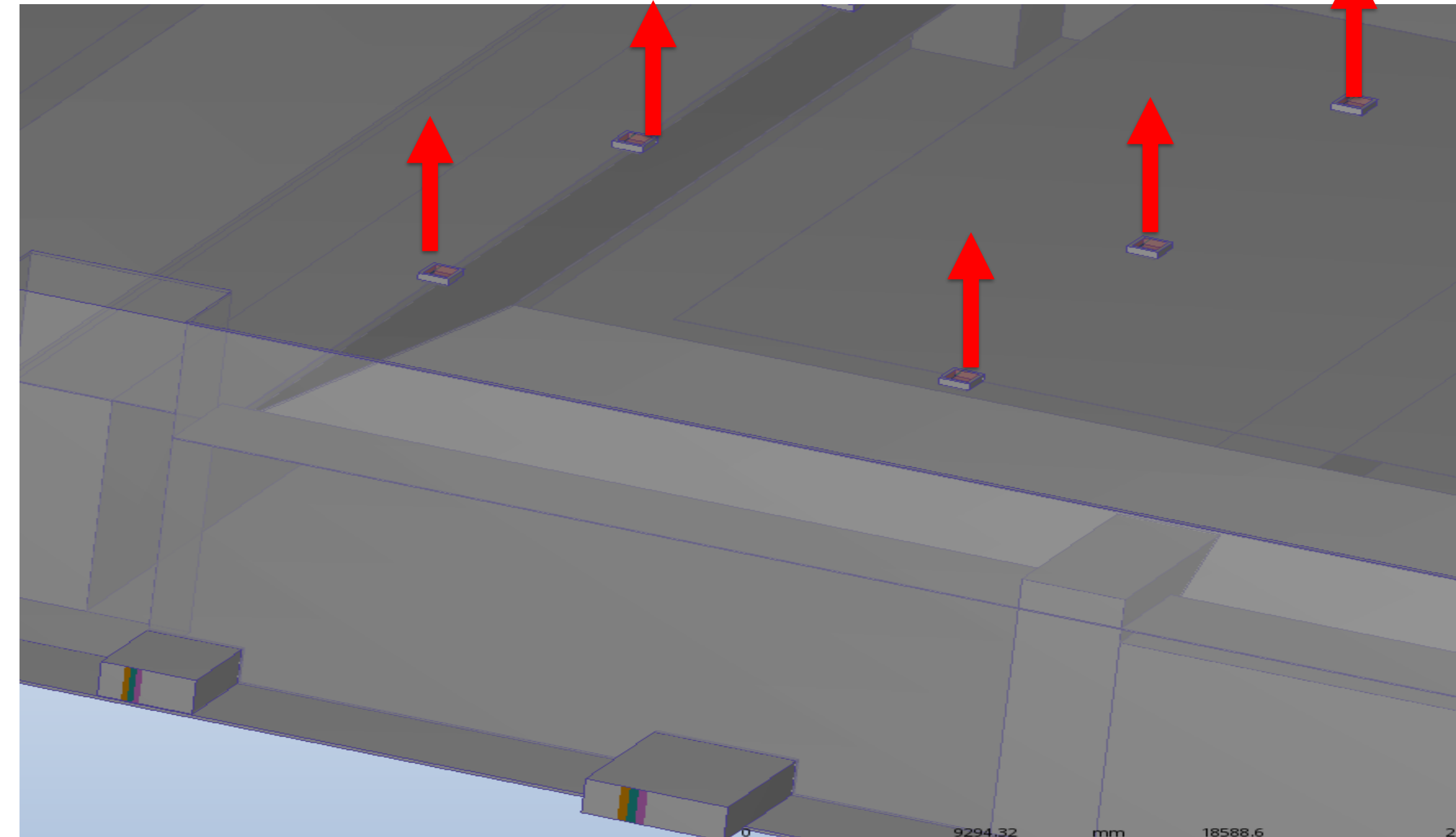
- Scalar of 1 applied to its top surface. Scalar 1 represents 100% smoke
- Total heat generation to its entire volume,





## Inlet BC

- Scalar (0) – represents clean air
- Temperature (Celsius)
- Pressure (0 Pa)



## Outlet BC

- Volume Flow Rate  
(ft<sup>3</sup>/min)



# Meshing

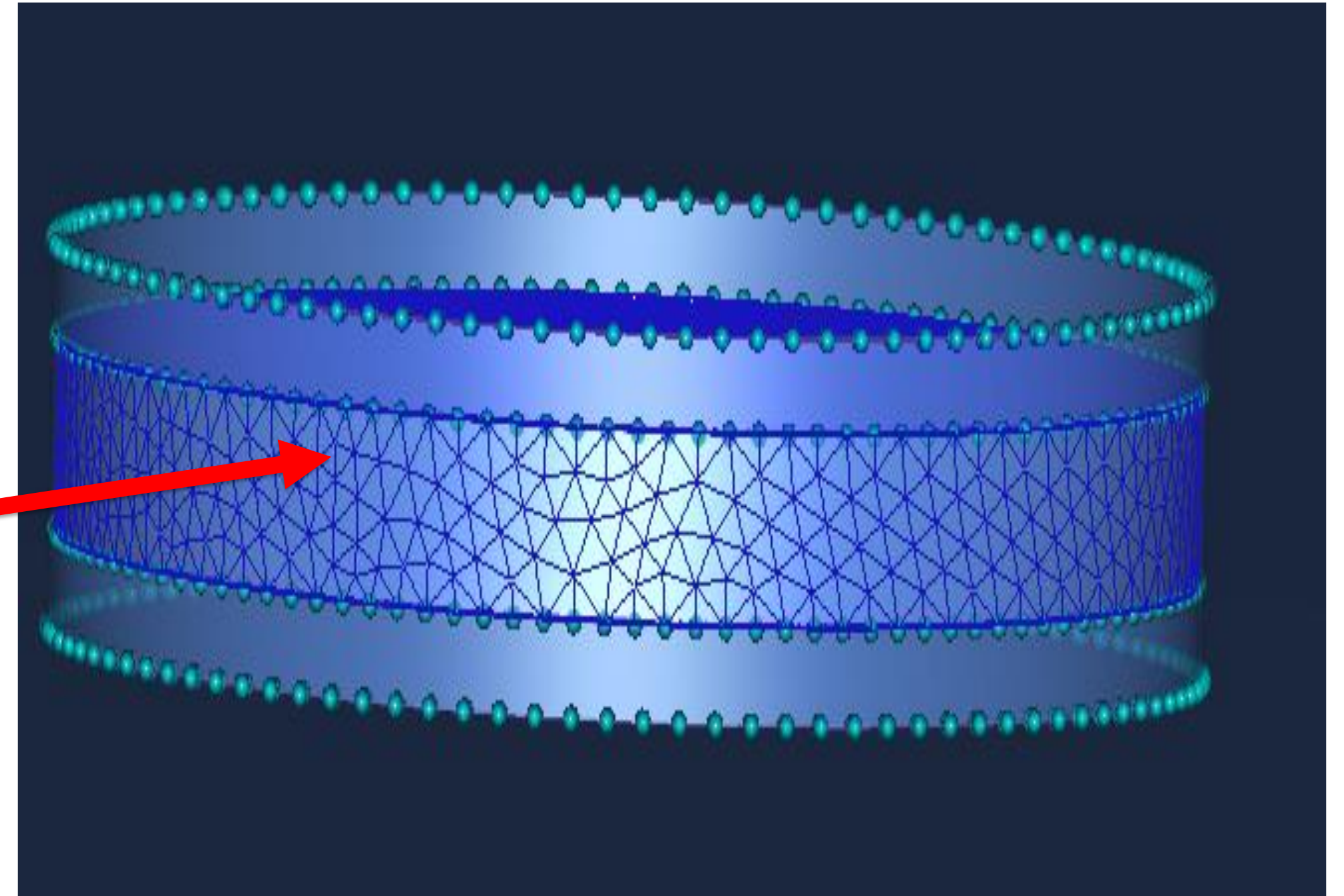




# Meshing of fire part

To capture smoke flow and heat release from fire source

- Fire part - uniform mesh 4-5 elements
- Air above and below - a good uniform mesh

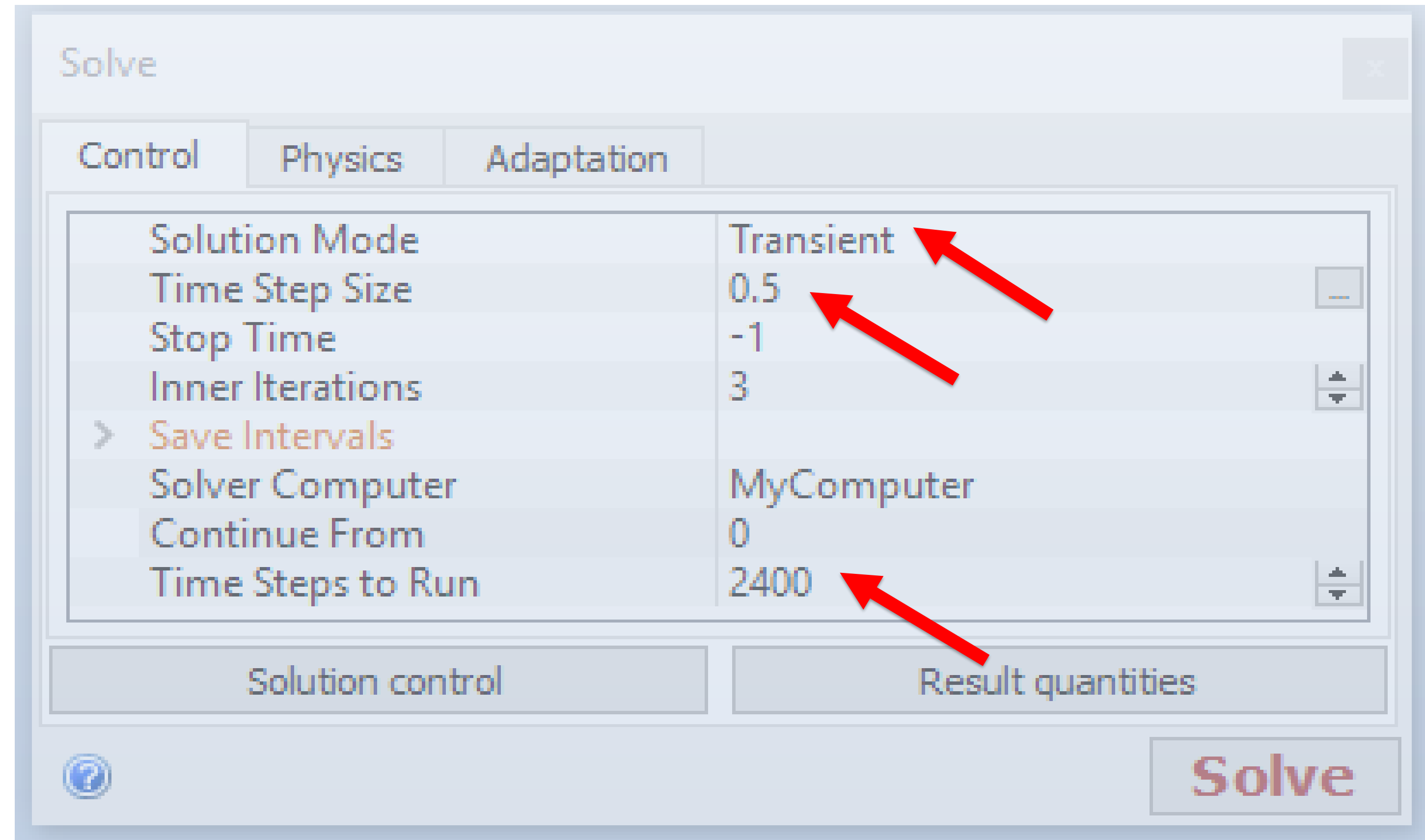
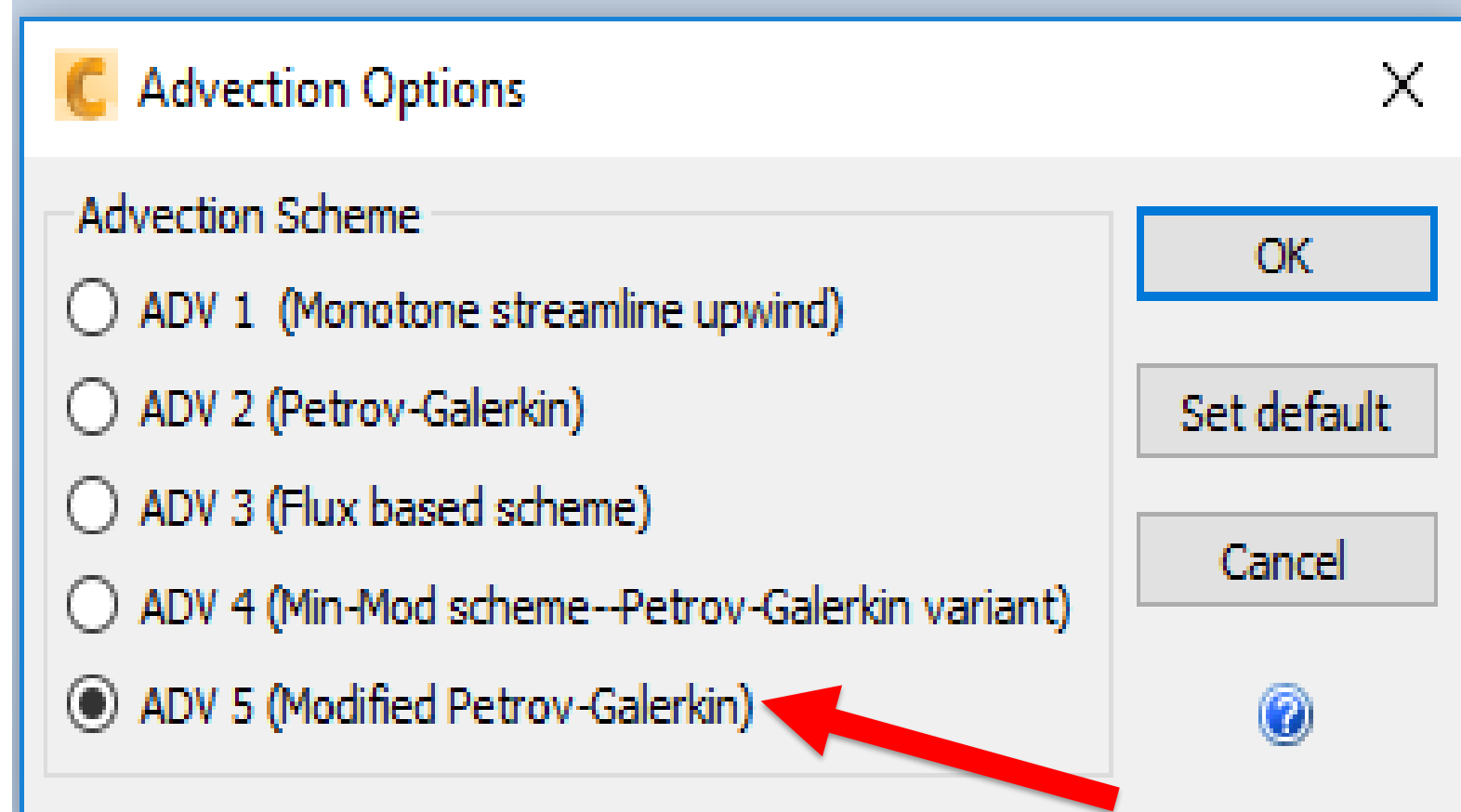
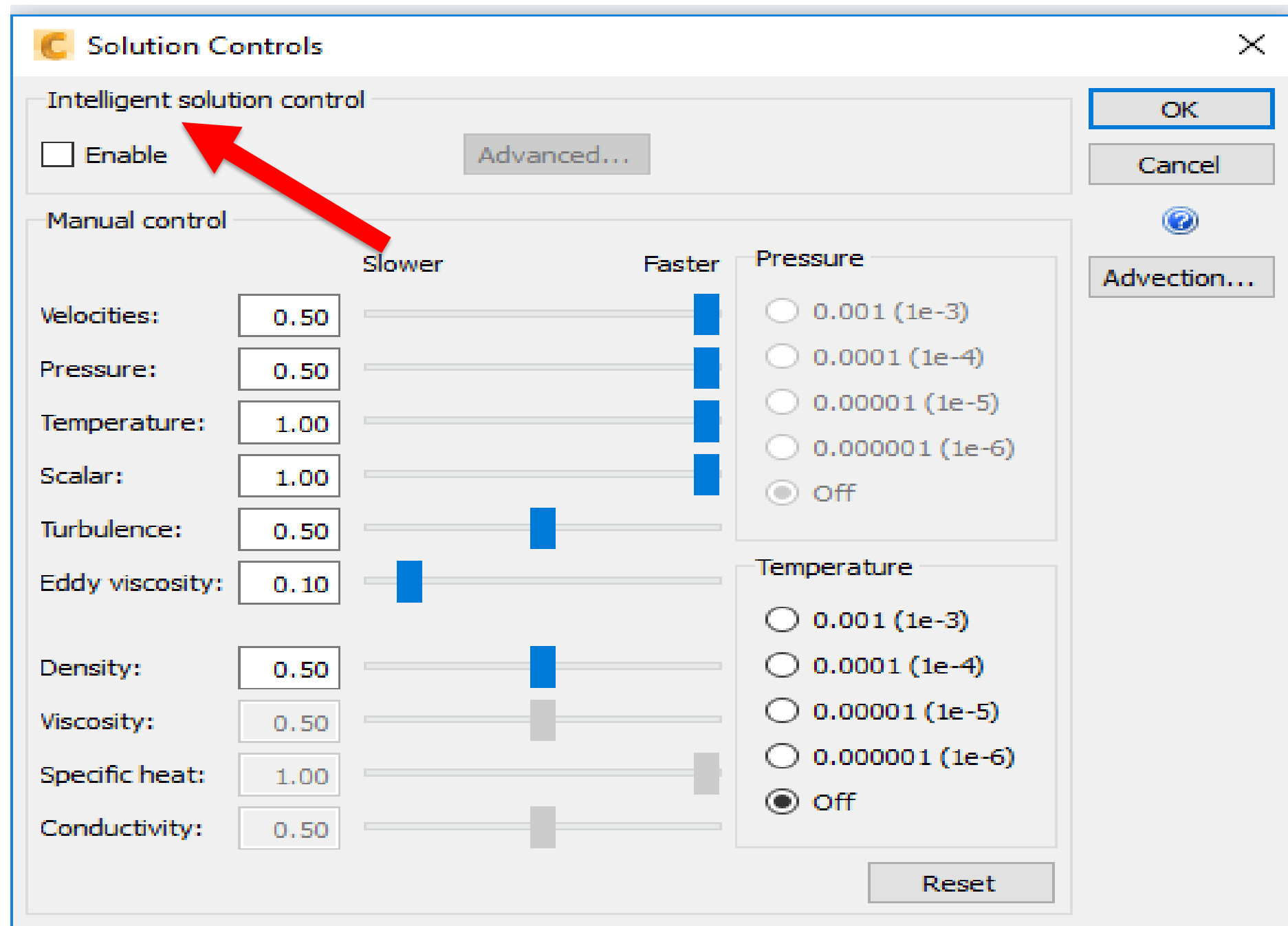




# Solver Settings



# Solver settings for smoke analysis (Intelligent control-off, Advection 5)



Transient analysis with time step size specified



# Solver settings for smoke analysis (Smoke visibility parameters)

**Result Quantities**

**Basic flow values**

- ☒ Velocity
- ☒ Pressure
- ☒ Temperature
- ☒ Scalar

**Fluid properties**

- ☒ Density
- ☐ Viscosity
- ☐ Conductivity
- ☐ Specific heat

**Turbulence**

- ☐ Turbulent kinetic energy
- ☐ Turbulent energy dissipation
- ☐ Effective viscosity
- ☐ Effective conductivity
- ☐ Wall model y +
- ☐ Wall shear stress
- ☐ Turbulent intensity

**Compressible**

- ☐ Mach number
- ☐ Total pressure
- ☐ Total temperature

**Miscellaneous**

- ☐ Wall forces
- ☐ Absolute velocity
- ☐ Absolute static pressure
- ☐ Vorticity
- ☐ Strain rate
- ☐ Pressure coefficient
- ☒ Shear stress
- ☐ Temperature gradient
- ☐ Stream function
- ☐ Local mean age (LMA)
- ☒ Wall heat flux
- ☒ Wall film coefficient
- ☐ Thermal comfort
- ☒ Smoke visibility

**Buttons:** All, None, Options..., Factors..., Parameters..., OK, Cancel

## Results Quantities

- Visibility On

**Smoke Visibility Parameters**

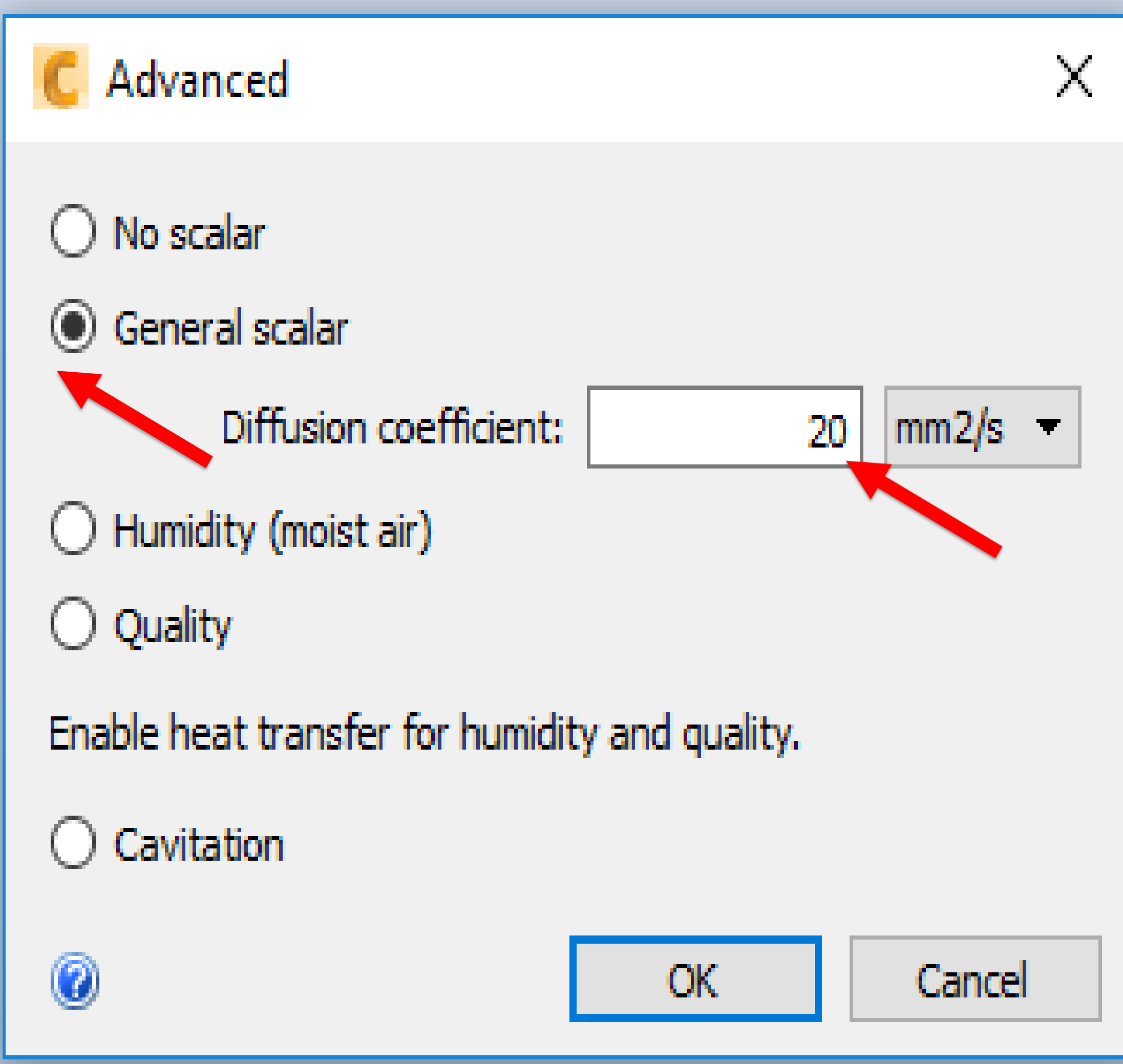
Extinction coefficient: 37000 ft<sup>2</sup>/lb

Sign visibility constant: 8

Combustion particulate yield: 0.1

**Buttons:** OK, Cancel

# Solver settings for smoke analysis (Inputs on smoke as scalar)



Advanced

☐ No scalar

☒ General scalar

Diffusion coefficient:  mm<sup>2</sup>/s

☐ Humidity (moist air)

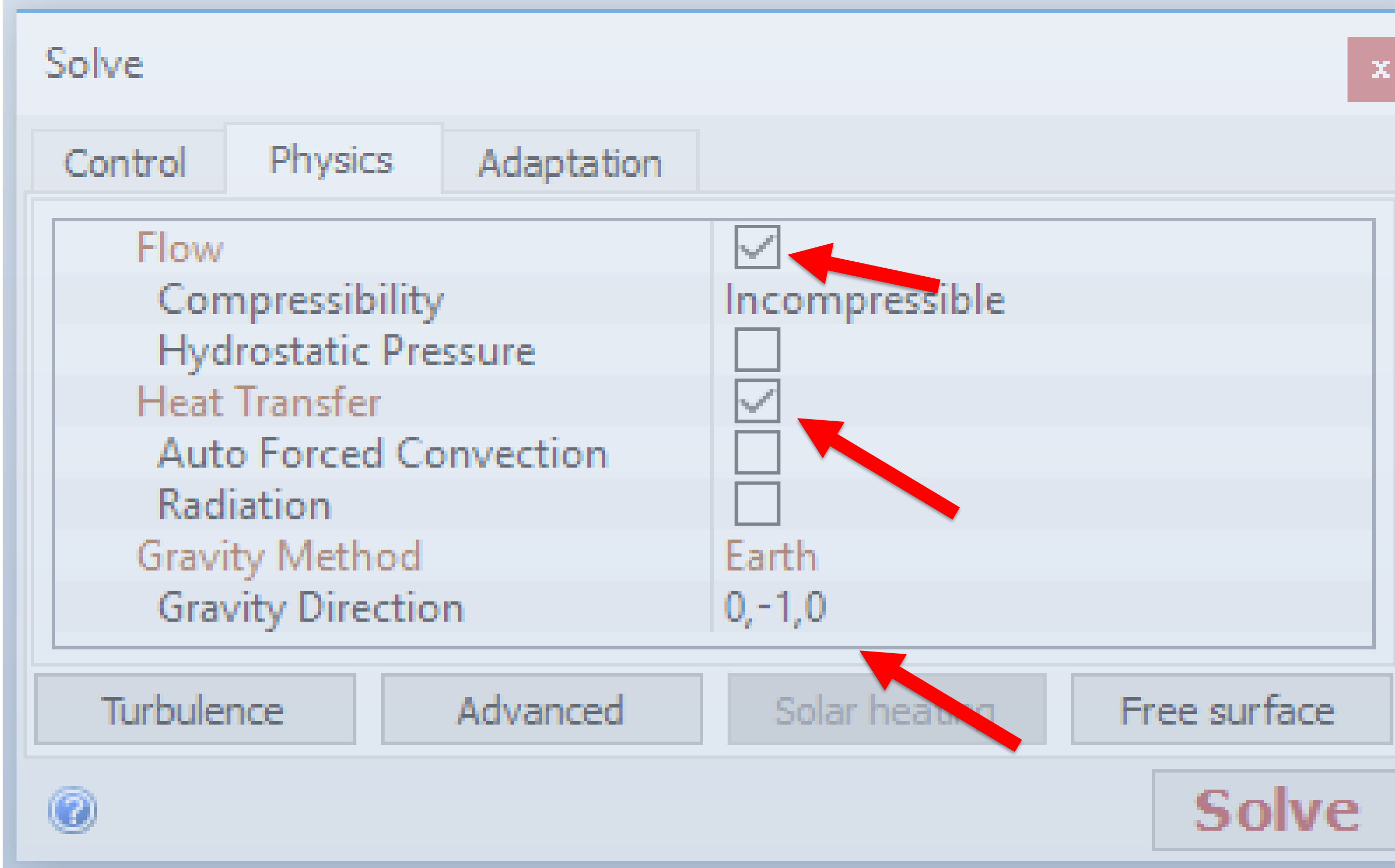
☐ Quality

Enable heat transfer for humidity and quality.

☐ Cavitation

OK Cancel

This dialog box is titled 'Advanced' and contains several radio button options. The 'General scalar' option is selected. A red arrow points to this option. Below it, the 'Diffusion coefficient' is set to '20' in a text box, with a unit dropdown menu showing 'mm<sup>2</sup>/s'. Another red arrow points to the '20' value. At the bottom, there are 'OK' and 'Cancel' buttons. A red arrow points to the 'OK' button.



Solve

Control Physics Adaptation

Flow	<input checked="" type="checkbox"/>
Compressibility	Incompressible
Hydrostatic Pressure	<input type="checkbox"/>
Heat Transfer	<input checked="" type="checkbox"/>
Auto Forced Convection	<input type="checkbox"/>
Radiation	<input type="checkbox"/>
Gravity Method	Earth
Gravity Direction	0,-1,0

Turbulence Advanced Solar heating Free surface

Solve

This dialog box is titled 'Solve' and has three tabs: 'Control', 'Physics', and 'Adaptation'. The 'Physics' tab is active. It contains a table of settings. The 'Flow' checkbox is checked, with a red arrow pointing to it. The 'Heat Transfer' checkbox is also checked, with a red arrow pointing to it. The 'Gravity Method' is set to 'Earth' and the 'Gravity Direction' is set to '0,-1,0'. At the bottom, there are buttons for 'Turbulence', 'Advanced', 'Solar heating', and 'Free surface'. A red arrow points to the 'Solar heating' button. At the bottom right, there is a large 'Solve' button.

- Run Flow and Thermal together.
- Gravity set in downwards direction
- General Scalar ON .
- Non-zero value for the diffusion coefficient.



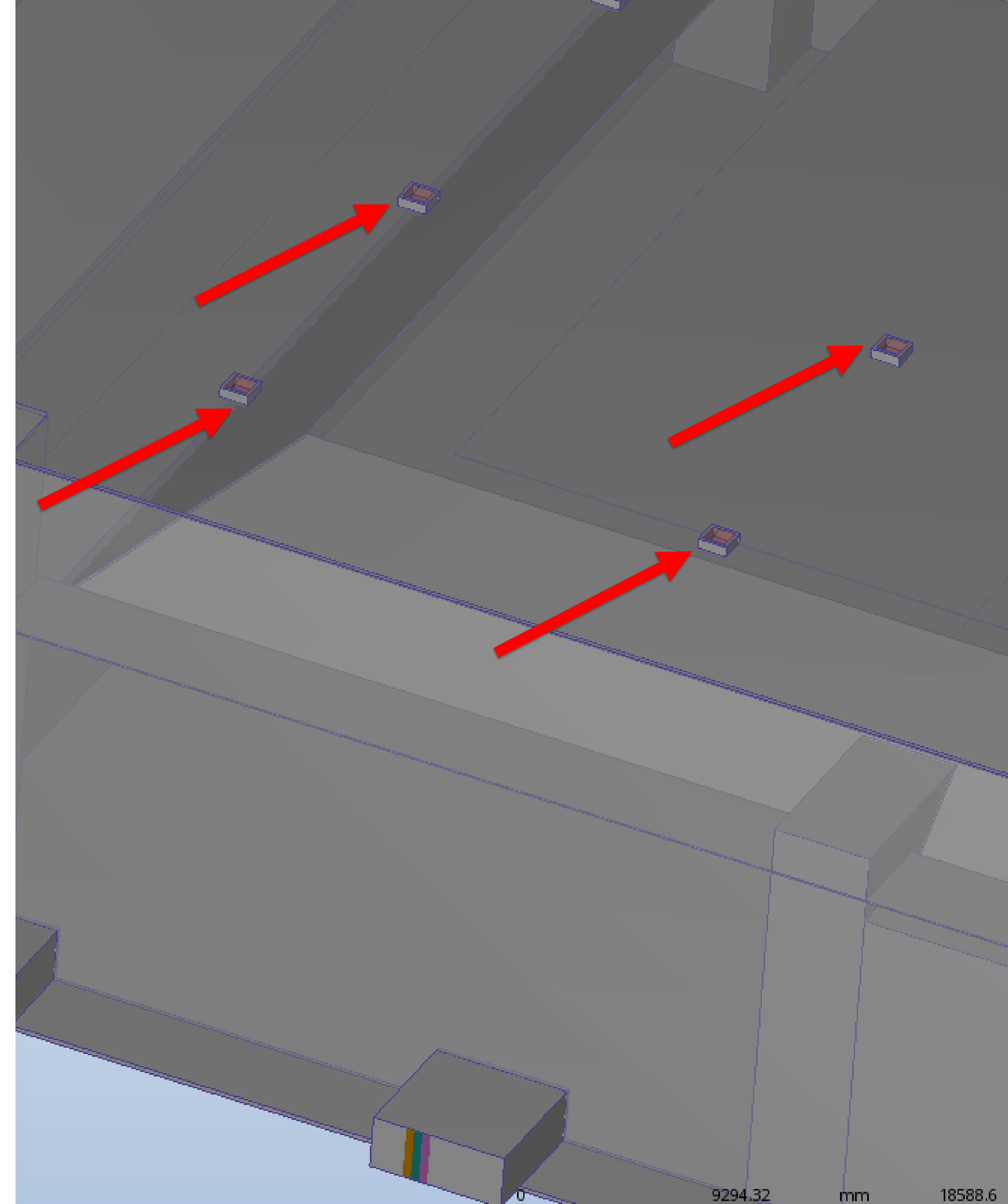
# Key Results for Design Performance



# Smoke Simulation Performance Evaluation

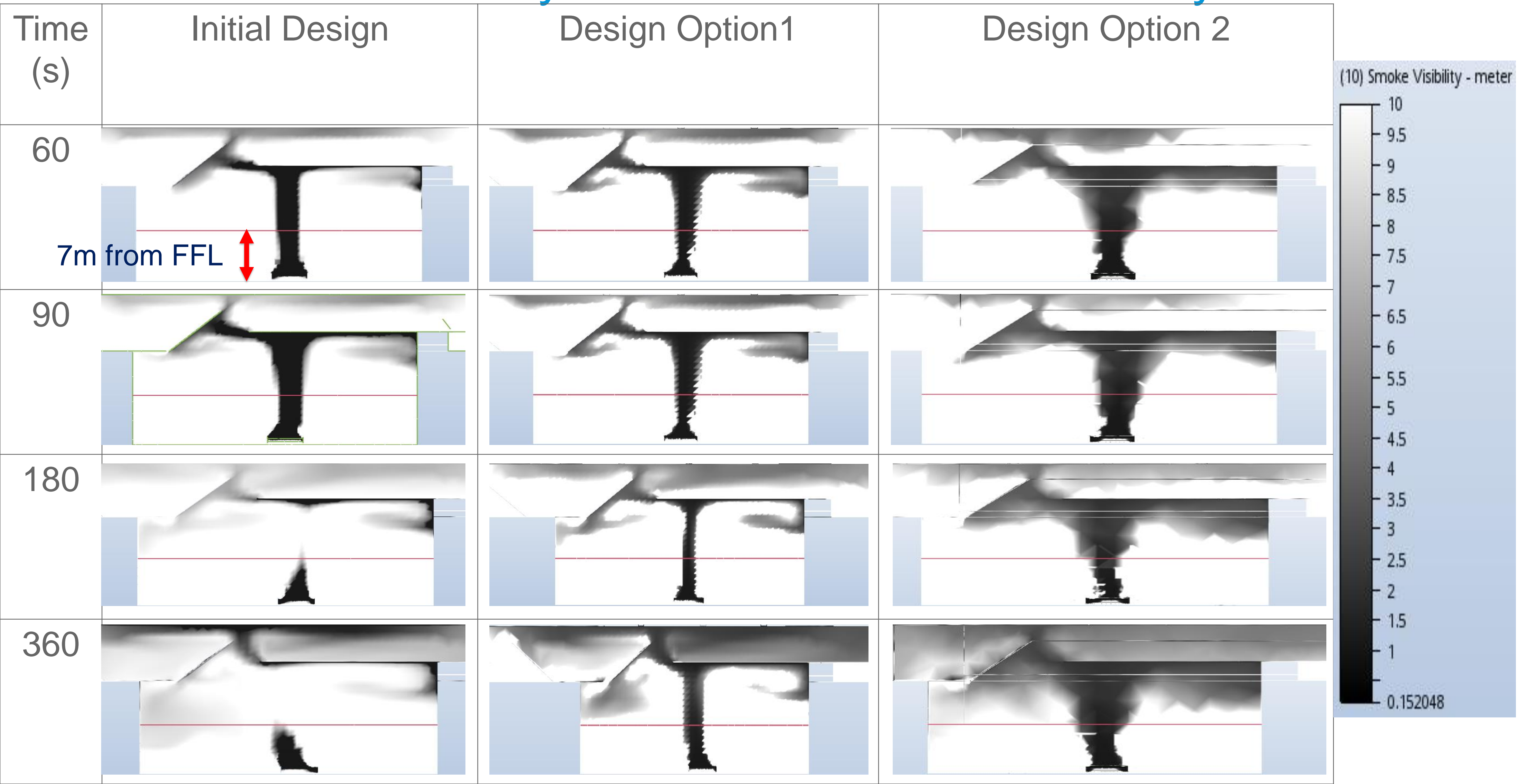
## Design Options based on Outlet BC:

- Initial Design: 20000 CFM
- Design Option 1: 15000 CFM
- Design Option 2 :12000 CFM



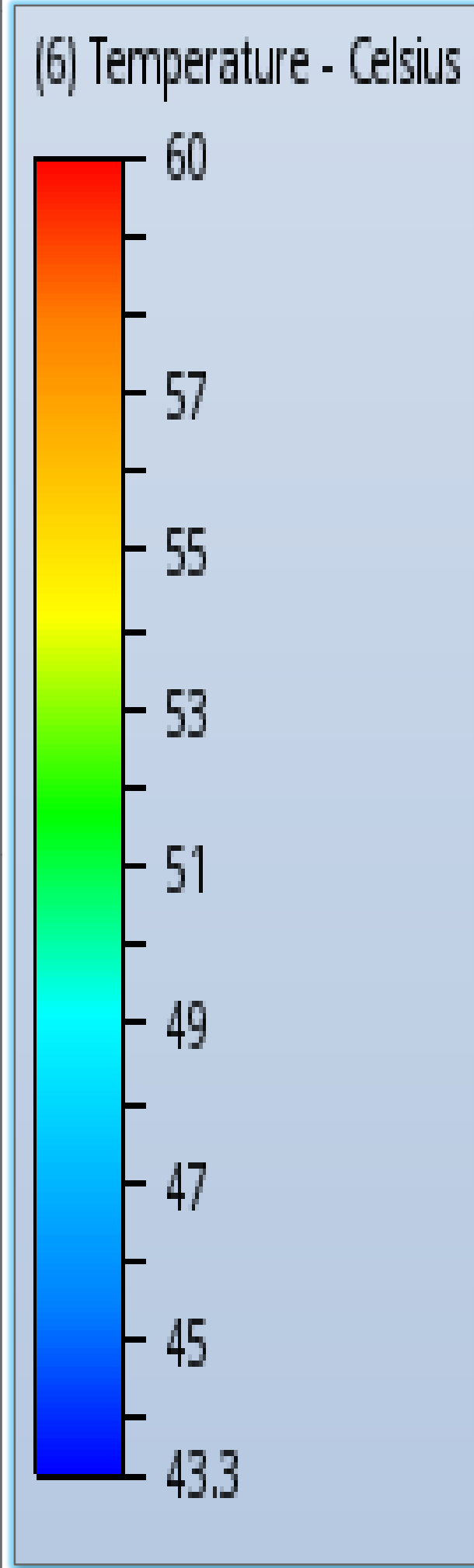
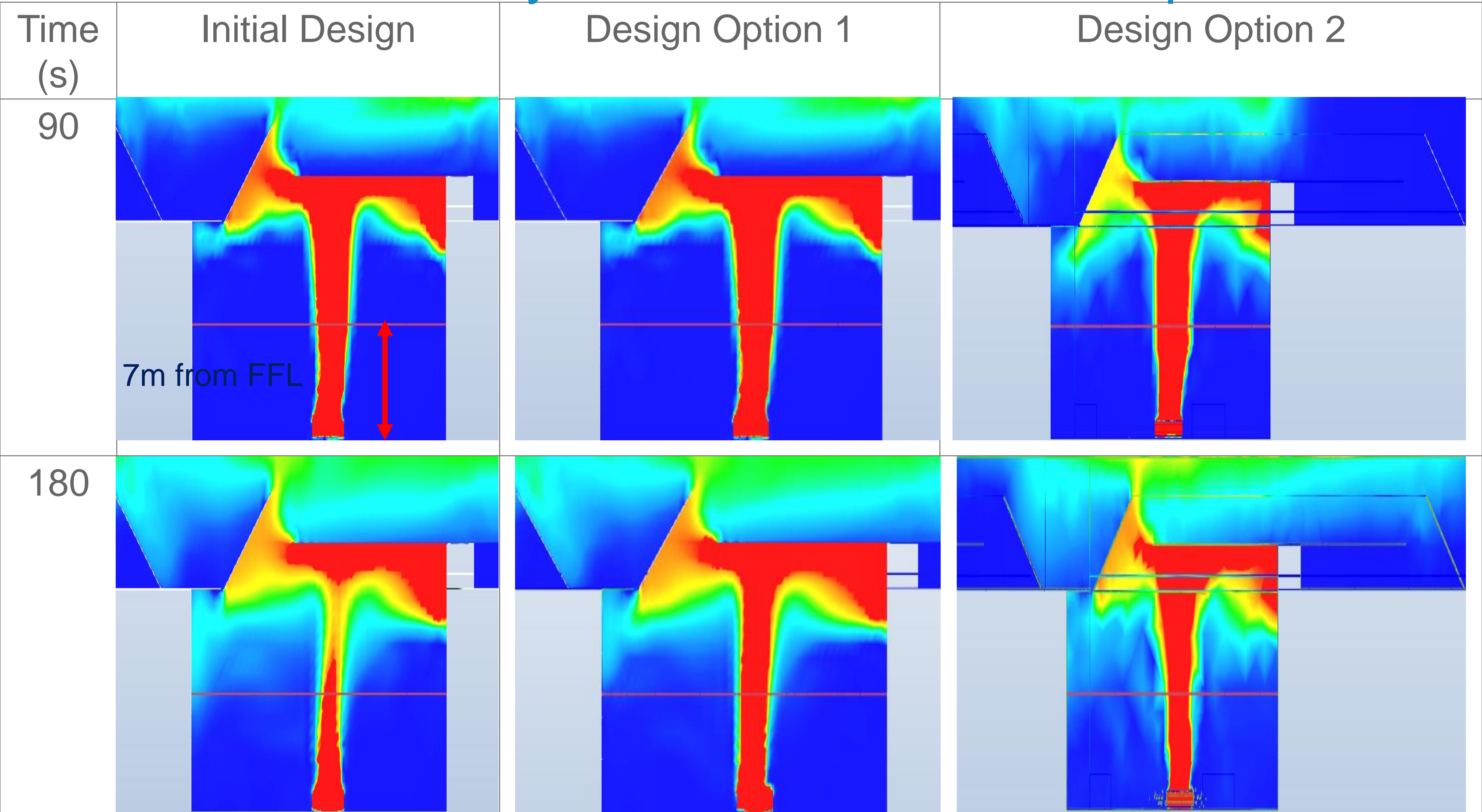


# CFD analysis results –Smoke Visibility



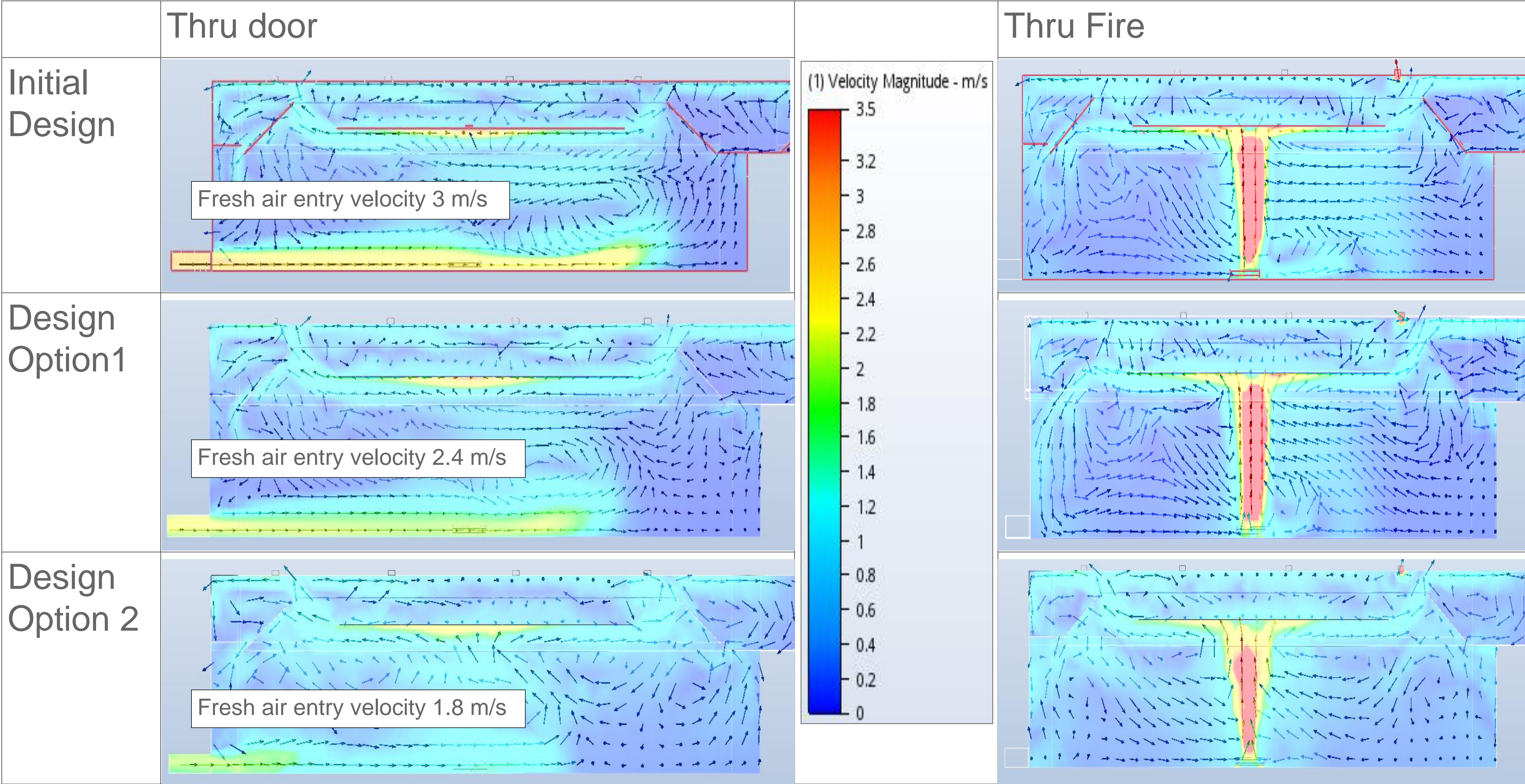


# CFD analysis results –Smoke Temperature





# CFD analysis results –Air flow field at 90 s from start of fire





# Outcome- Smoke simulation and extraction

For all design options:

- Smoke visibility of 10 m for a height of 7m achieved
- Smoke of temperature of less than 60<sup>0</sup> C achieved.
- Replacement/ makeup air velocity at entrance does not exceed 2.4 m/s (limit is 5 m/s)

*However, Design Option 2 (12000 CFM) is recommended due to cost effectiveness as well as less risk of smoke deflection*



# Simulation Strategies and Techniques

## Basement Car Park Ventilation



# Can CFD simulation be useful in ventilation design for basement car park?

## Need:

- Evaluate if ducted ventilation system for normal mode based on ACH will work

## Goal of CFD simulation:

- Analyze LMA values
- Recommend better, cost effective design

## Design performance criteria:

- LMA in normal mode 950 seconds or less (~15 minutes)

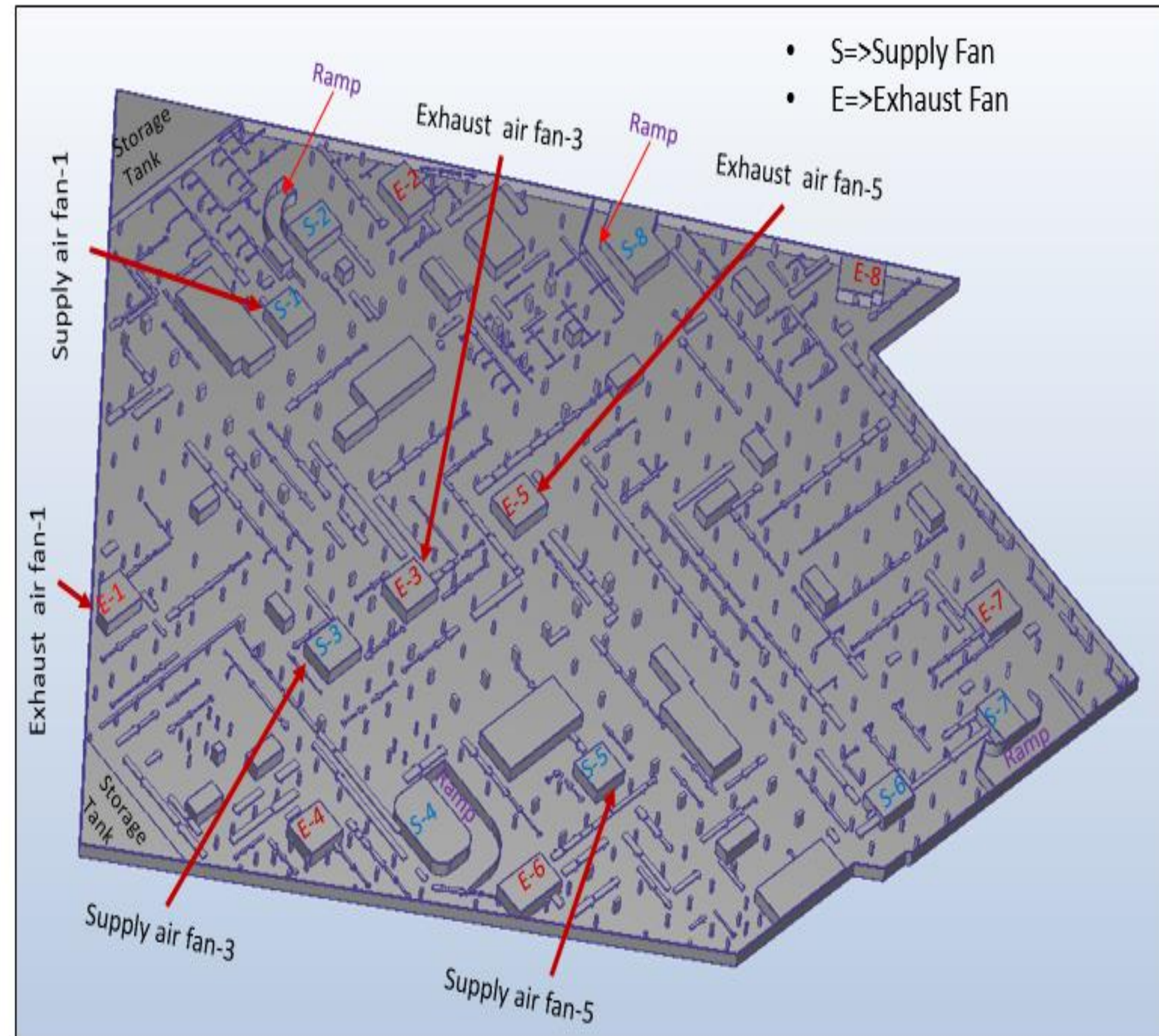


# CAD Model

## Basement Car Park

### Basement car park details

- Basement area = 46500 sq. m
- Height = 3.15 m
- No. entry/exit ramps = 4
- Normal mode ventilation = 6 ACH
- Total no. of fans = 8





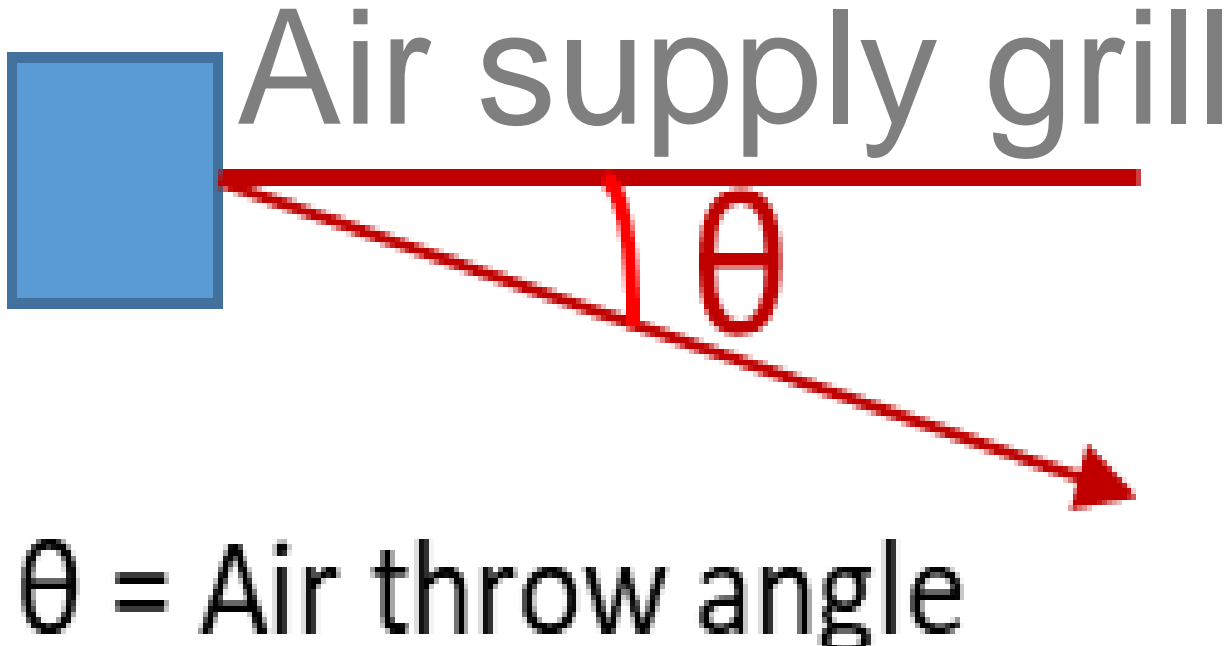
# Key Results for Design Performance

LMA values at the occupied level (1.7m from the floor)





# Basement Ventilation Performance Evaluation










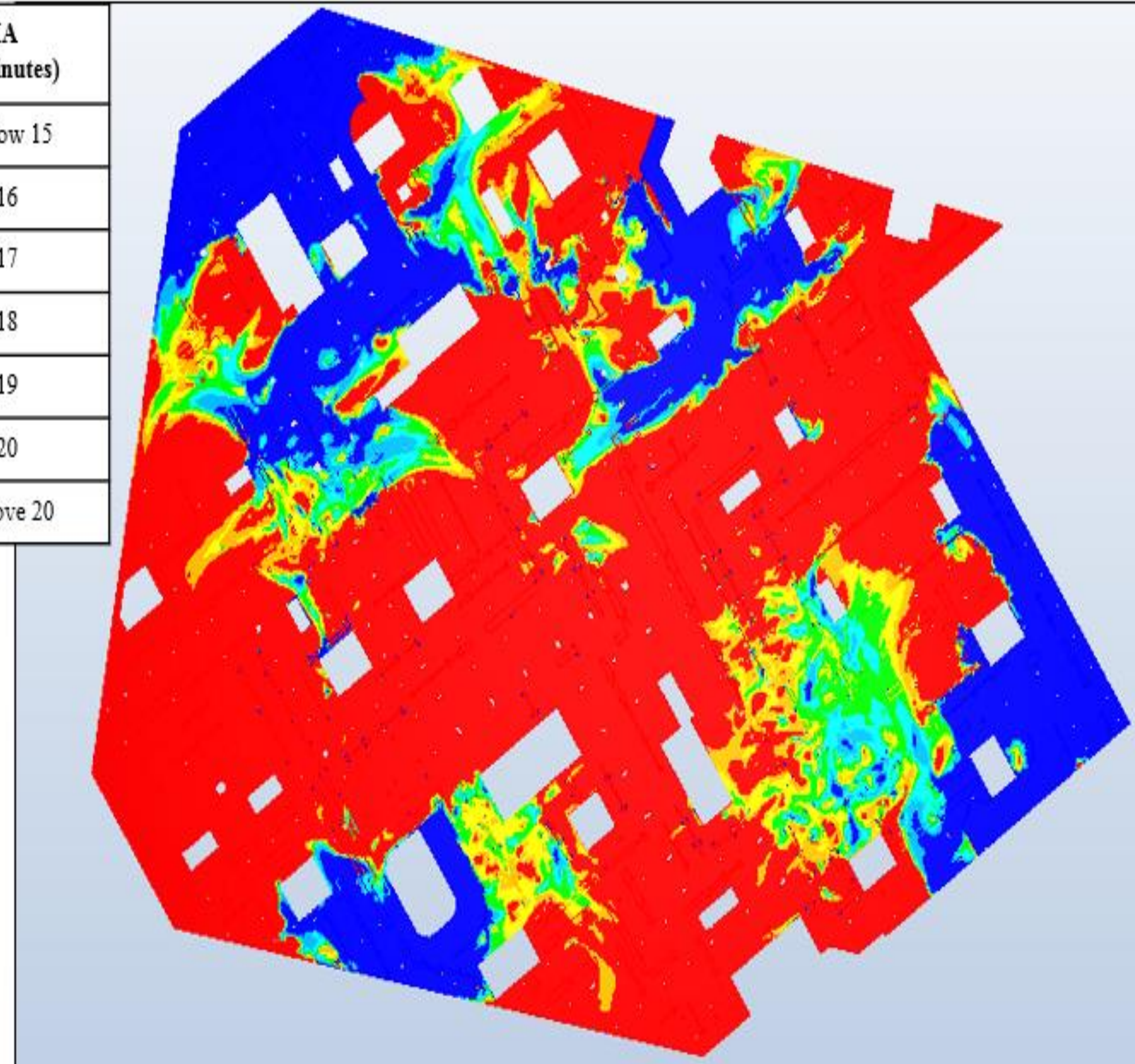
	Supply air flow direction ( $\theta$ )	Exhaust air flow direction ( $\theta$ )	Flow rate
Initial Design	0 degrees (horizontal)	0 degrees (horizontal)	Based on 6 ACH from 8 fans
Design Option 1	90 degrees (vertically downwards)	0 degrees (horizontal)	30% extra CFM from all fans (equivalent to 7.18 ACH)
Design Option 2	90 degrees (vertically downwards)	0 degrees (horizontal)	5 fans with 30% extra CFM, 3 fans with CFM doubled
Design Option 3	90 degrees (vertically downwards)	90 degrees (vertically downwards)	5 fans with 30% extra CFM, 3 fans with CFM doubled
Design Option 4	45 degrees	0 degrees (horizontal)	Flow rates doubled for all fans.



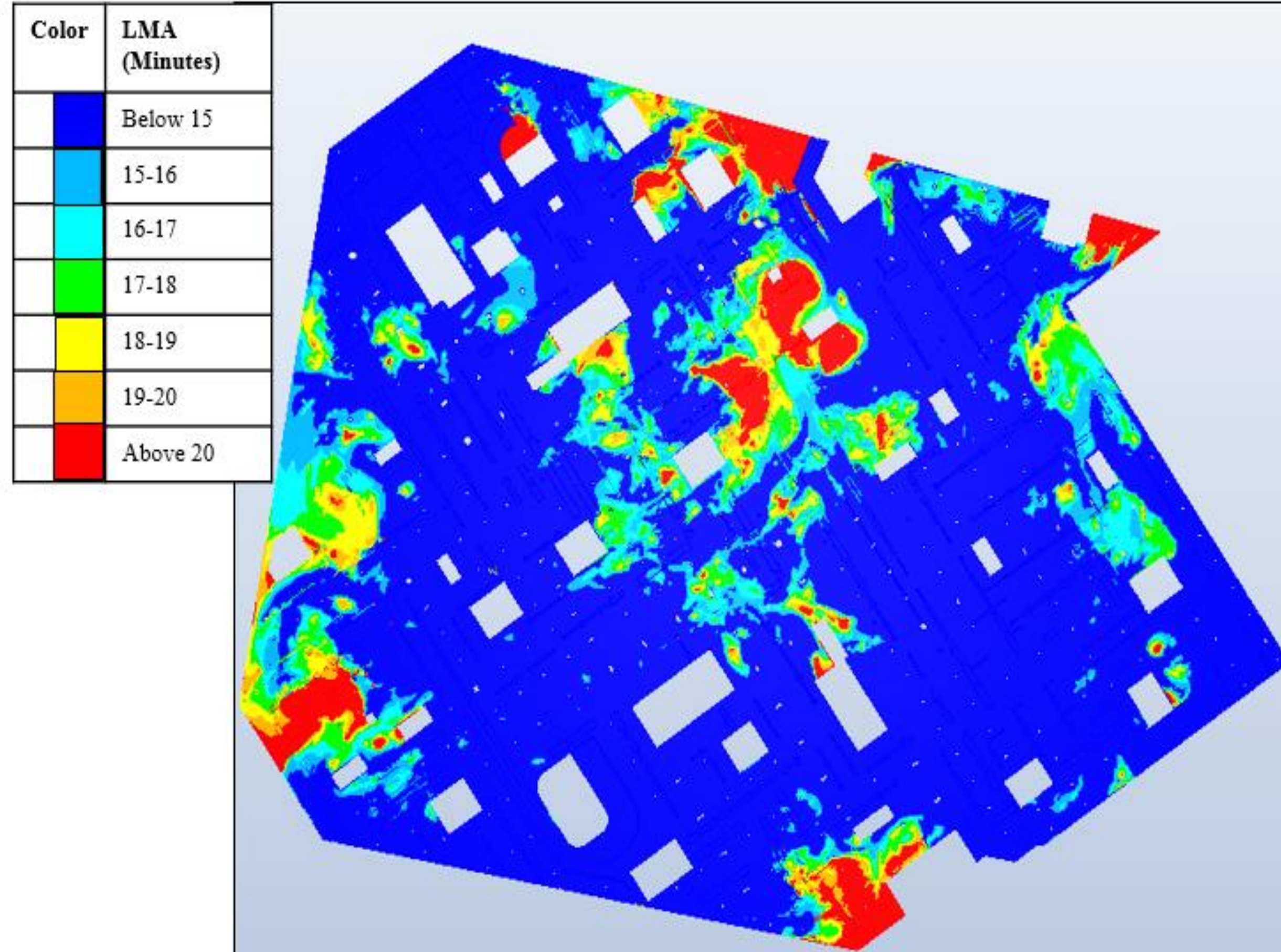
# Initial Design

- LMA values  $>15$  minutes for about 73% of area
- Design is not satisfactory

Color	LMA (Minutes)
	Below 15
	15-16
	16-17
	17-18
	18-19
	19-20
	Above 20



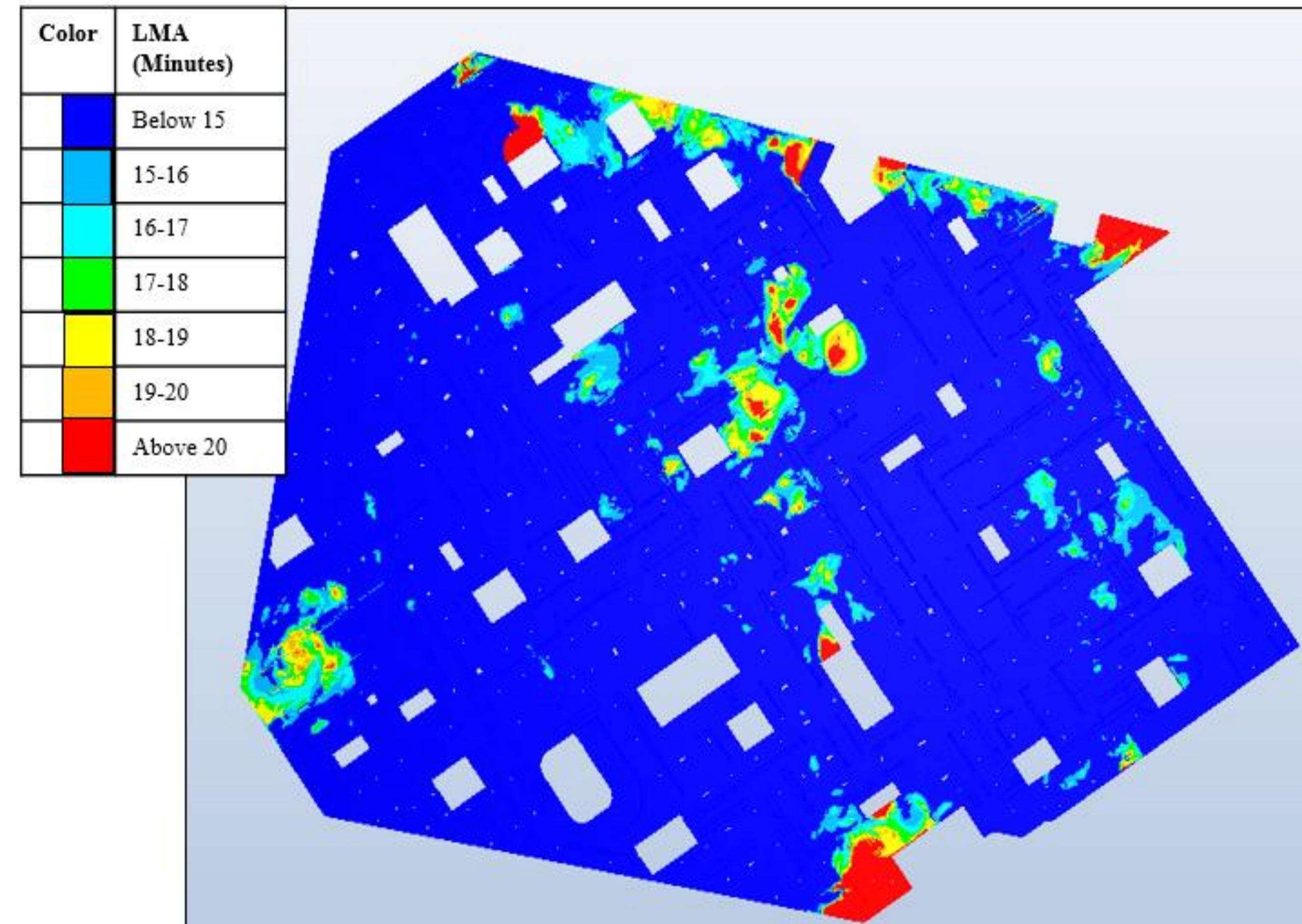




## Design Option 1:

All fans capacity increased by 30%  
S-H, E-H

LMA values <15 minutes for  
~73% of area

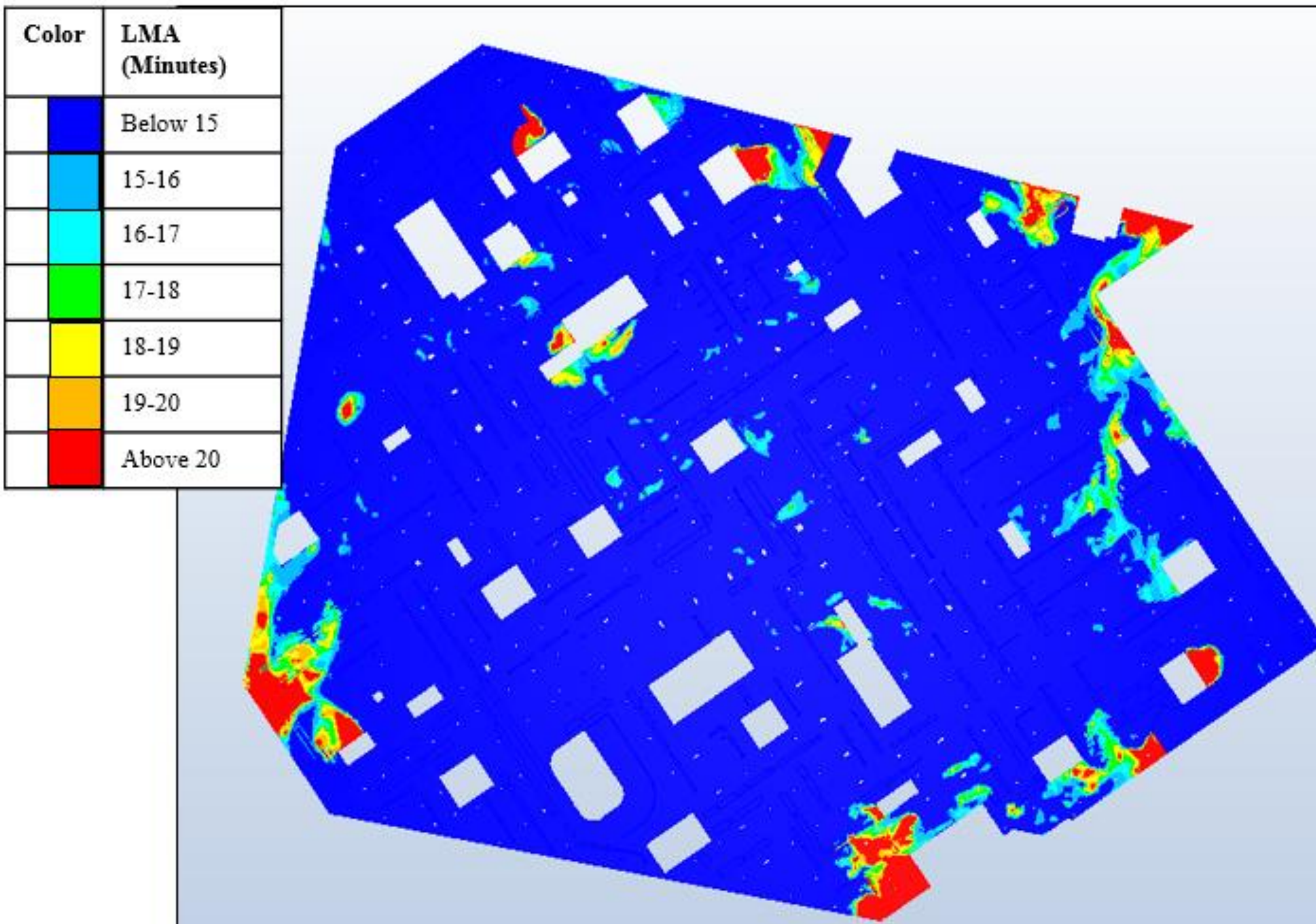


## Design Option 2:

5 fans with 30% extra CFM, 3 fans  
with CFM doubled, S-V, E-H

LMA values <15 minutes for ~  
89% of area

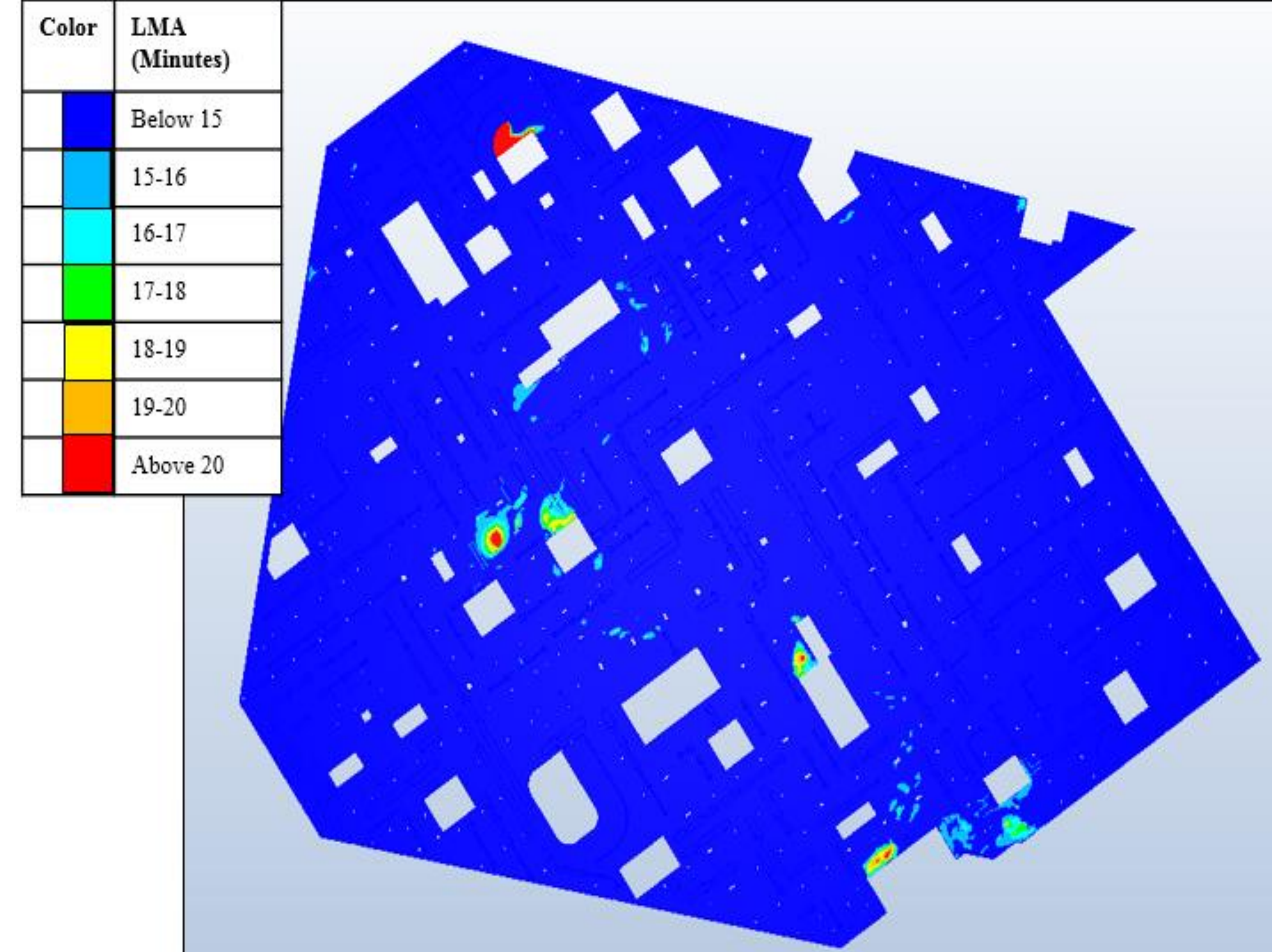




## Design Option 3:

5 fans with 30% extra CFM, 3 fans with CFM doubled, S-V, E-V

LMA values <15 minutes for ~90% of area



## Design Option 4 :

Flow rates doubled for all fans.

S-45, E-H

LMA value < 15 minutes for 98 % of area



# Outcome- Basement Car Park Ventilation

- Design Option-4 is the best design but ventilation with LMA values <15 minutes is achieved by increase in fan capacity i.e. fan capacity at 12 ACH.
- Design Option-3 provides a better, cost effective design for acceptable ventilation with 78% fan capacity, thereby saving cost of fans by 22%.



# Simulation Strategies and Techniques

## Free surface flow simulation- rain water leakage



# Why CFD simulation of rain water flow on roof?

## Issue:

- Due to heavy rain, water leaked from roof to passenger area in airport
- Water leakage seemed to be from skylight areas on the roof

## Goal of CFD simulation:

- Predict water levels at Skylight obstructions
- Recommend corrective action

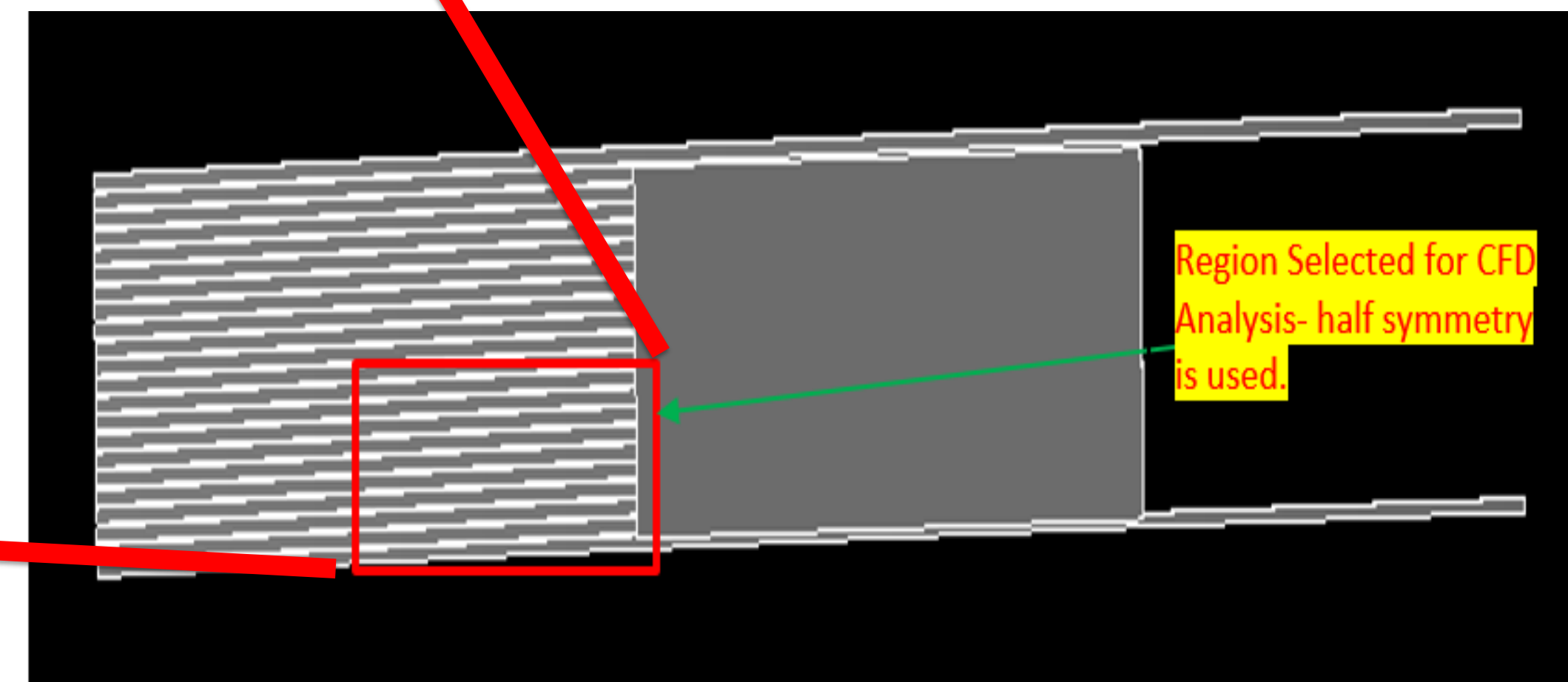
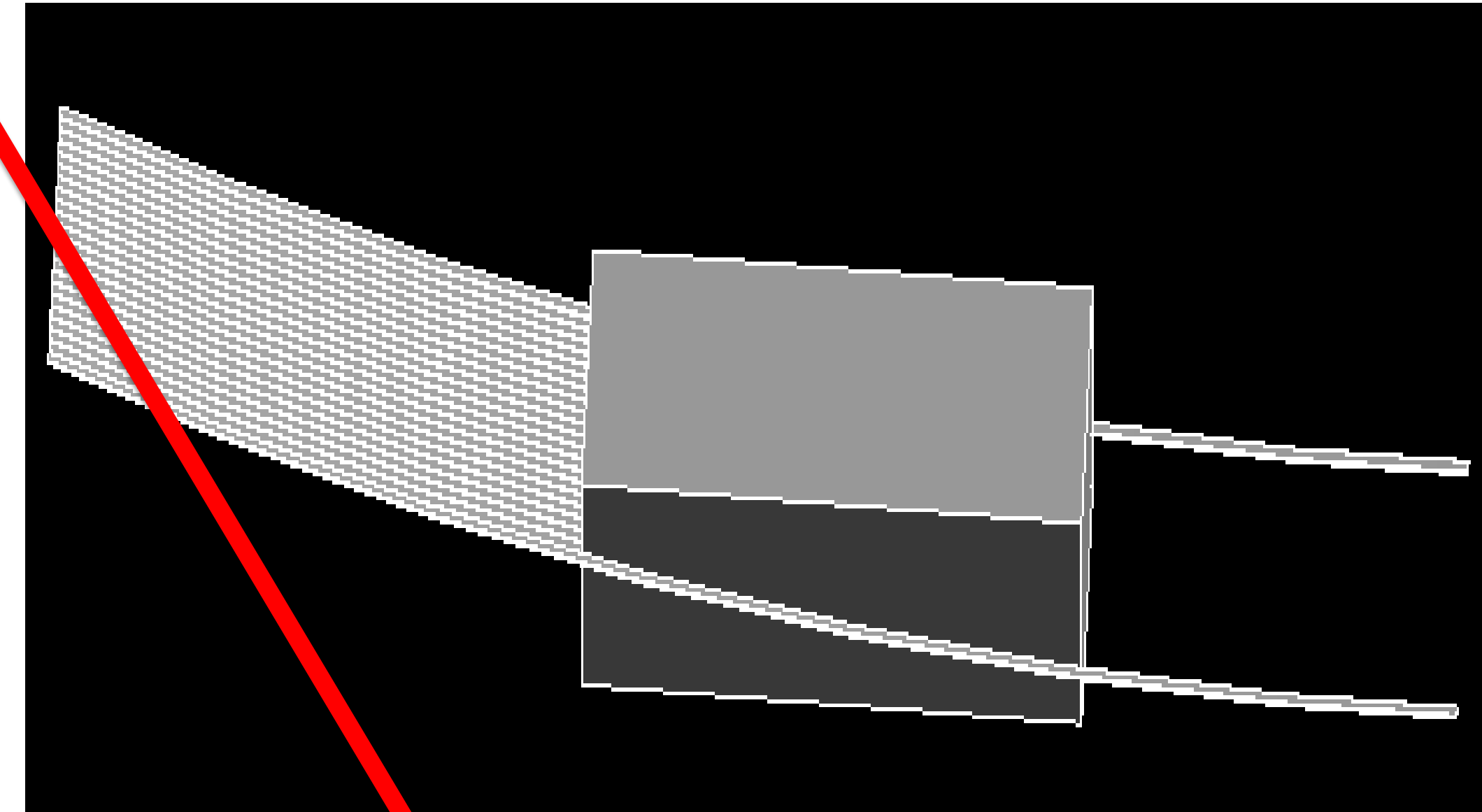
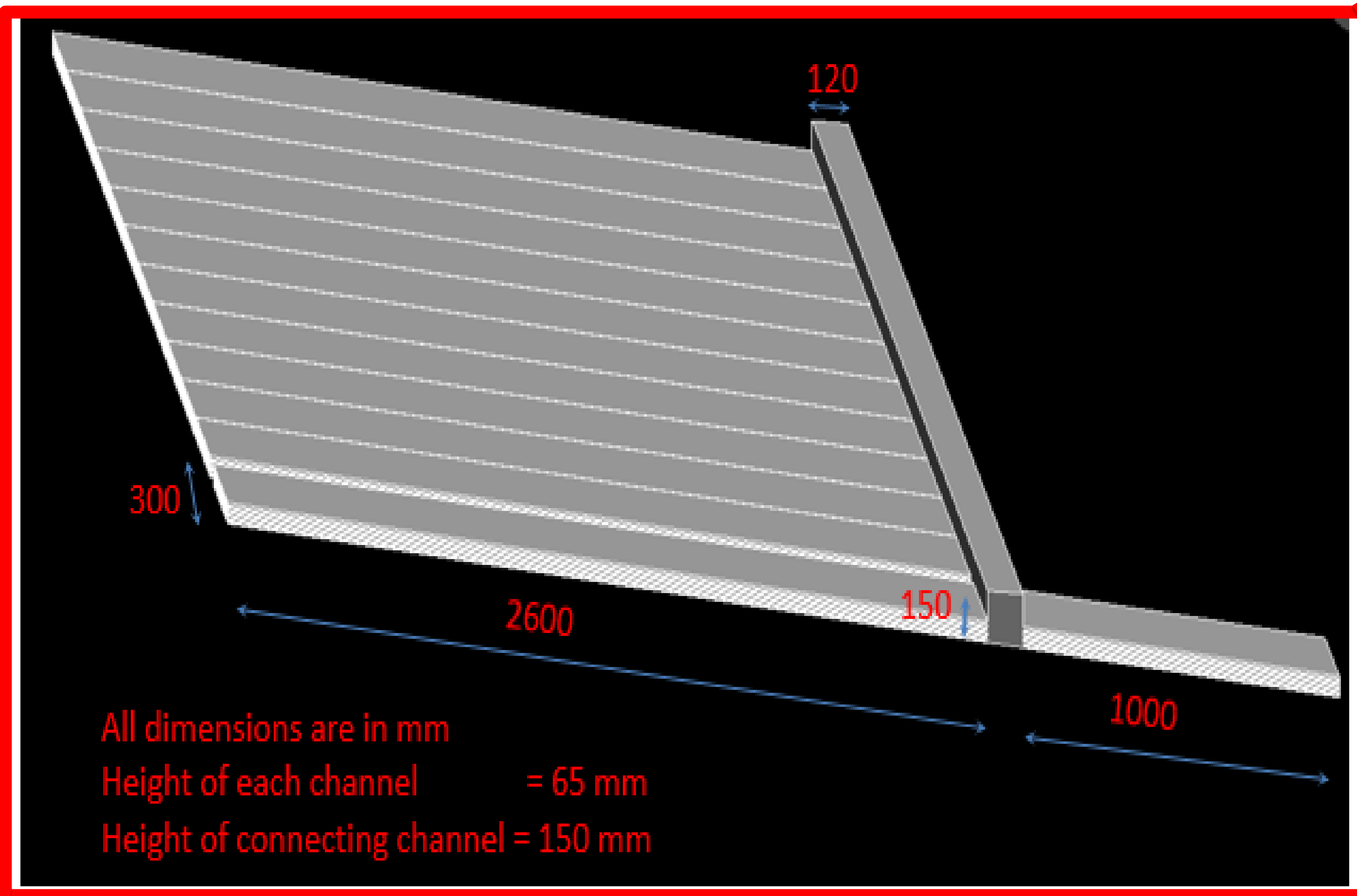
## Design performance criteria:

- Water level in roof channels below 65mm



# CAD Model

## Airport roof with skylight portion



# CFD modeling

## Free surface flow

### Material Properties:

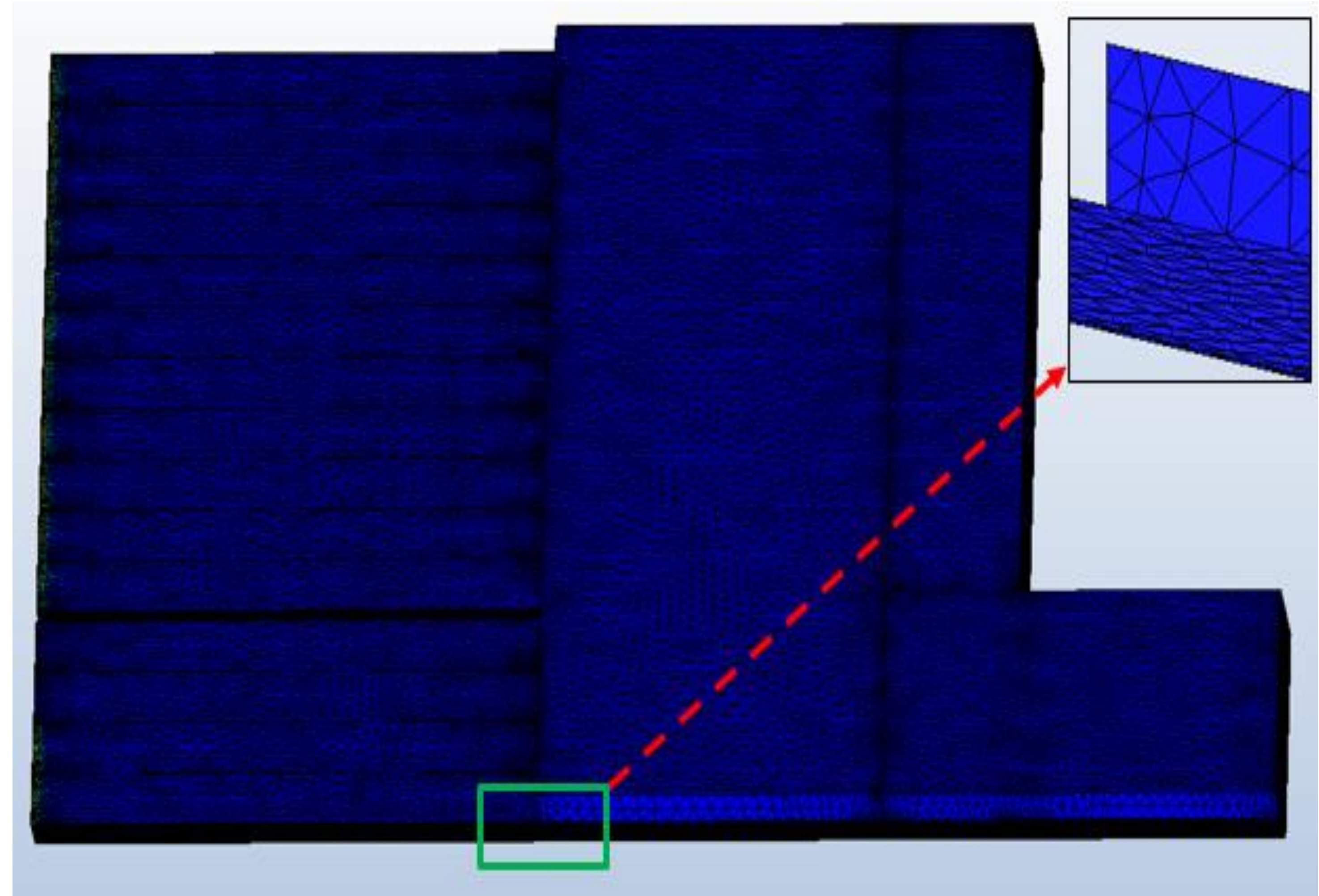
- Water : Fixed Quantity

### Boundary conditions:

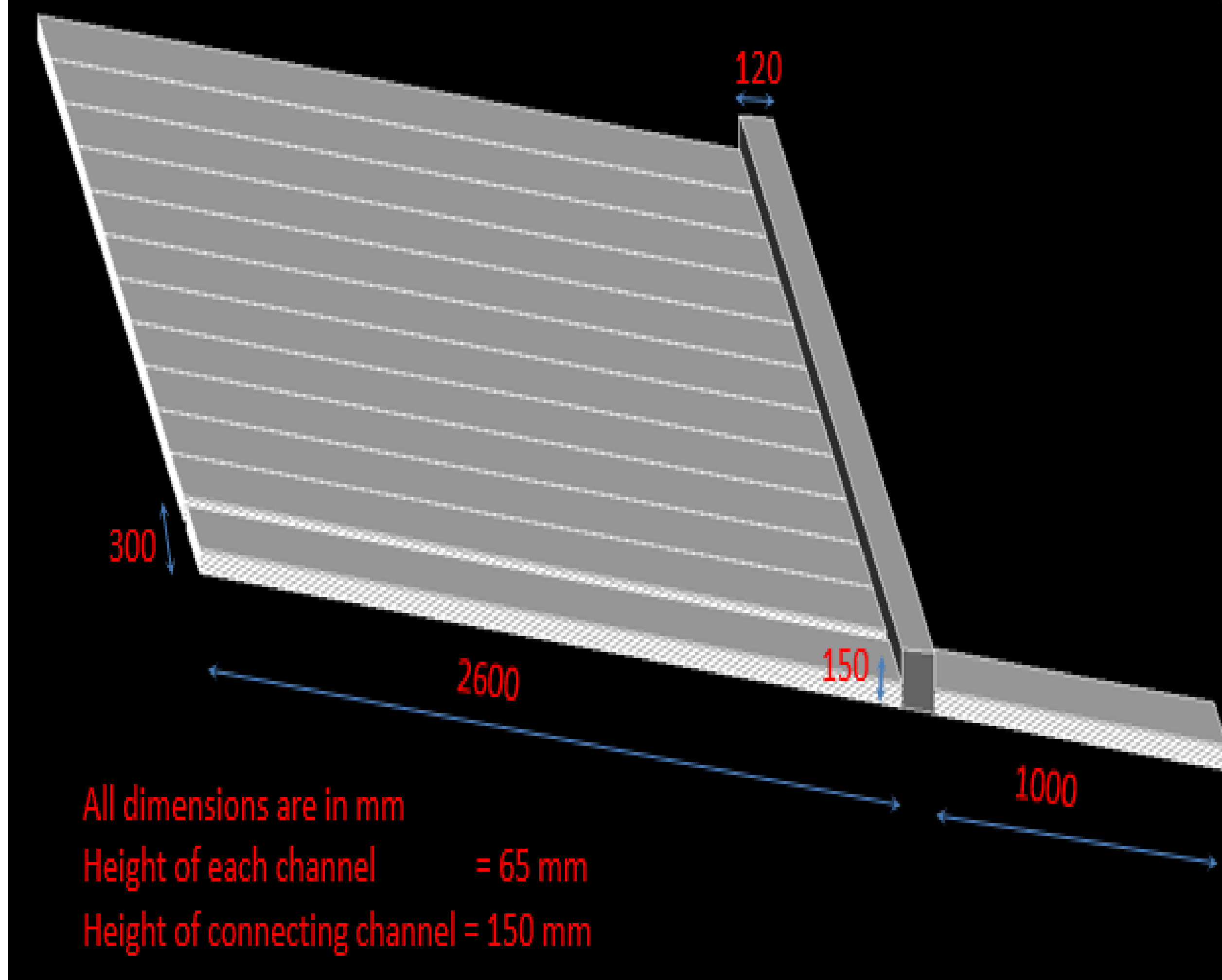
- Inlet : Velocity-0.041 m/s  
at each channel inlet.

### Solution Parameters:

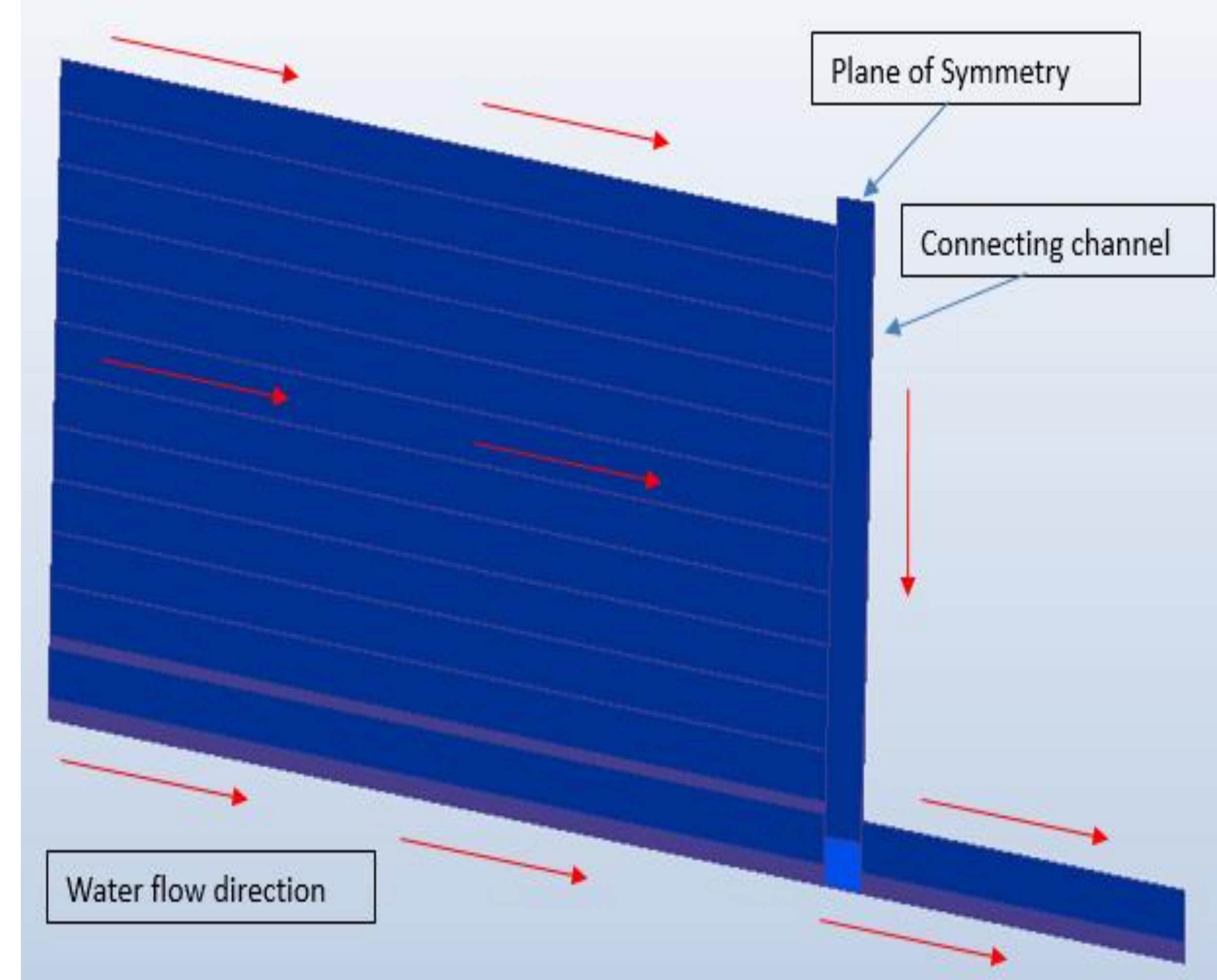
- Default values in CFD S/W



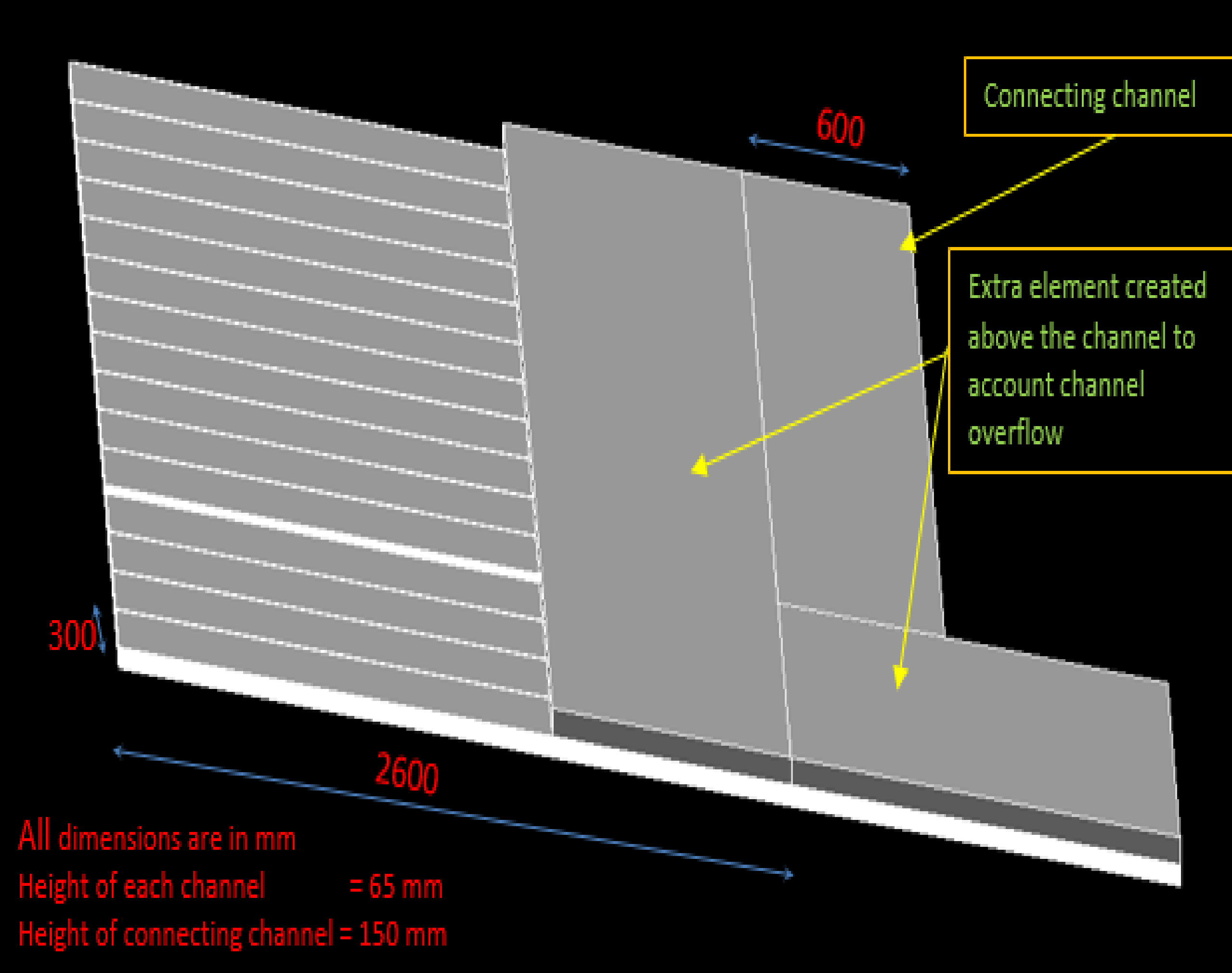




**CAD model- Initial design**  
 Connecting channel width = 120 mm

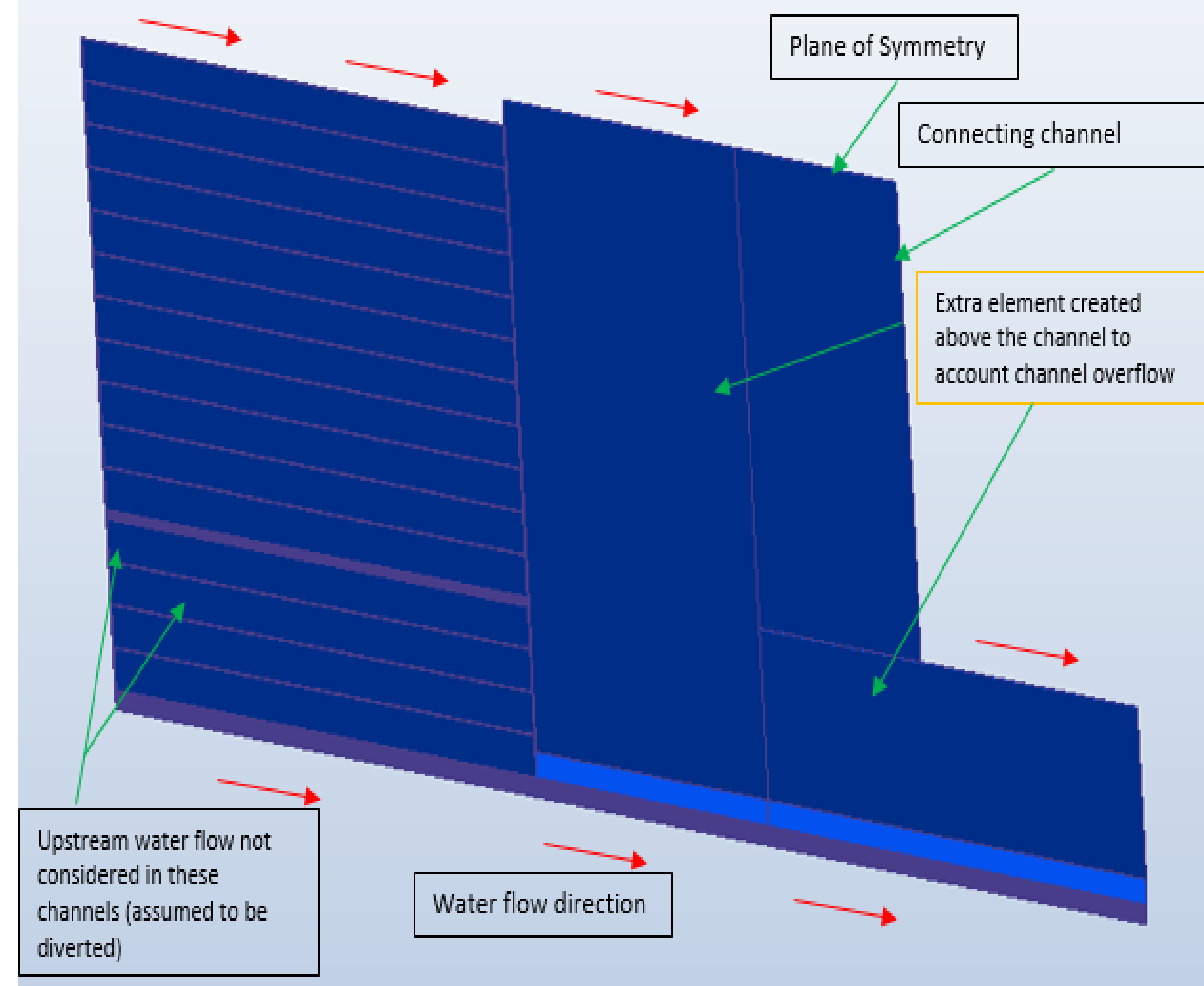


**CFD model- Initial design**



## CAD model- Final design

Connecting channel width = 600 mm



## CFD model-Final design

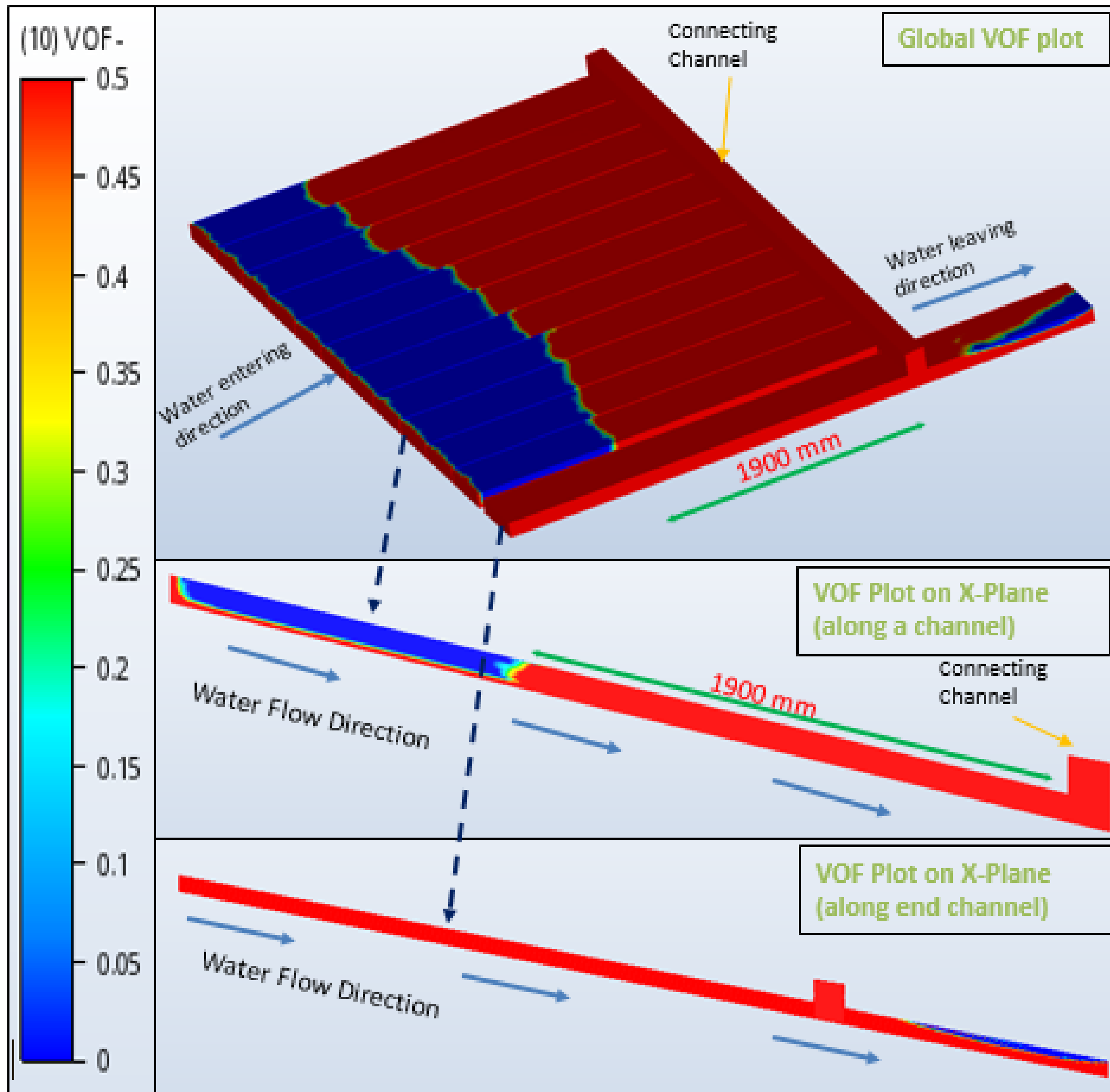


# Key Results for Design Performance

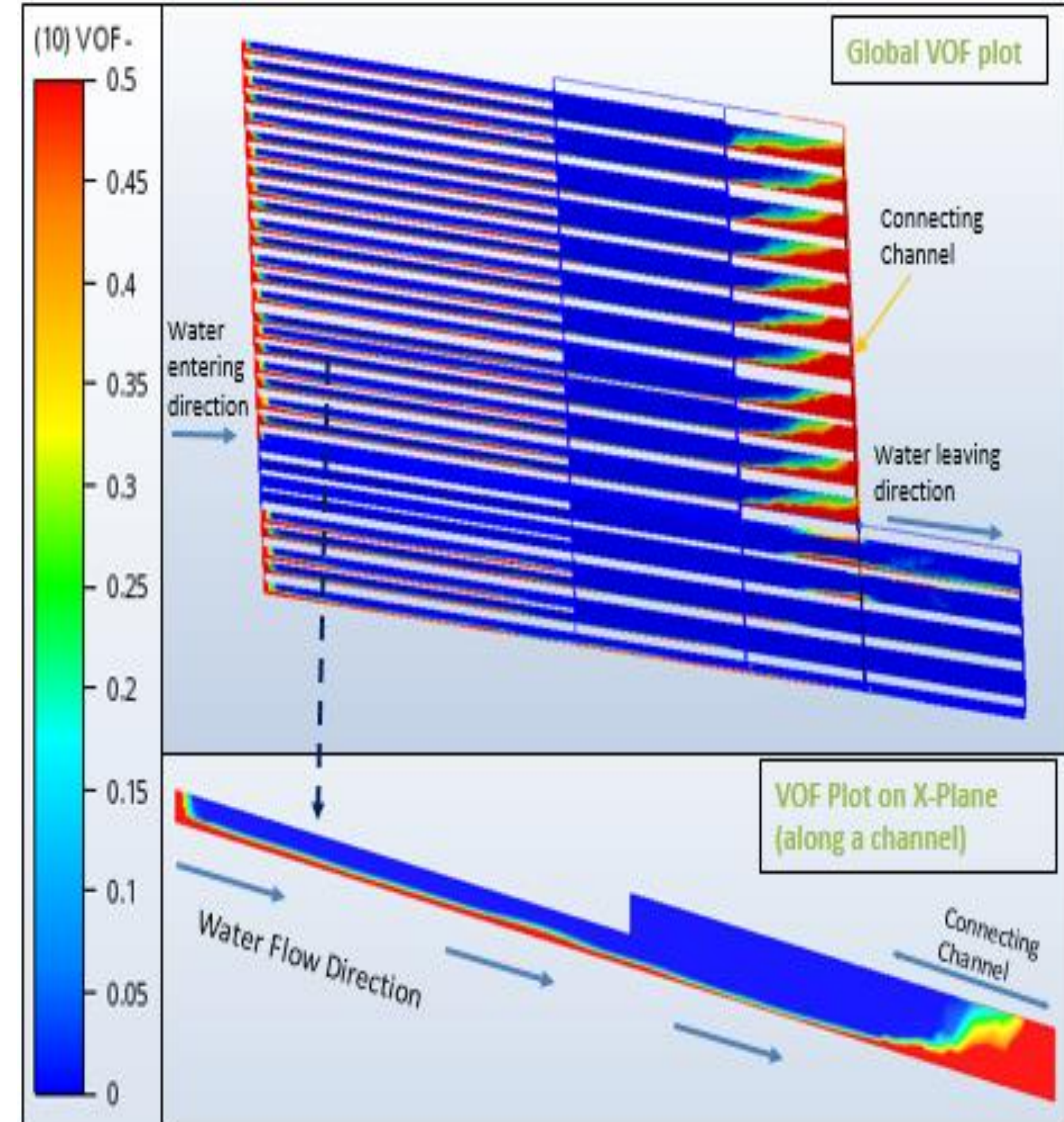
## VOF (Volume of Fluid) plots







CFD results- Initial design



CFD results-Final design



# Outcome- Rain water flow simulation- roof

For Final design:

- Water level in all channels except connecting channel is below 65 mm (No overflow).
- The level of water in the downstream side of the connecting channel (critical area) where it meets the long channel is 45 mm (Well below the height of channel).
- Hence water can not leak!

# Advantages and Limitations





# Advantages

- Easy to use
- Better designs
- Basis for innovative design
- Cost effective designs
- Avoidance of performance /warranty issues

# Limitations

- As good as assumptions
- Computationally intensive (better accuracy requires more time)
- Depends on limited test data for validation
- Relevant test data not available for real life scenarios
- Requires knowledge and experience



# How did I do?

- Your feedback is valuable
- AU speakers get feedback in real-time
- Fill out a class survey using AU mobile app.
- Your feedback results in better classes and a better AU experience.







# Thank you for listening....

Keep in touch!

[mmrajulu@Intecc.com](mailto:mmrajulu@Intecc.com)

For info on who we are and what we do...

[www.Intecc.com](http://www.Intecc.com)

[www.larsentoubro.com](http://www.larsentoubro.com)





Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2018 Autodesk. All rights reserved.

