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

AUTODESK. UNIVERSITY

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
- Simulation Strategies and Techniques
- Key Results for MEP Design Performance
- Advantages and Limitations

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.

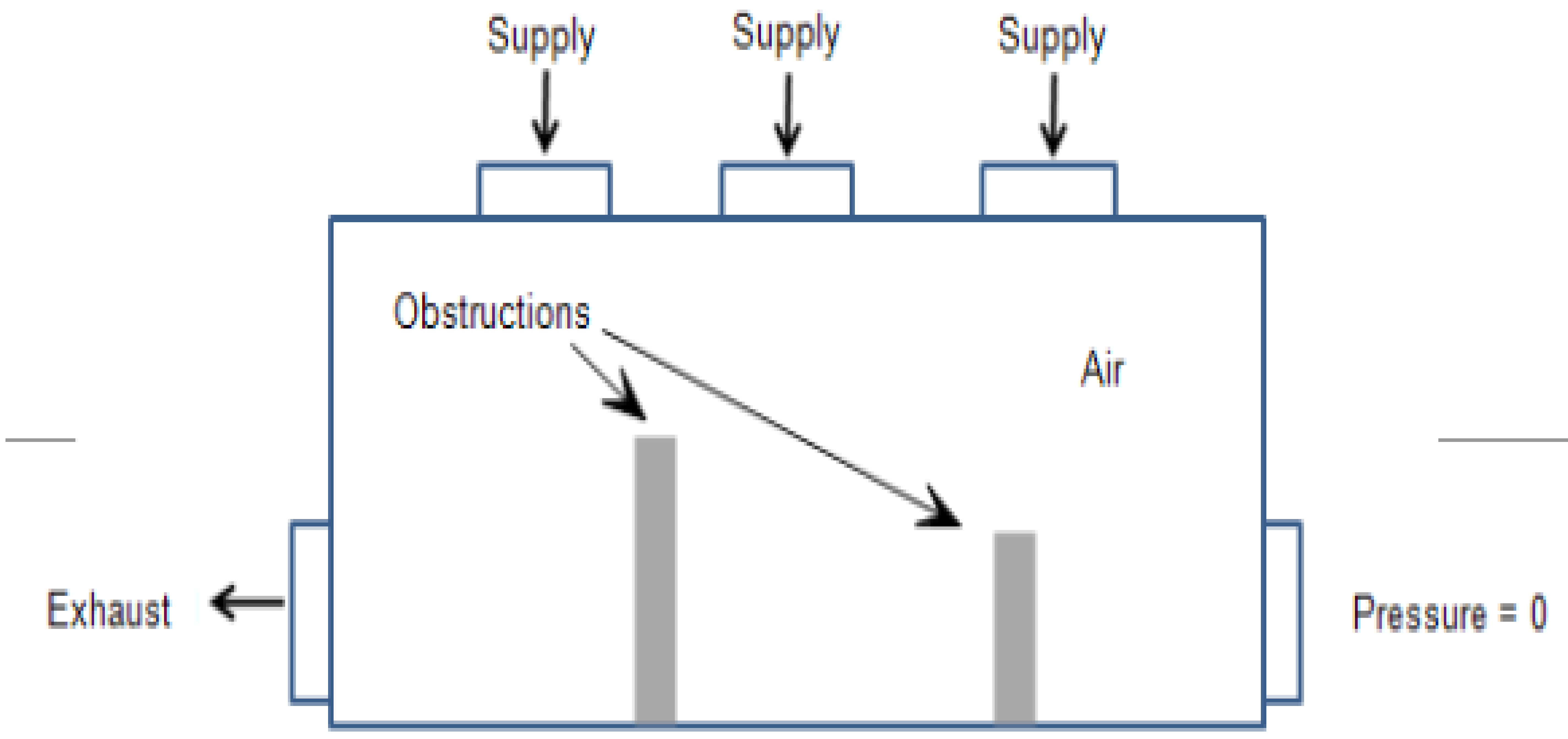
STAGE D

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

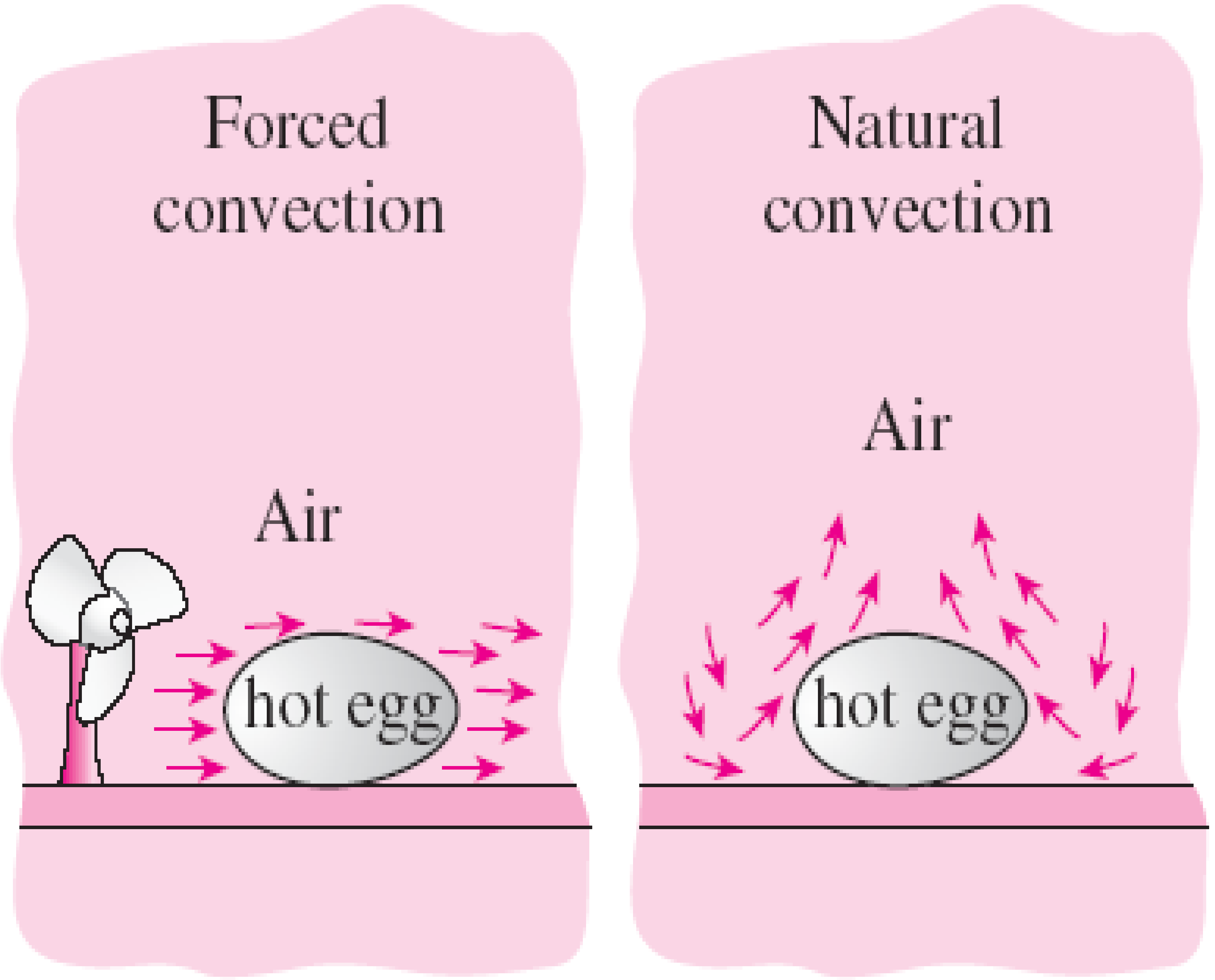
Heat Transfer modeling

Forced Convection

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

Natural Convection

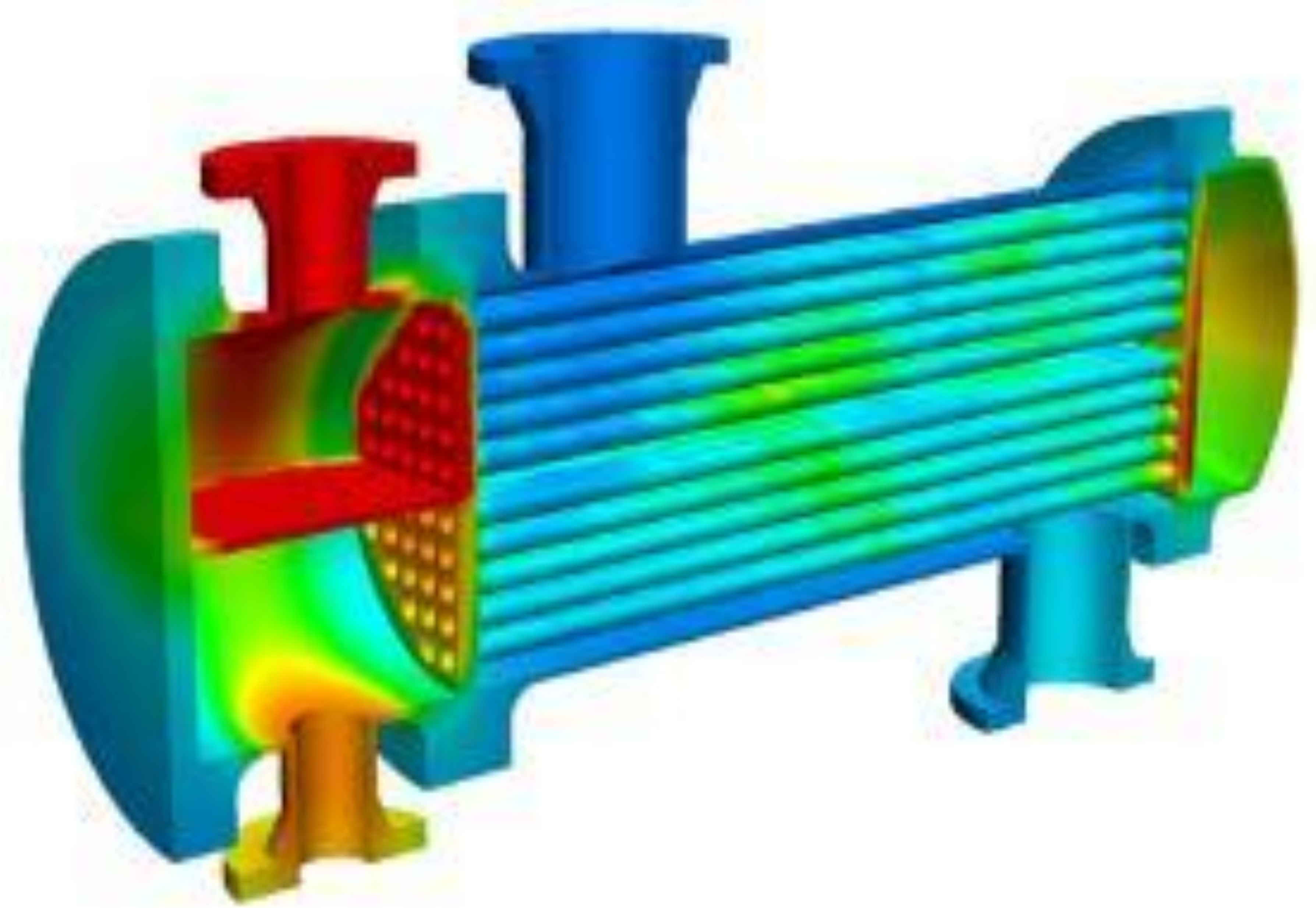
- Flow and heat transfer - coupled
- Buoyancy-driven flow



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

Heat Transfer modeling

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



Source: IMAGINiT Technologies blog

Mixed convection

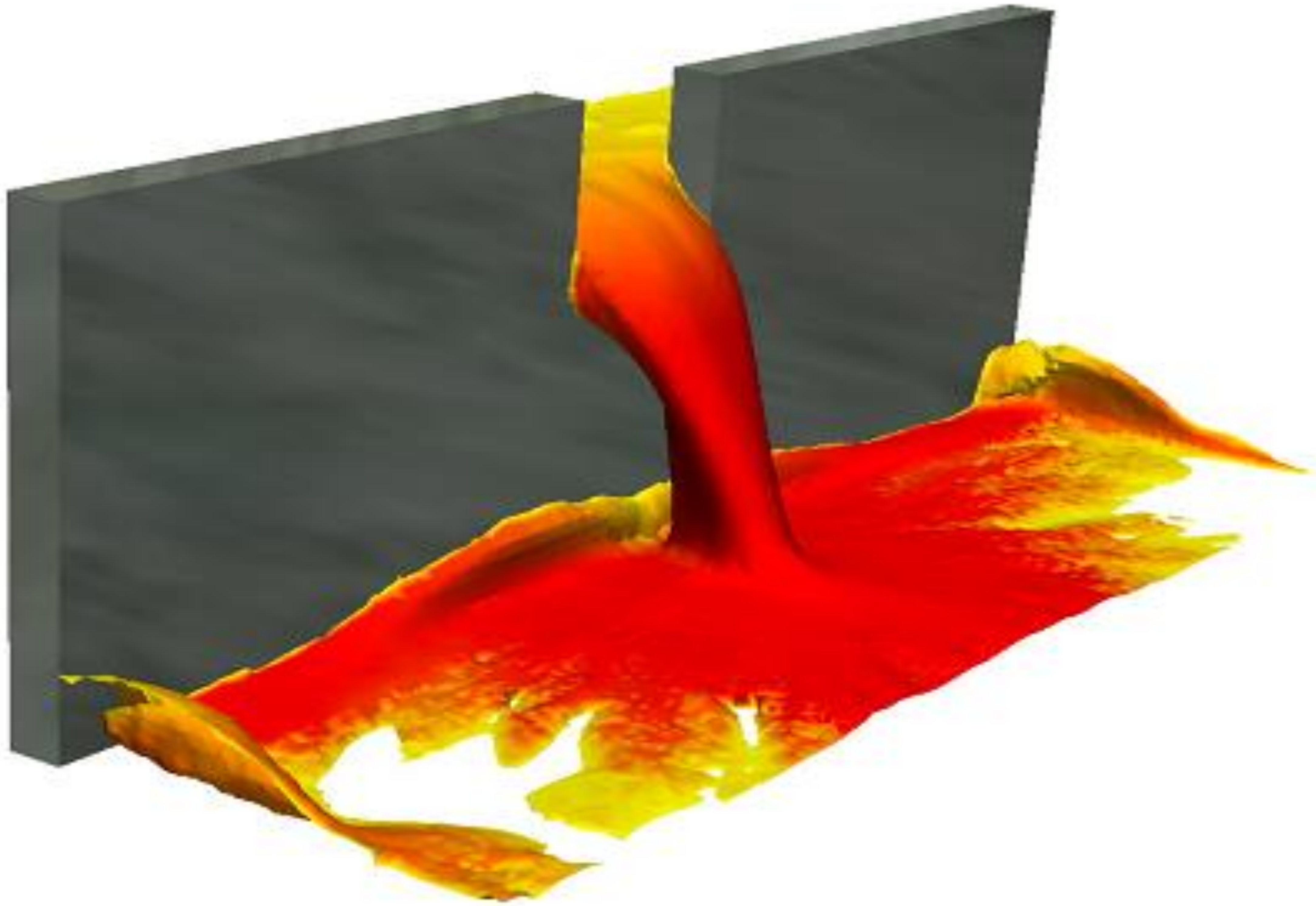
- Natural and forced convection.
- Appreciable buoyancy effects

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:

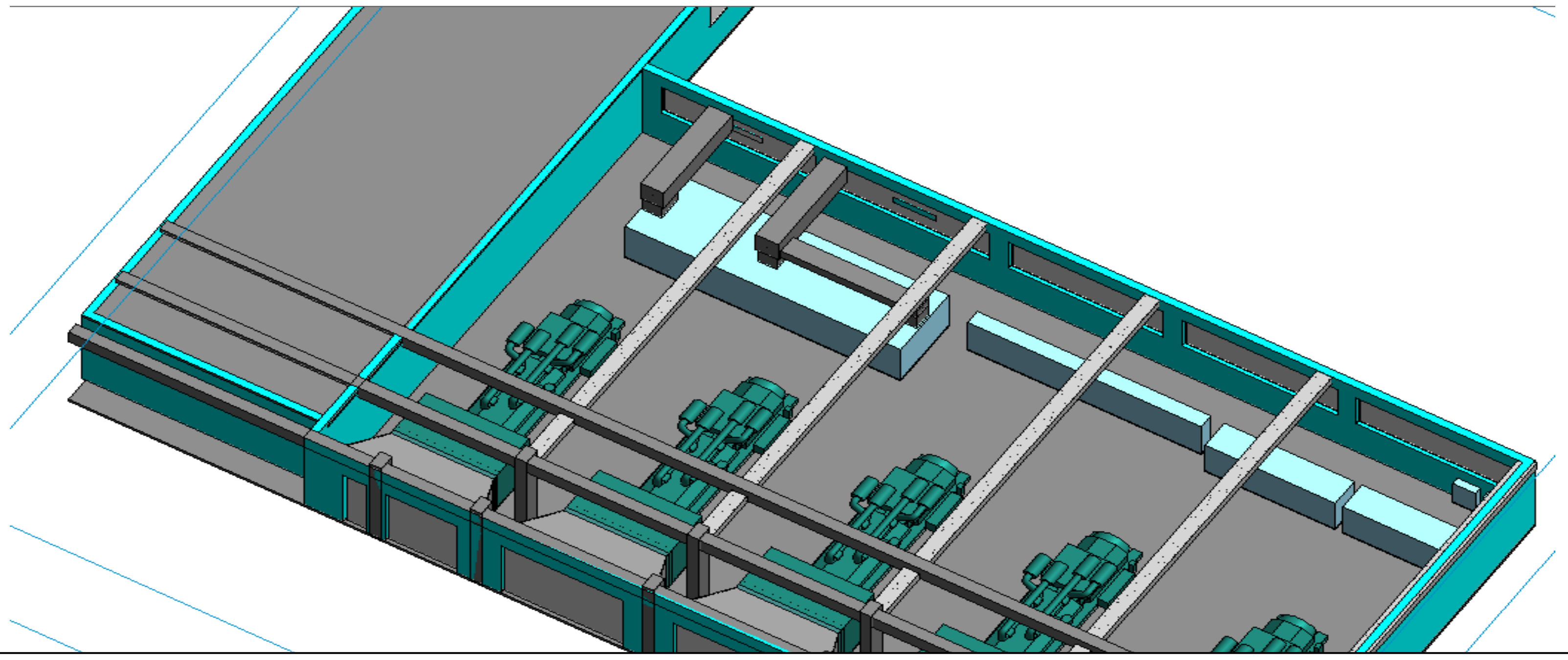
- air flow and temperature distribution
- recommend solution

Design performance criteria:

- air temperature $<50^0$ C around engine and 45^0 C around alternator
- bottom to top air flow

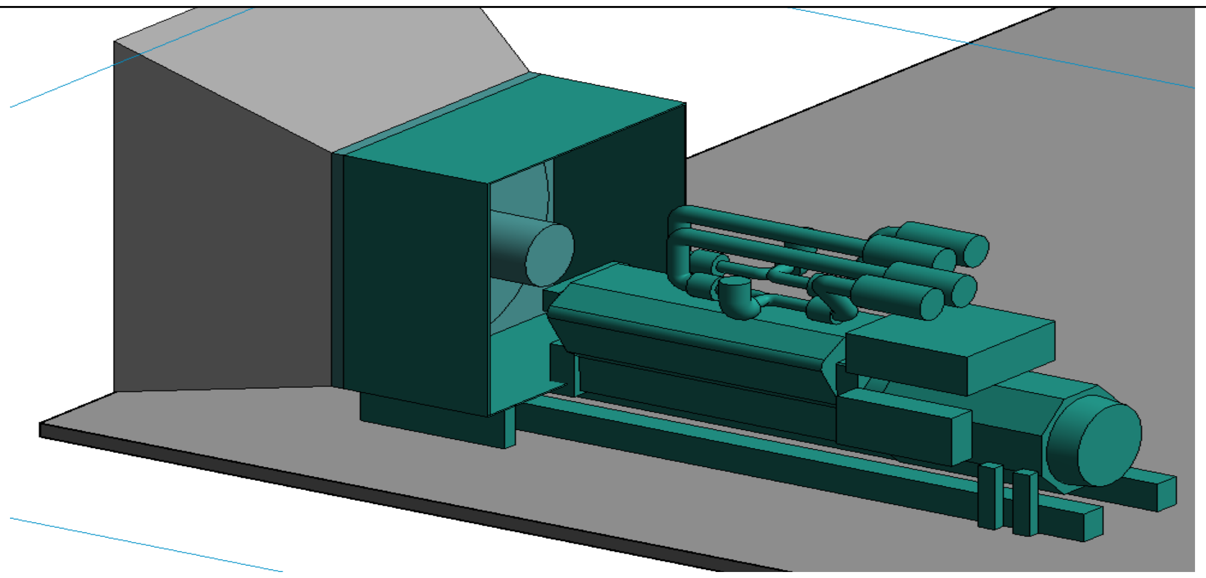
CAD Model

Create CAD model in Revit.

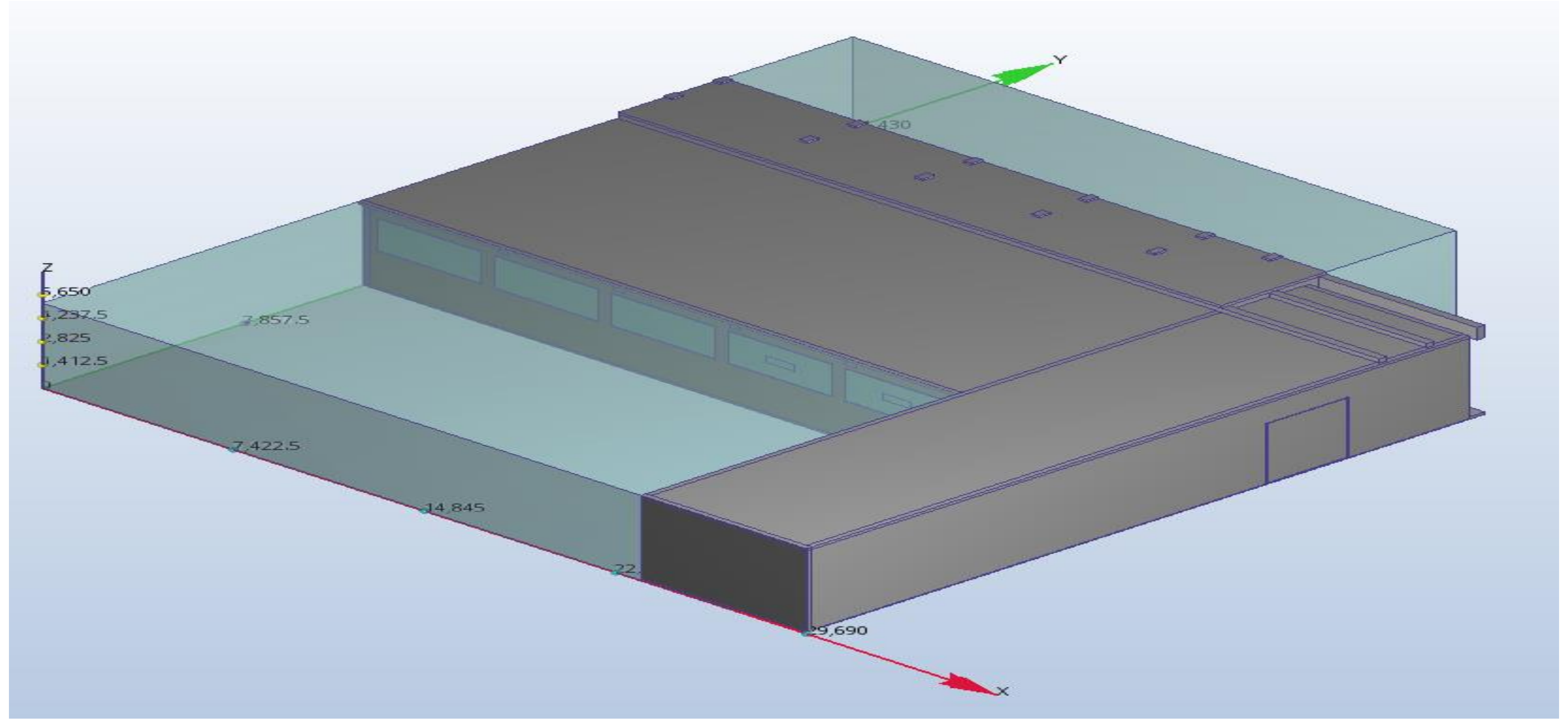


DG rating = 2000 KVA (5 nos.)

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



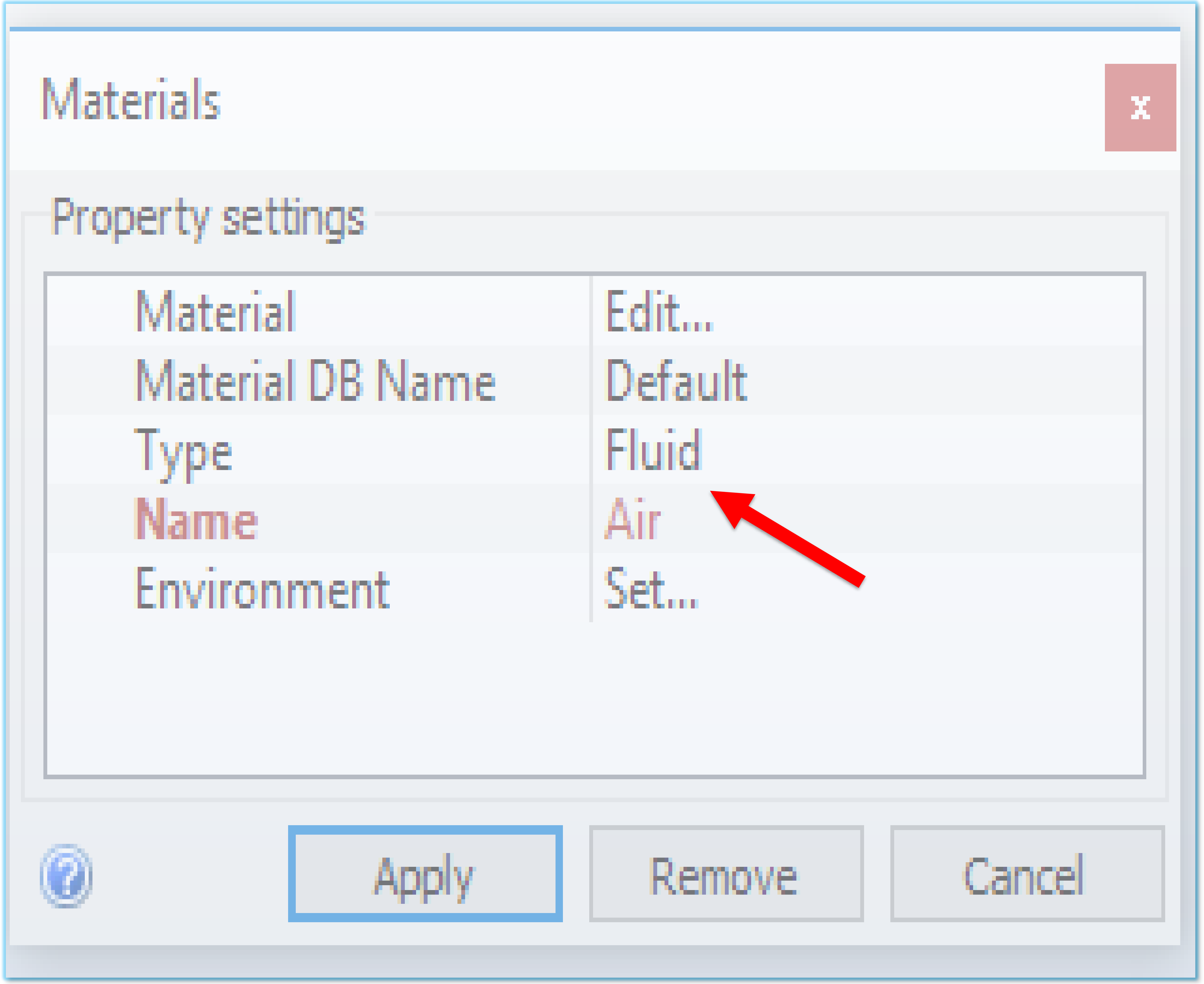
- Simplify geometrical details

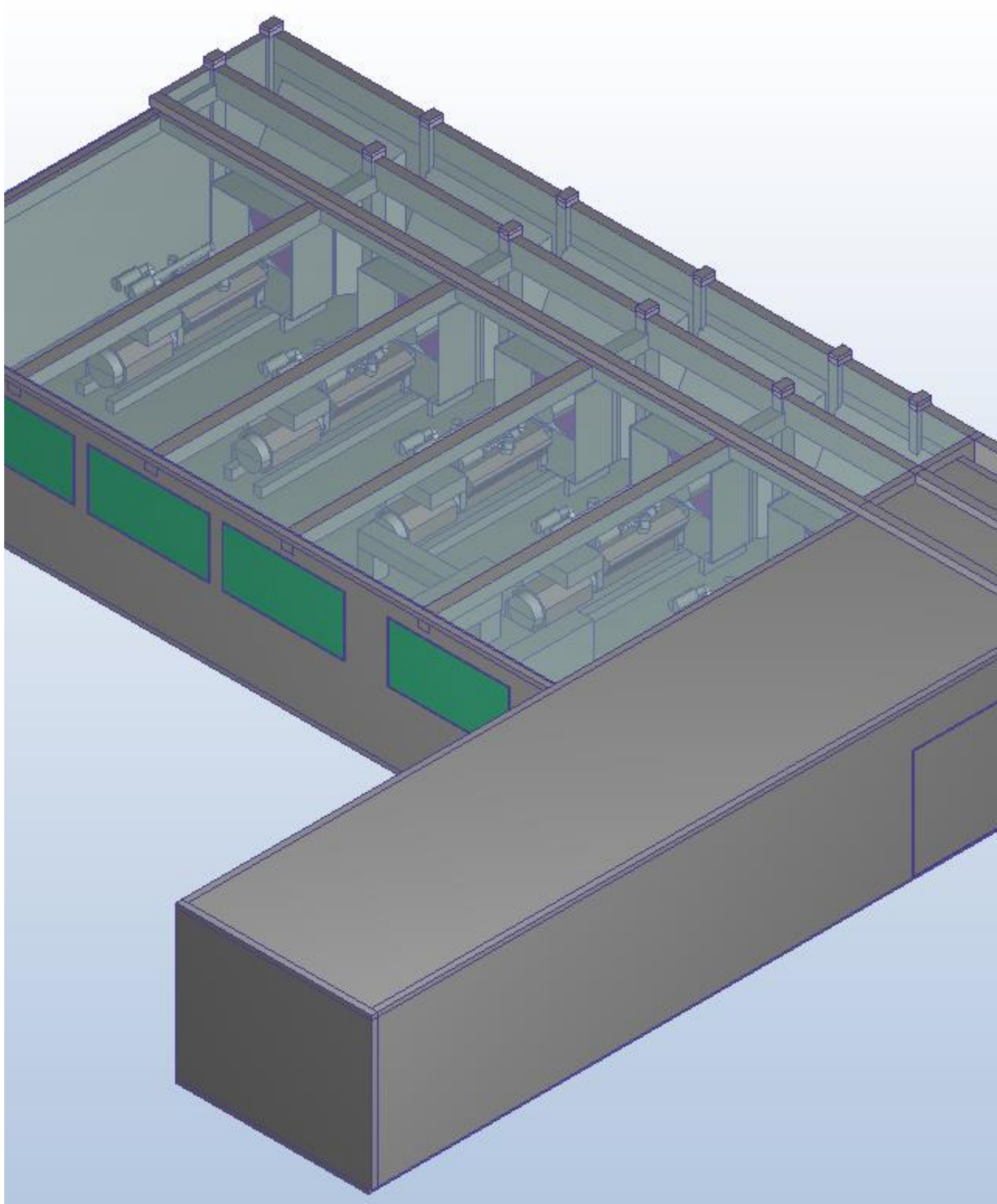


- Add inlet and outlet extensions

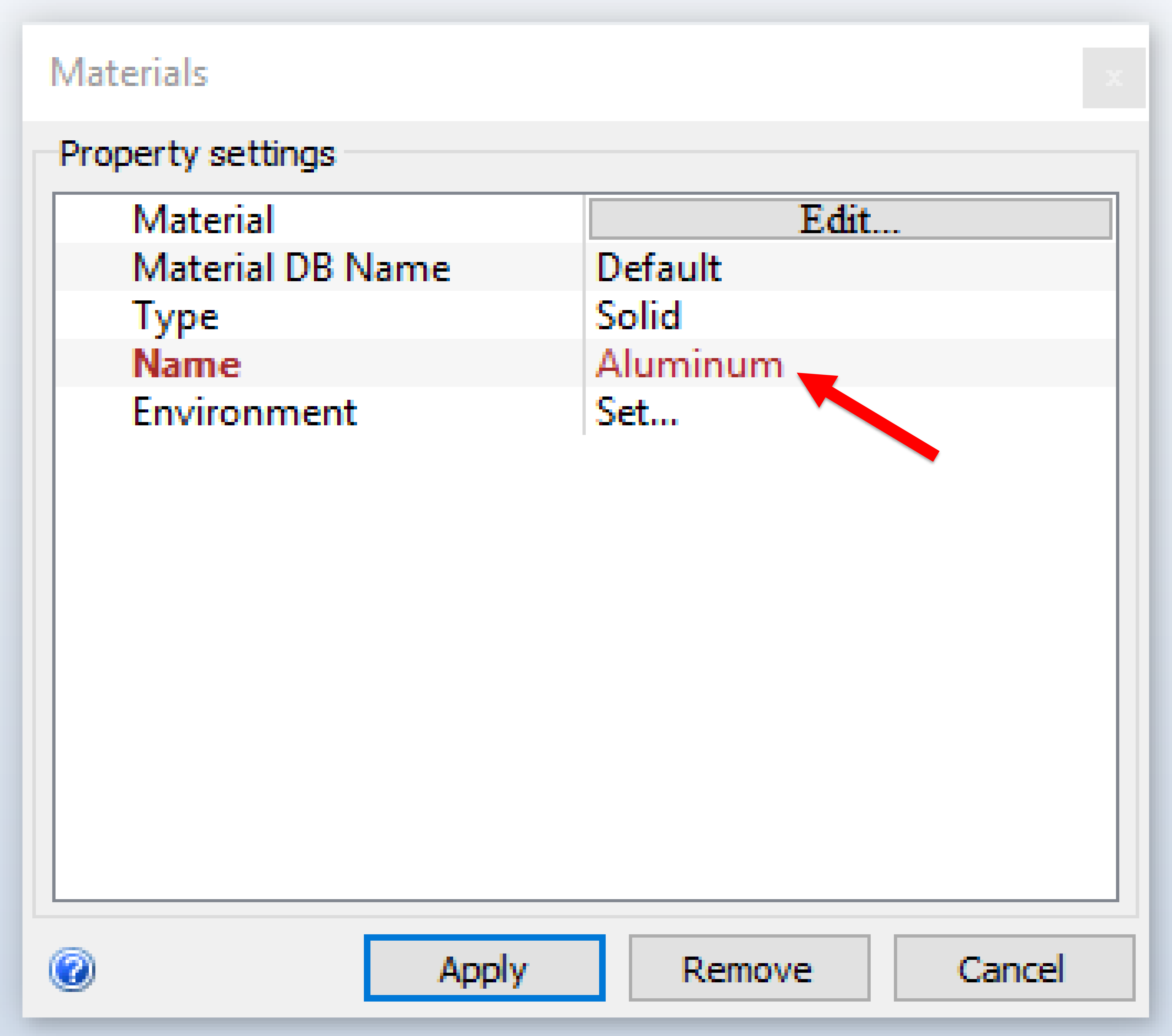
Materials

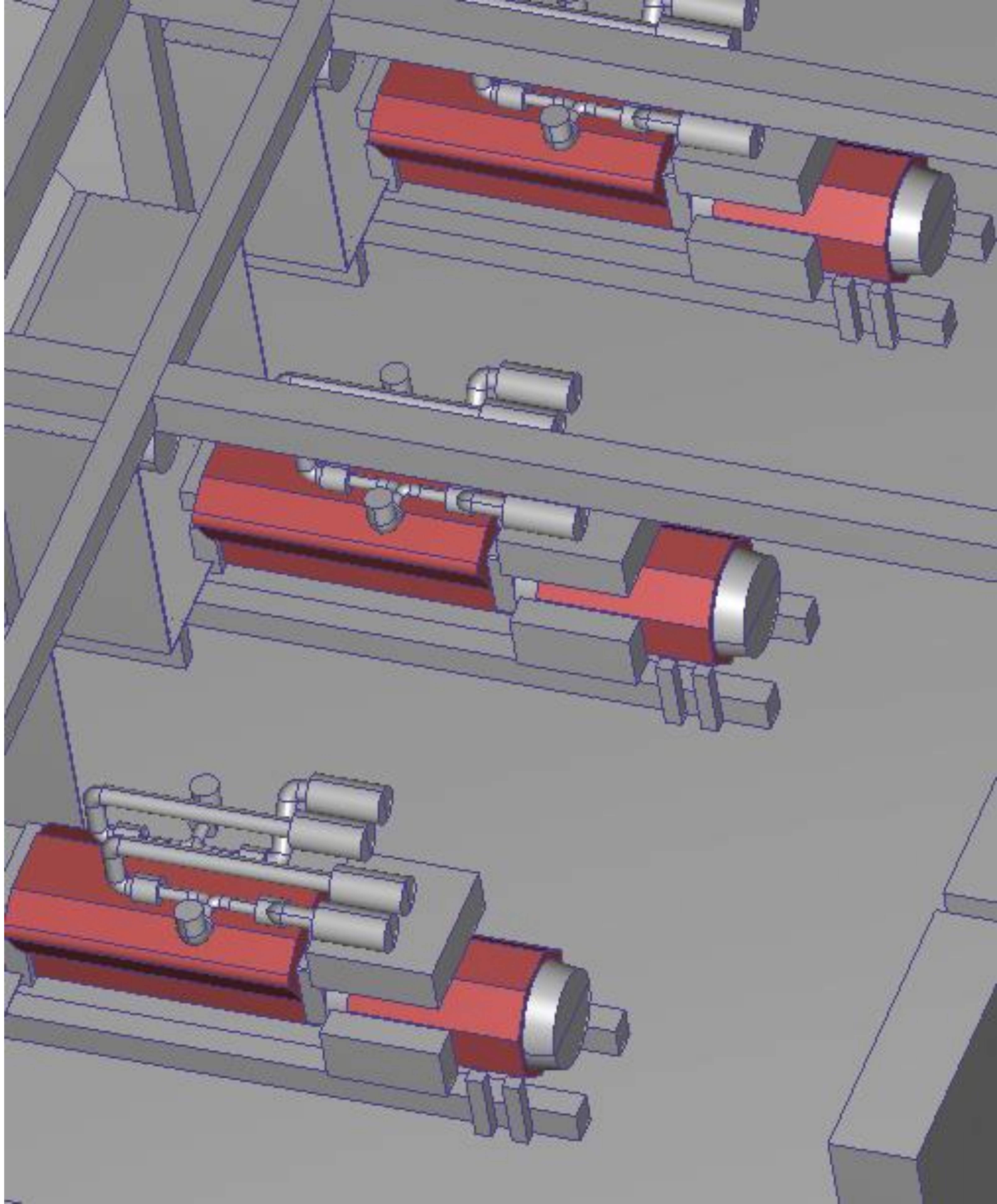
Air volume and assign air material



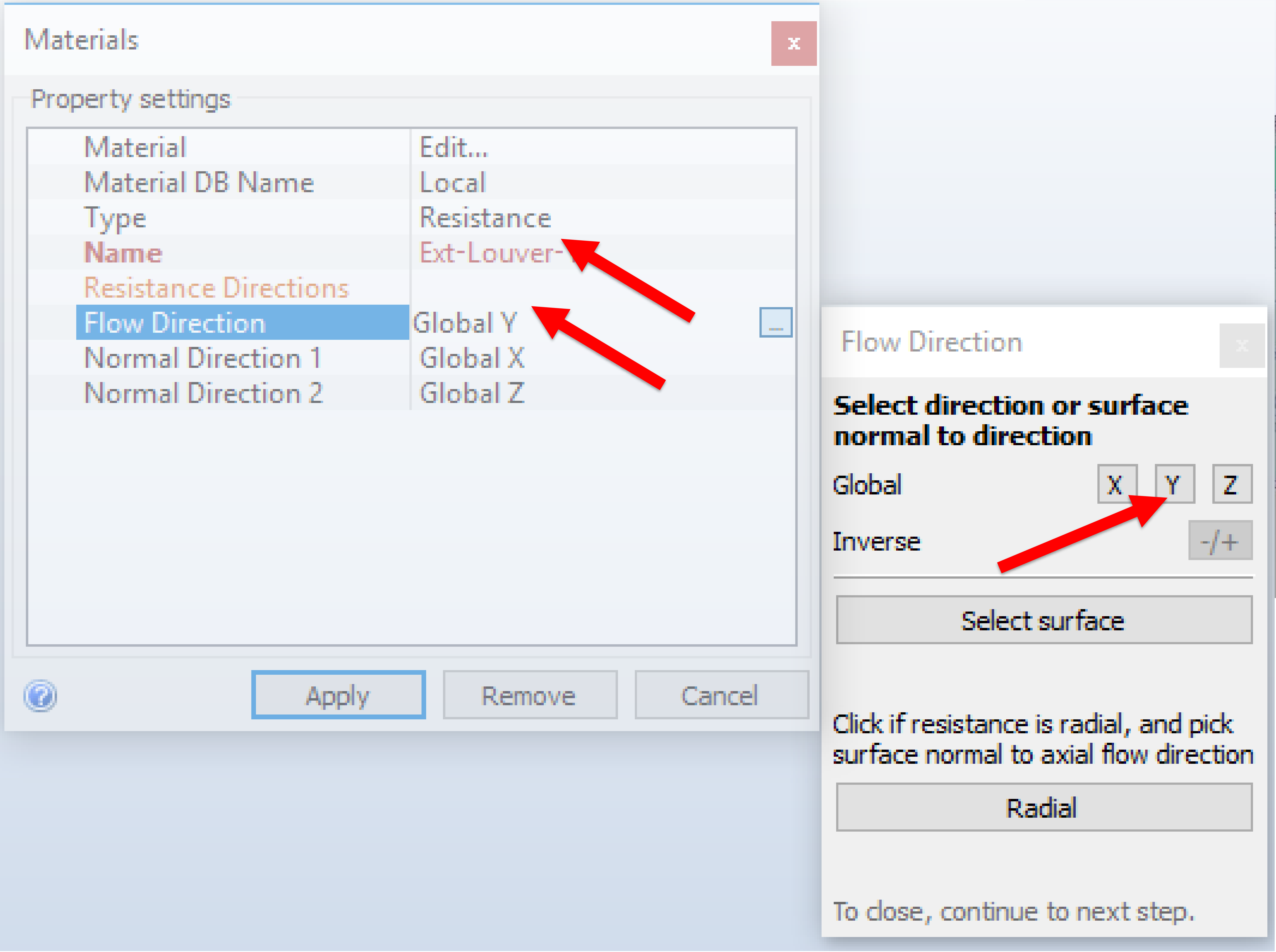


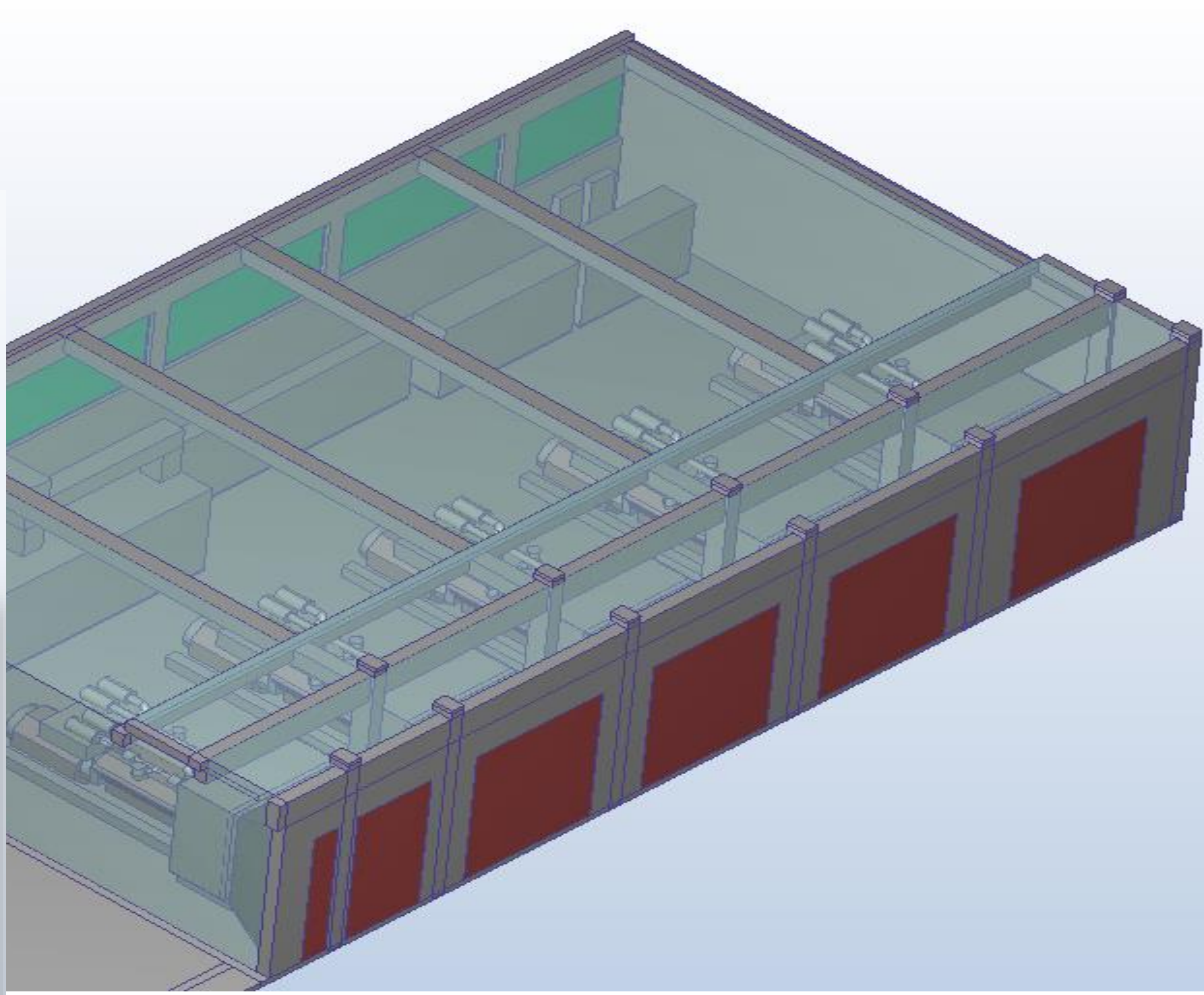
Aluminum material assigned for engine and alternator





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

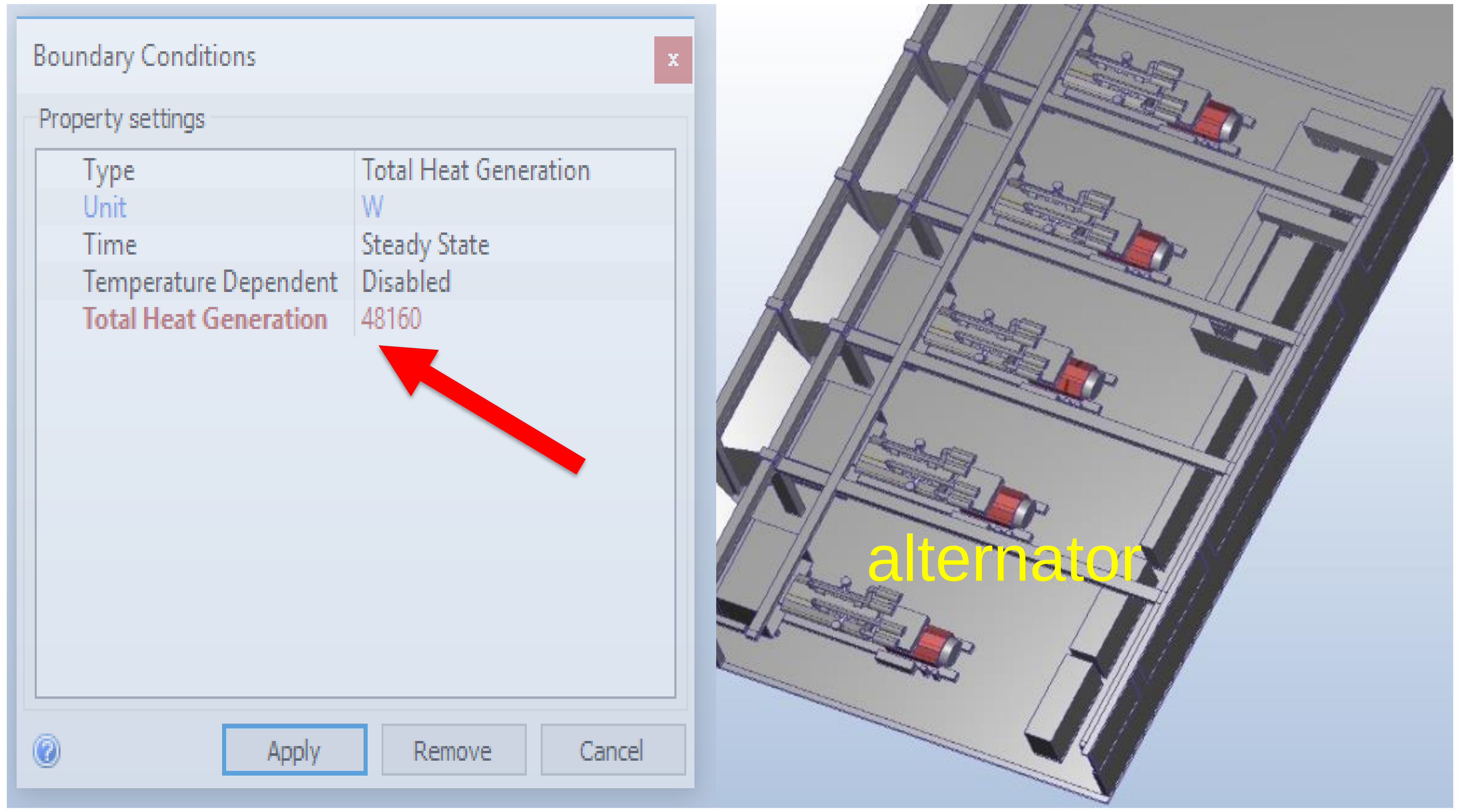


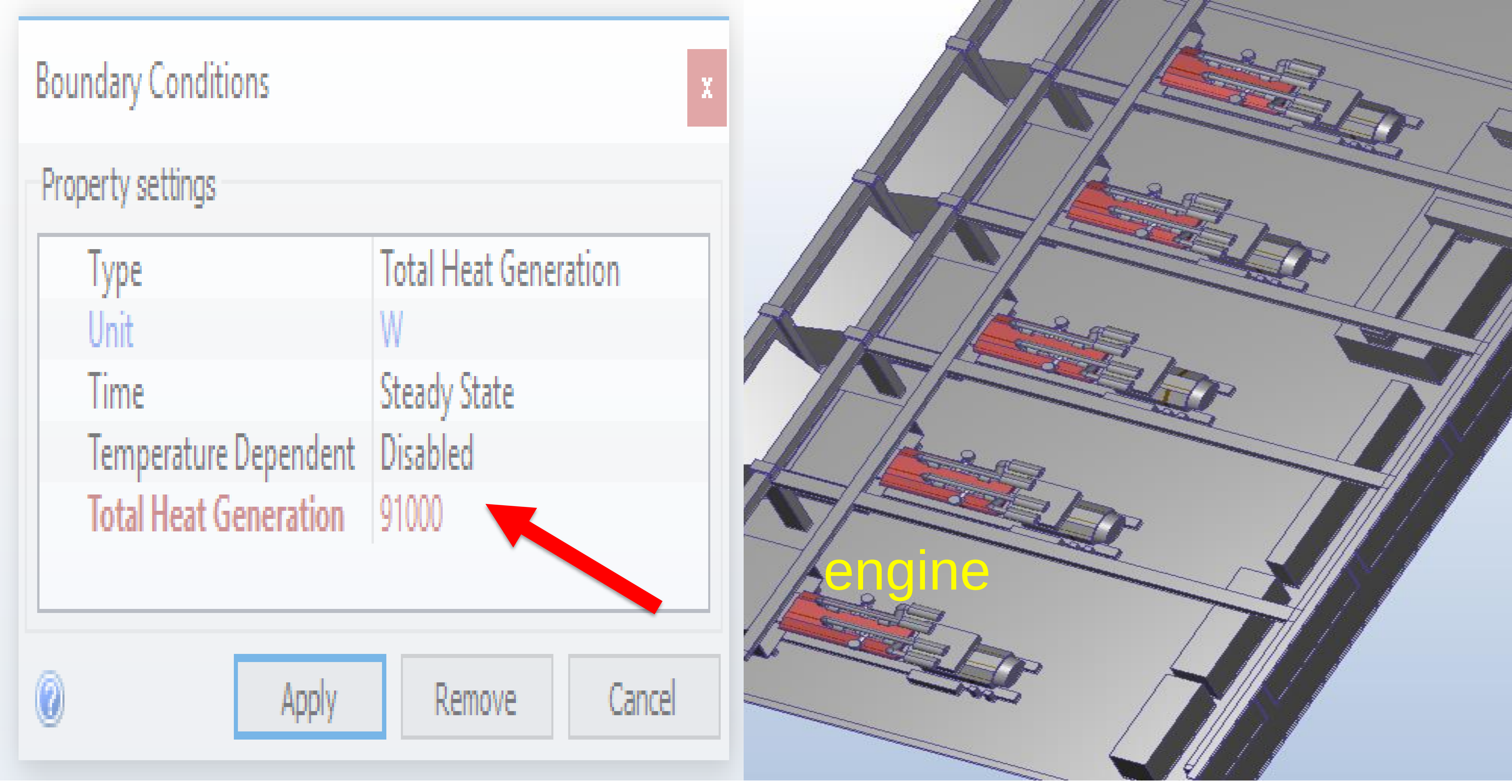


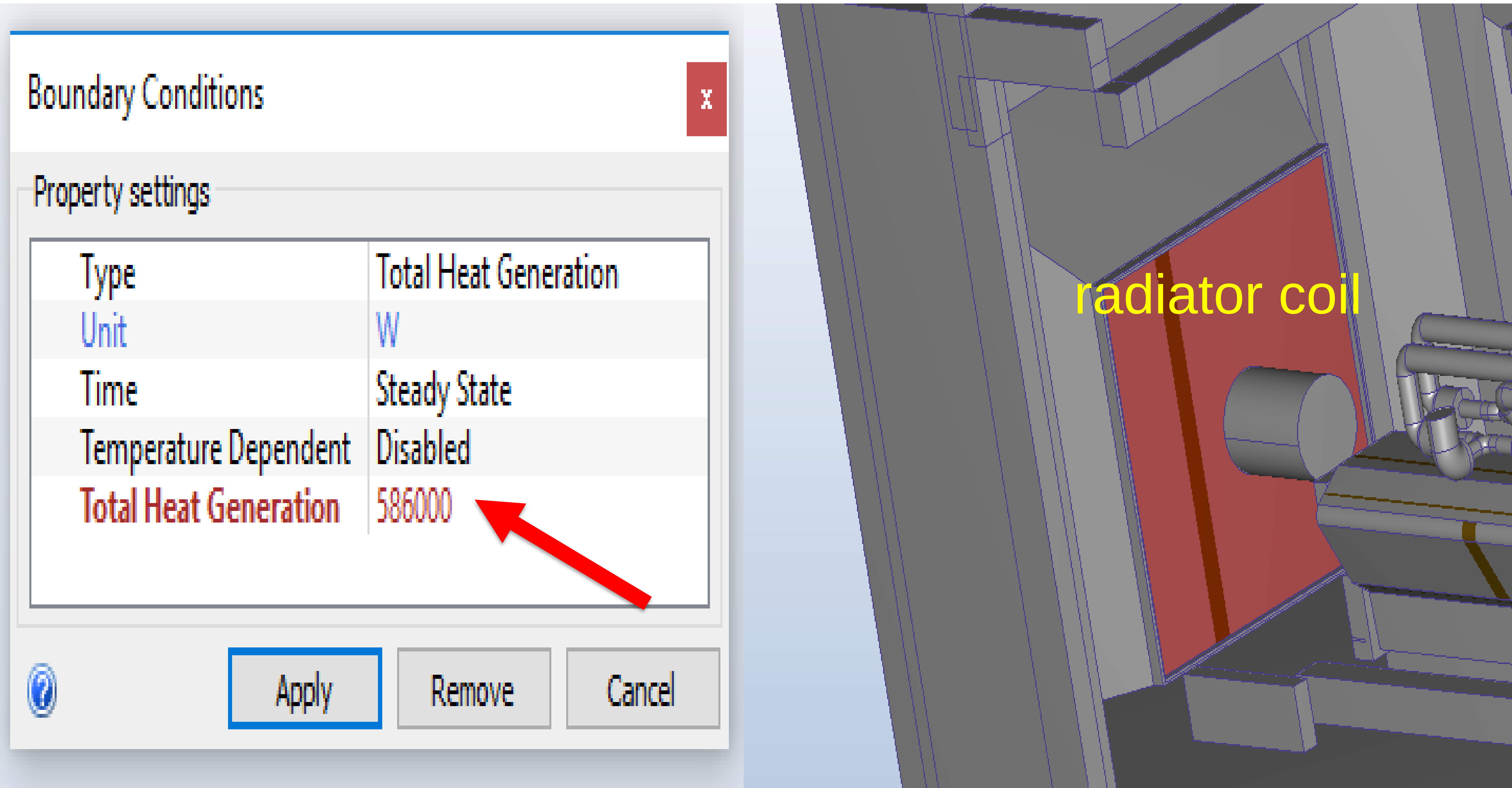
Free area ratio of 0.55 in “Through-Flow” direction. In other directions, “0” free area ratio assigned to make flow in one direction.

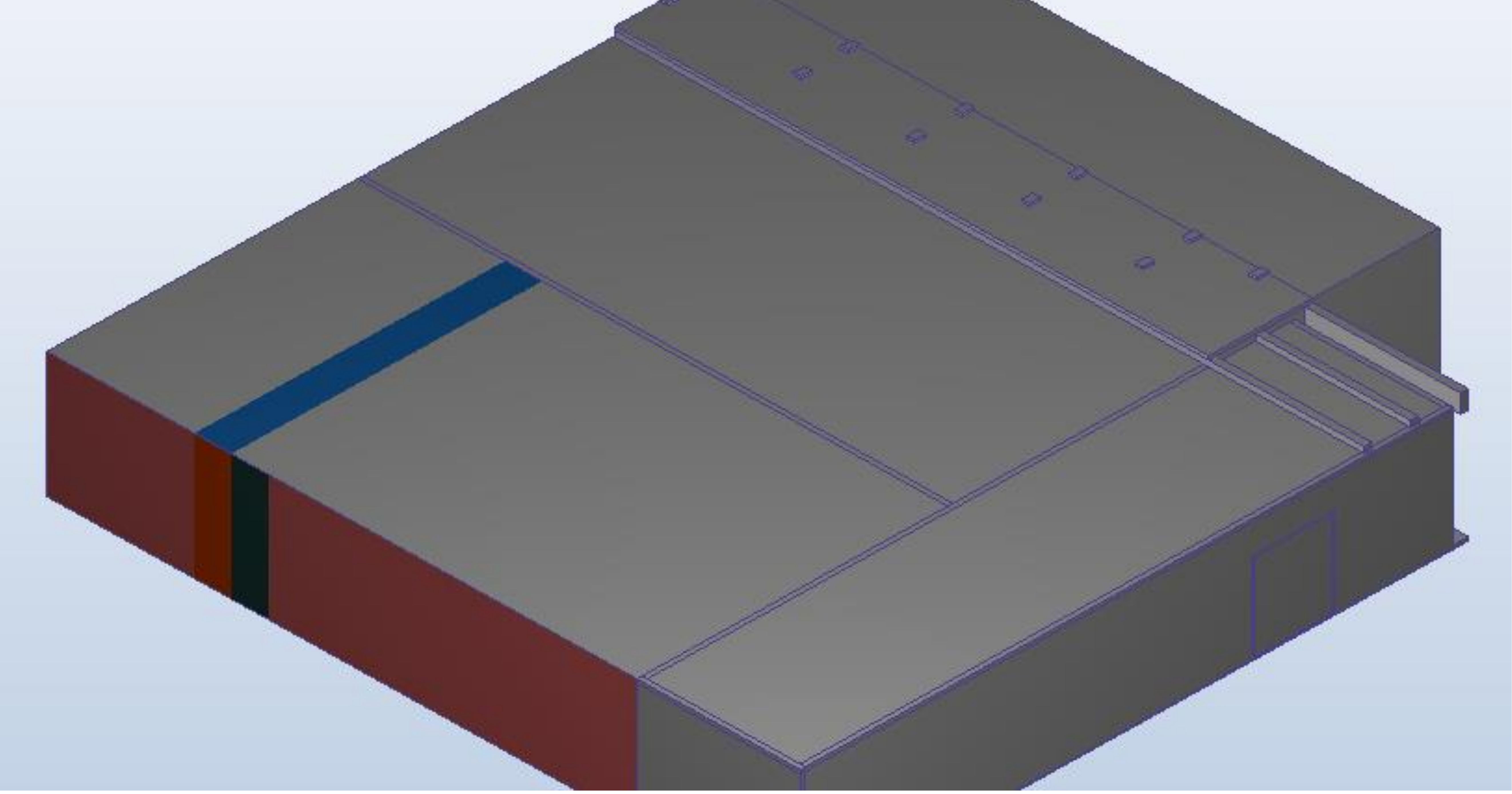
Boundary Conditions

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

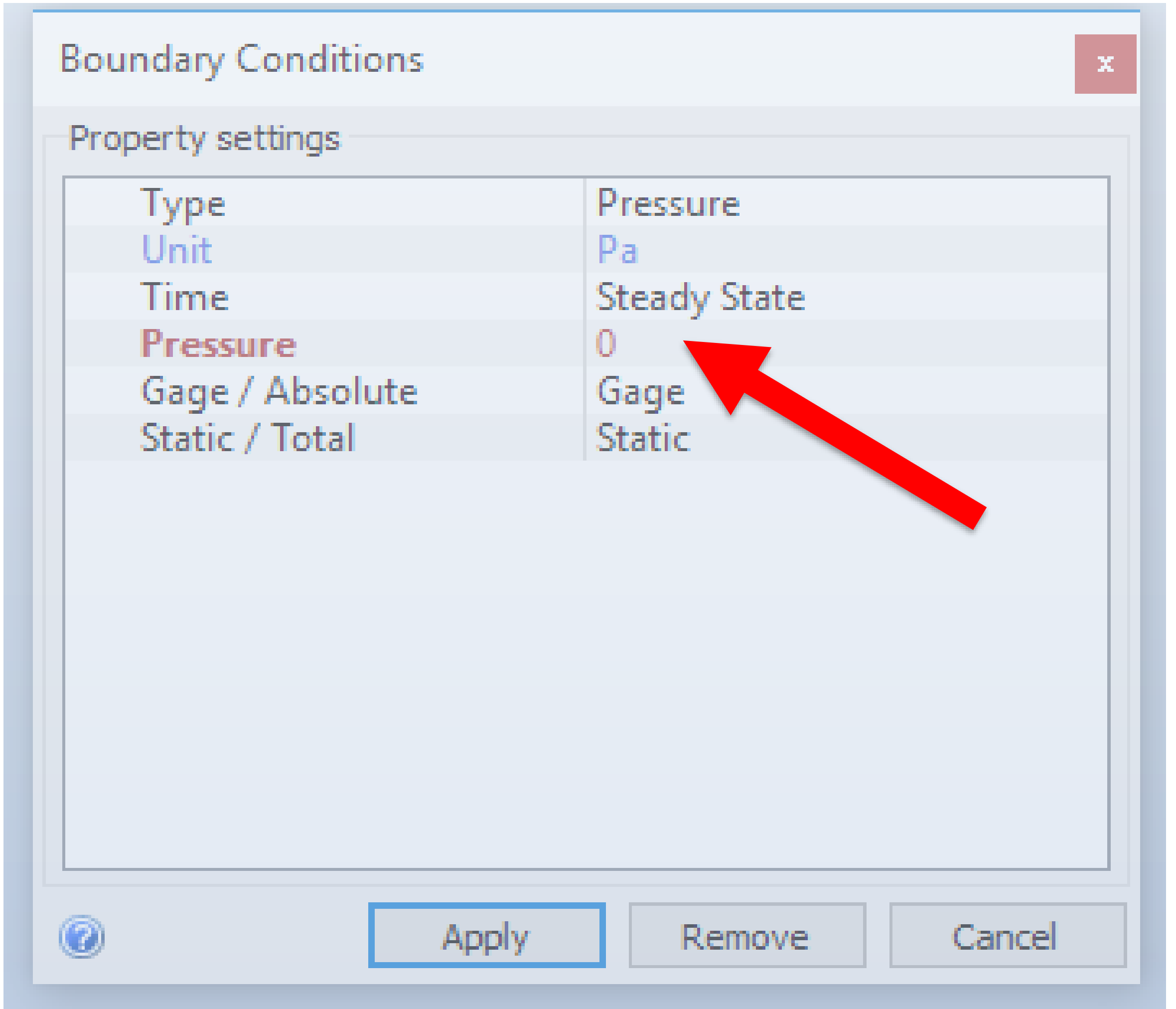


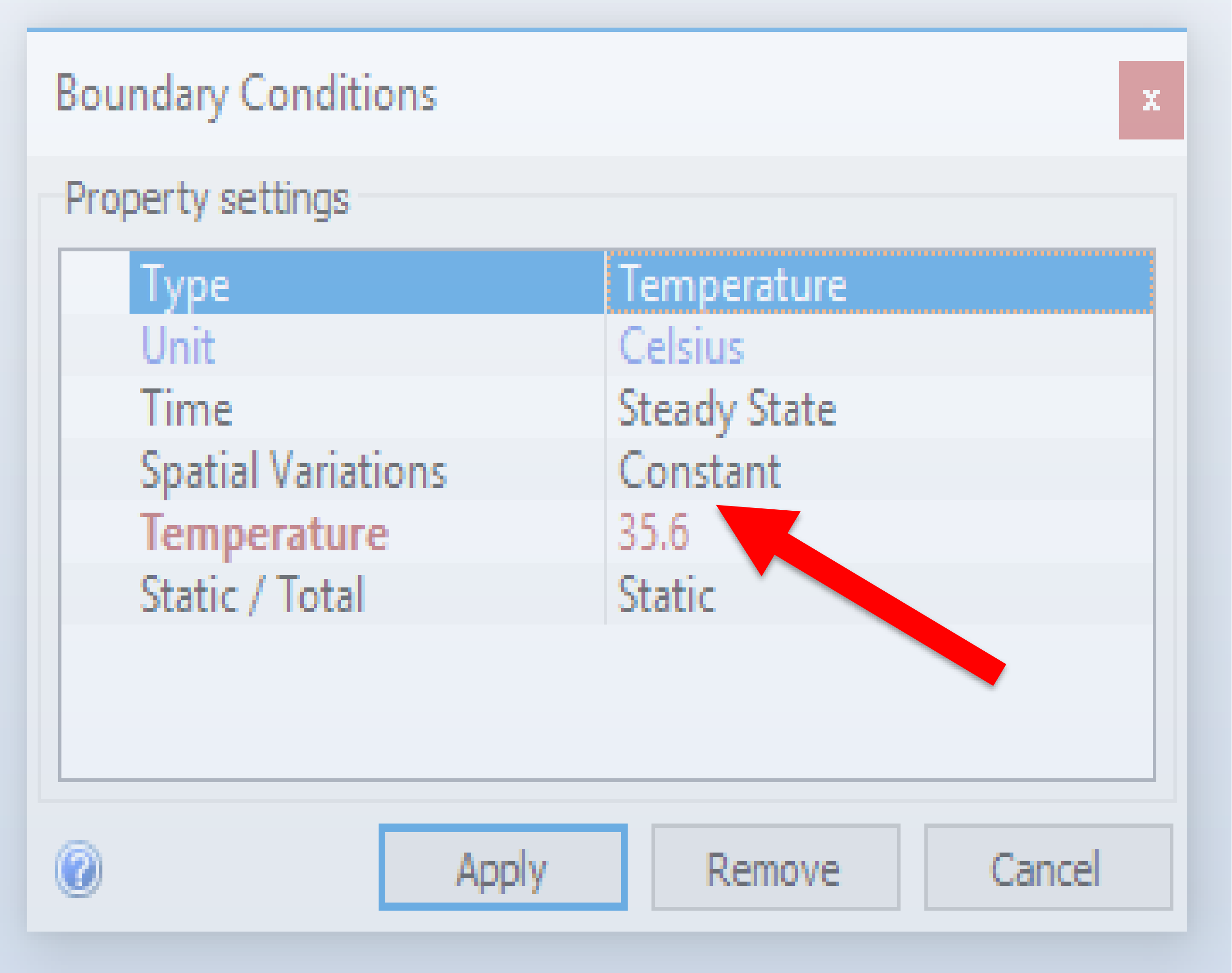


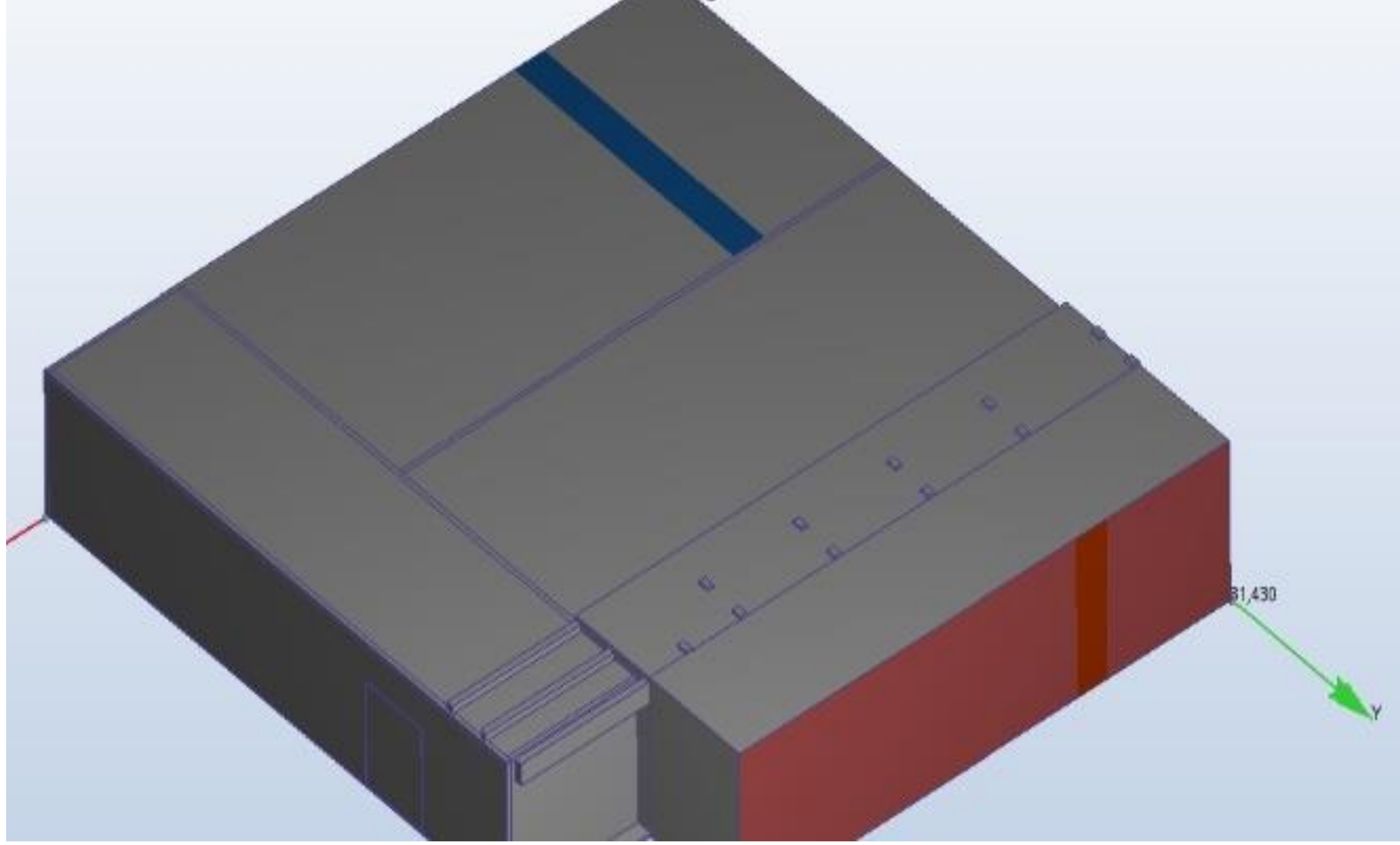




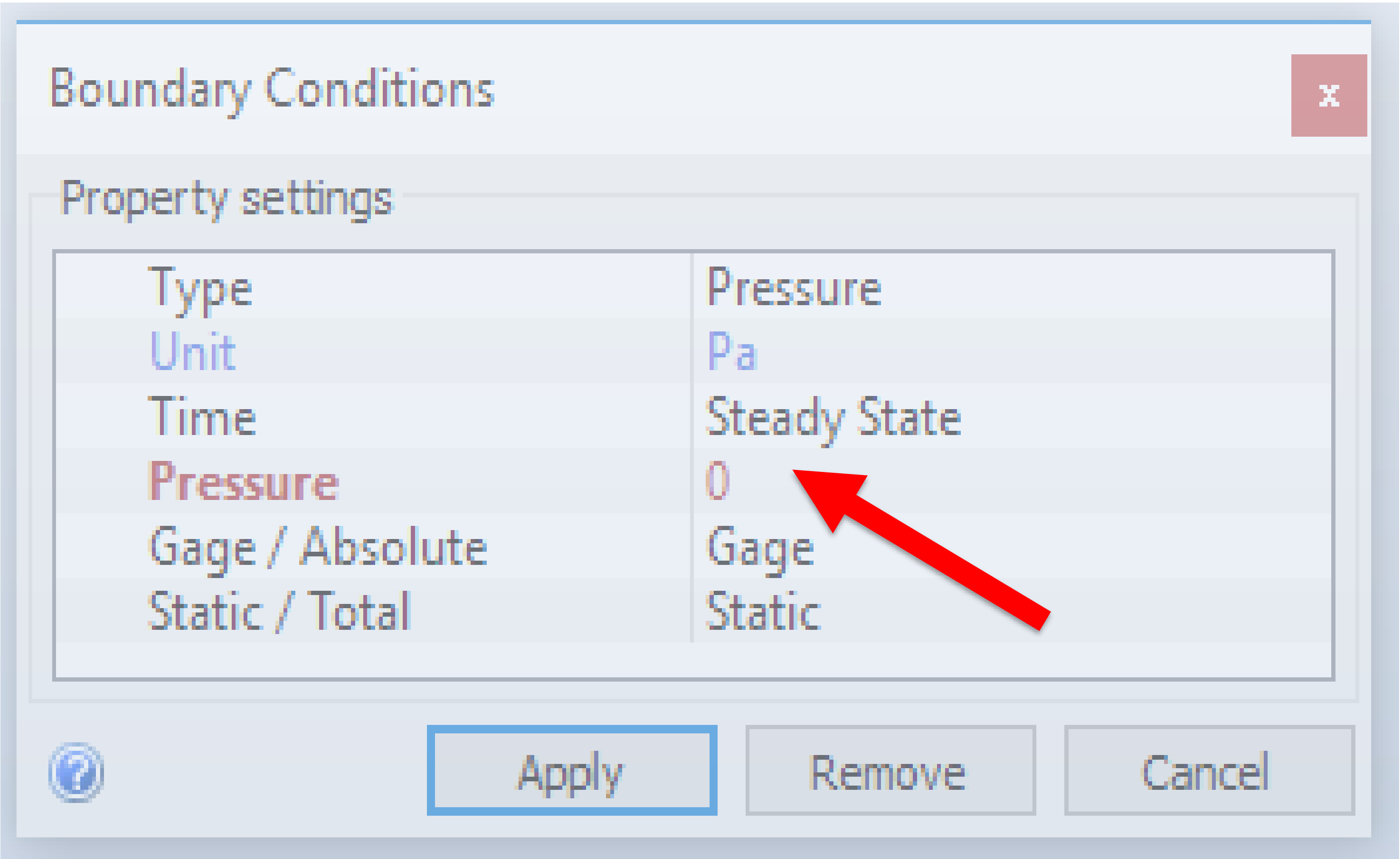
Inlet BC



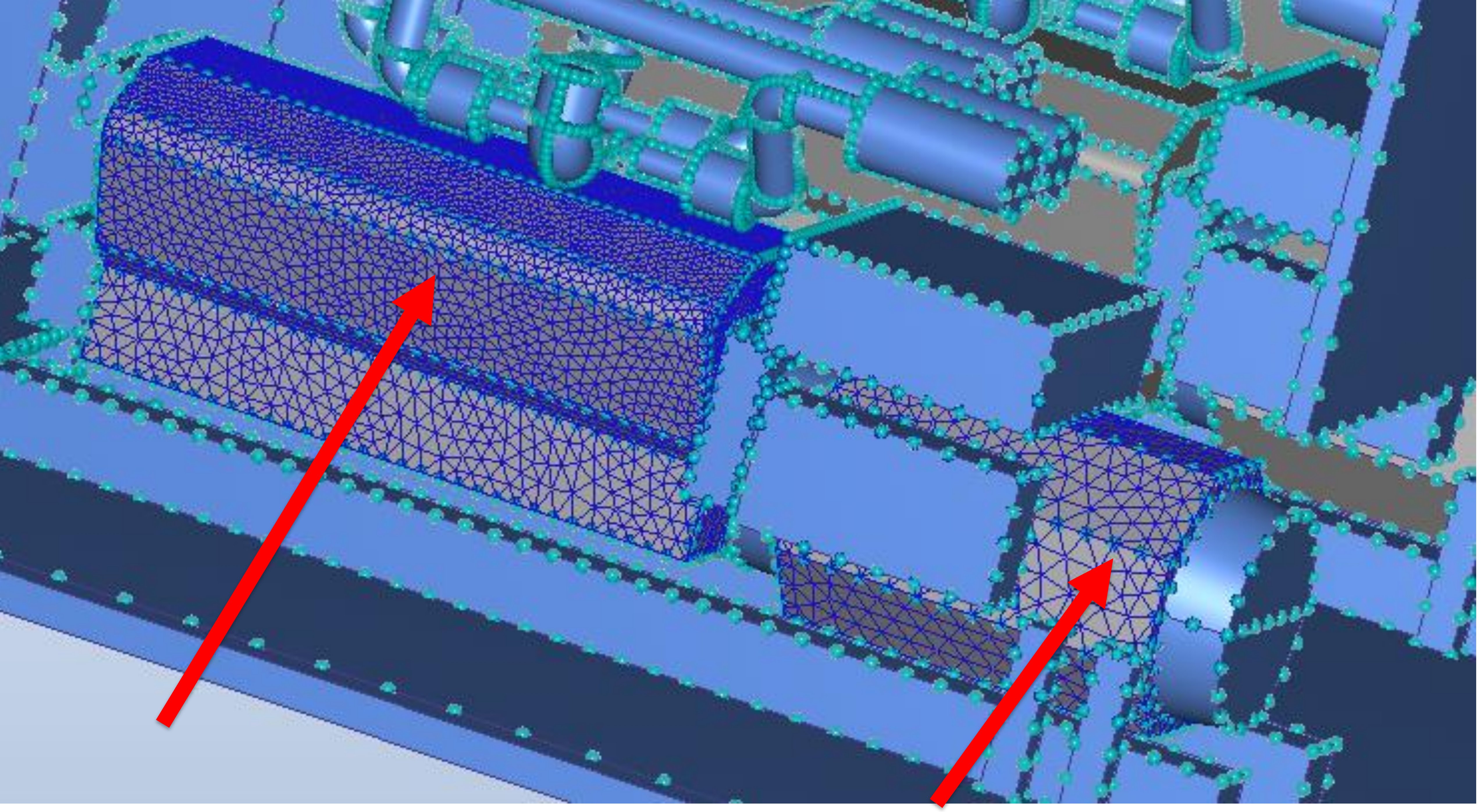




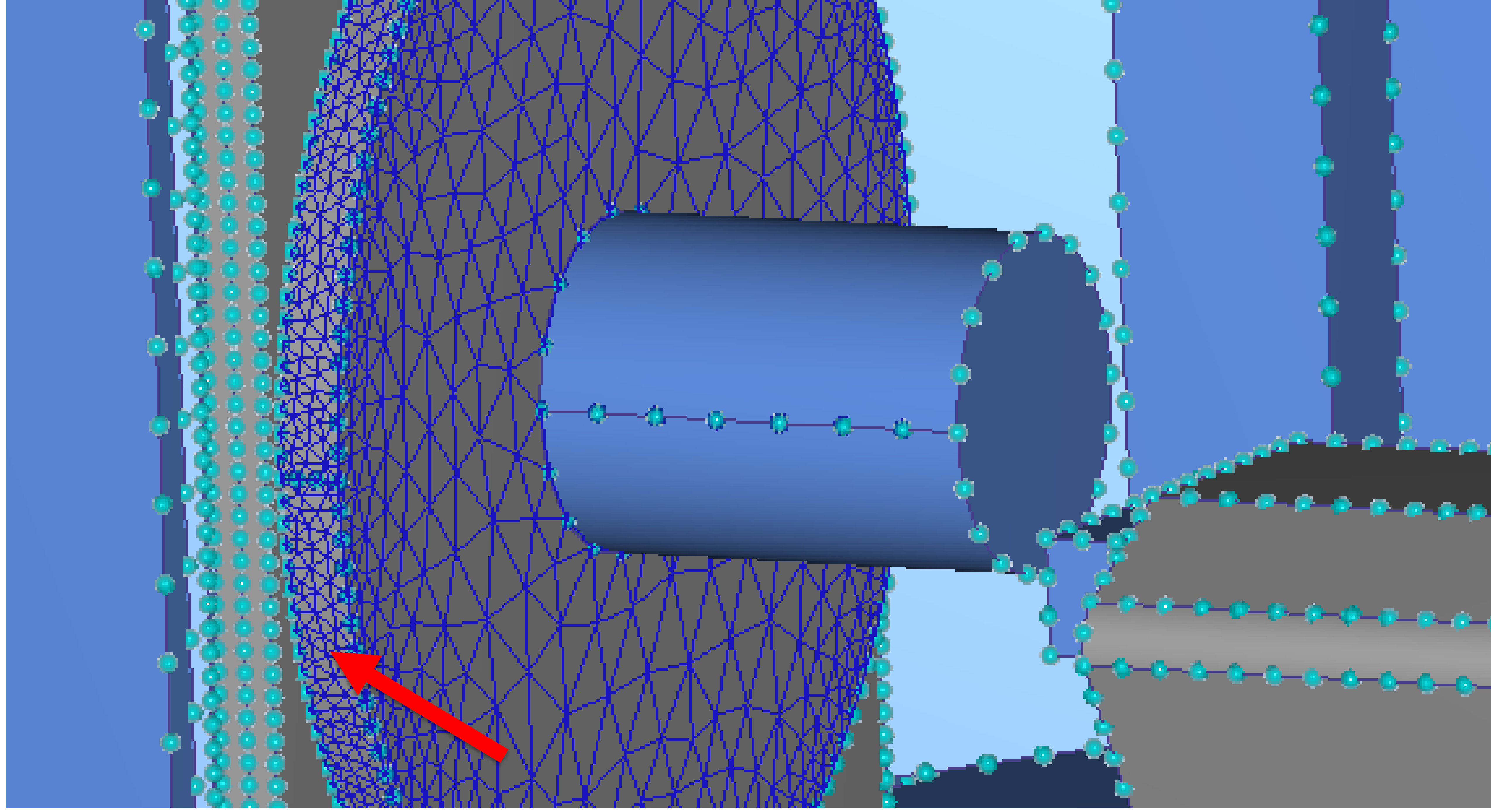
Outlet BC



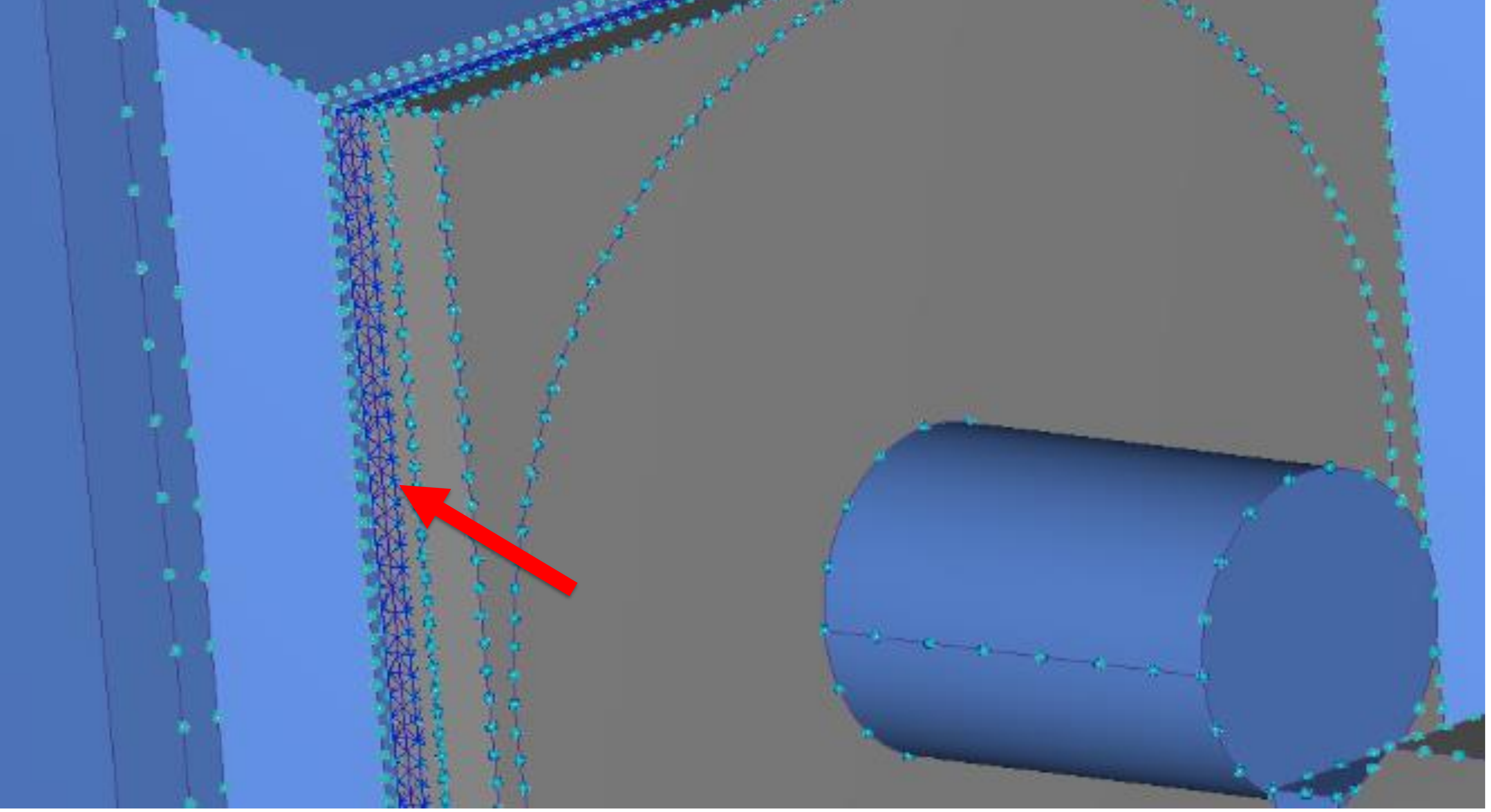
Meshing



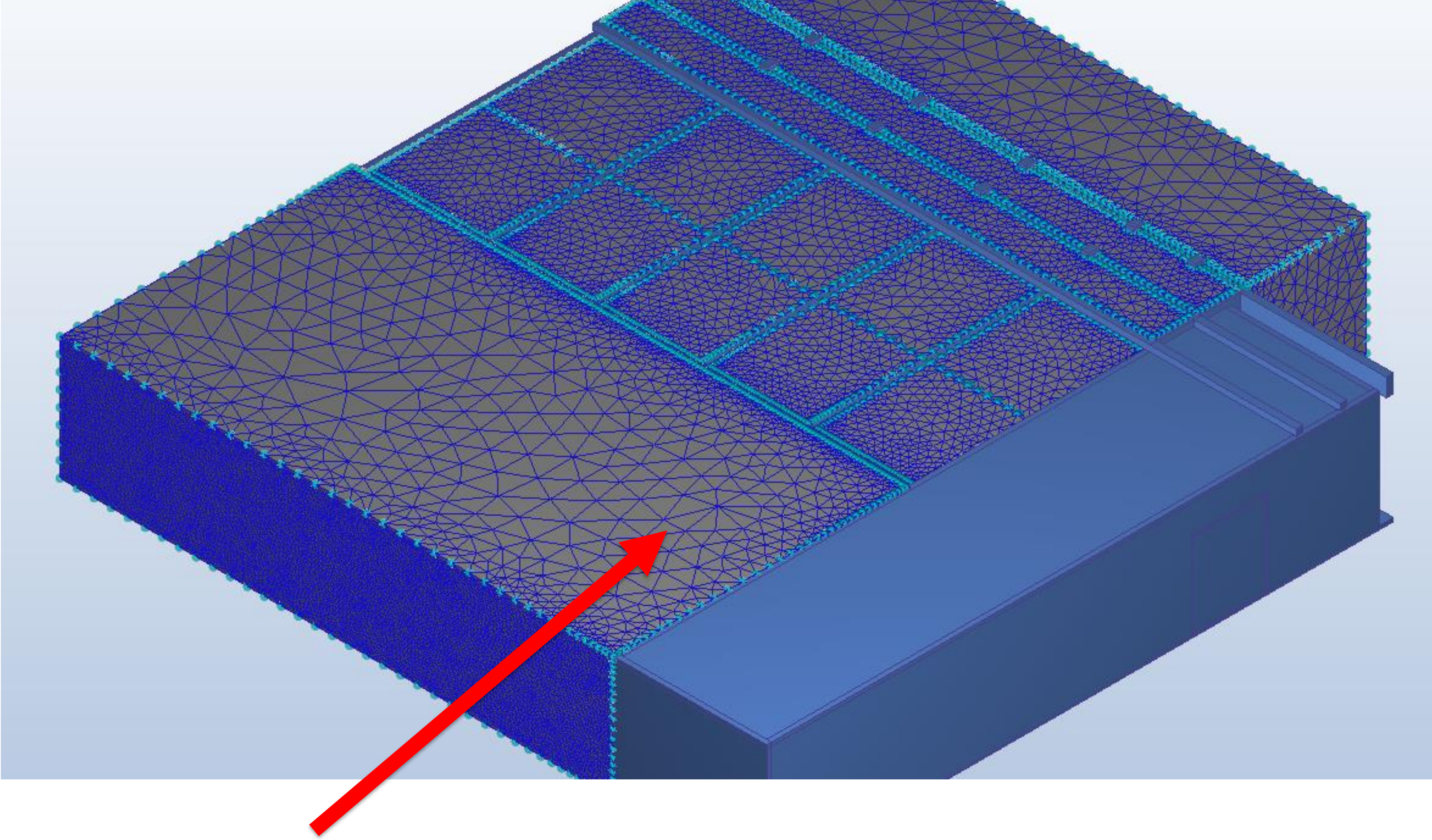
Fine mesh on engine and alternator to capture volumetric heat loss



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



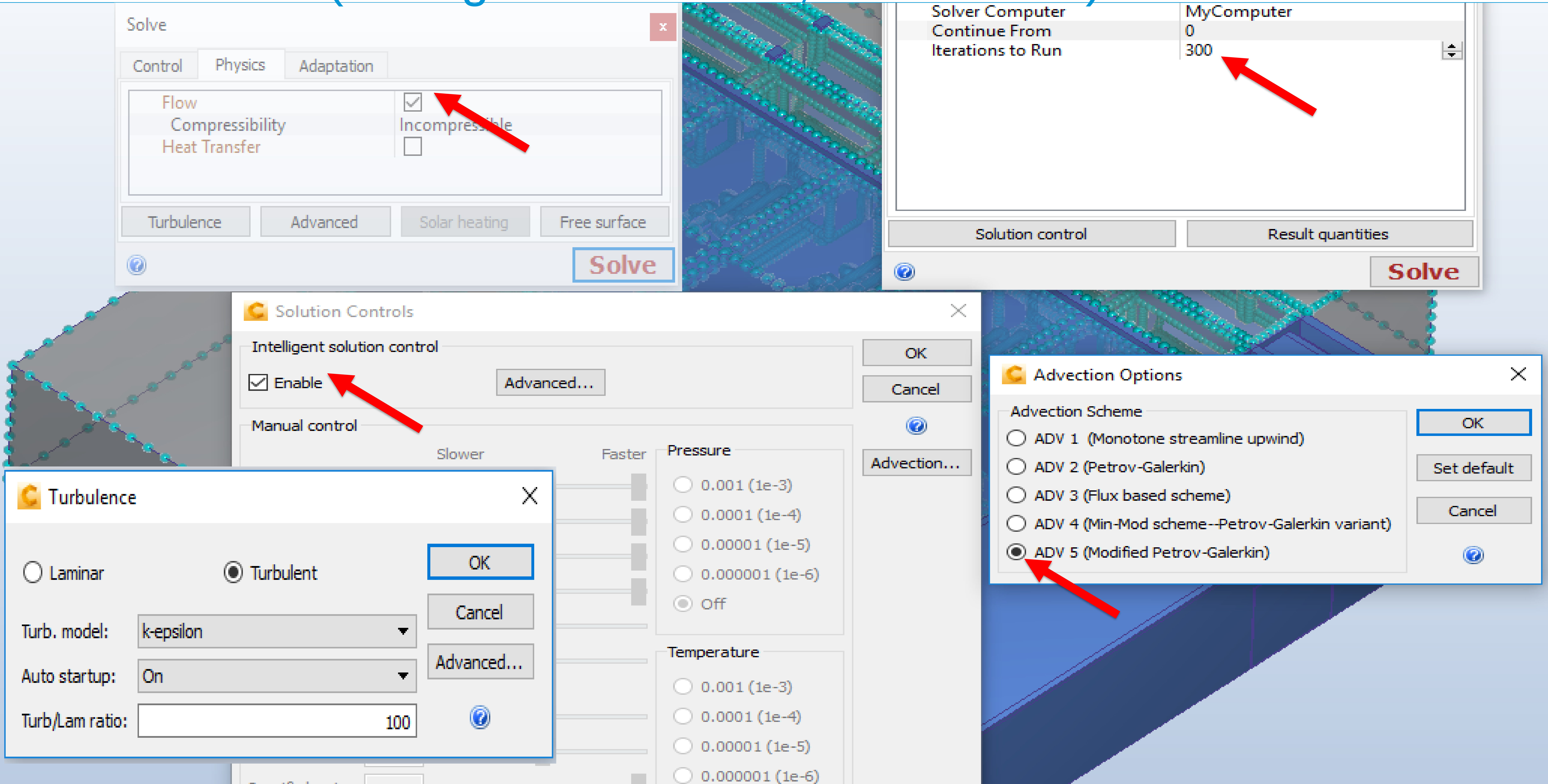
Uniform mesh to capture flow through radiator coil



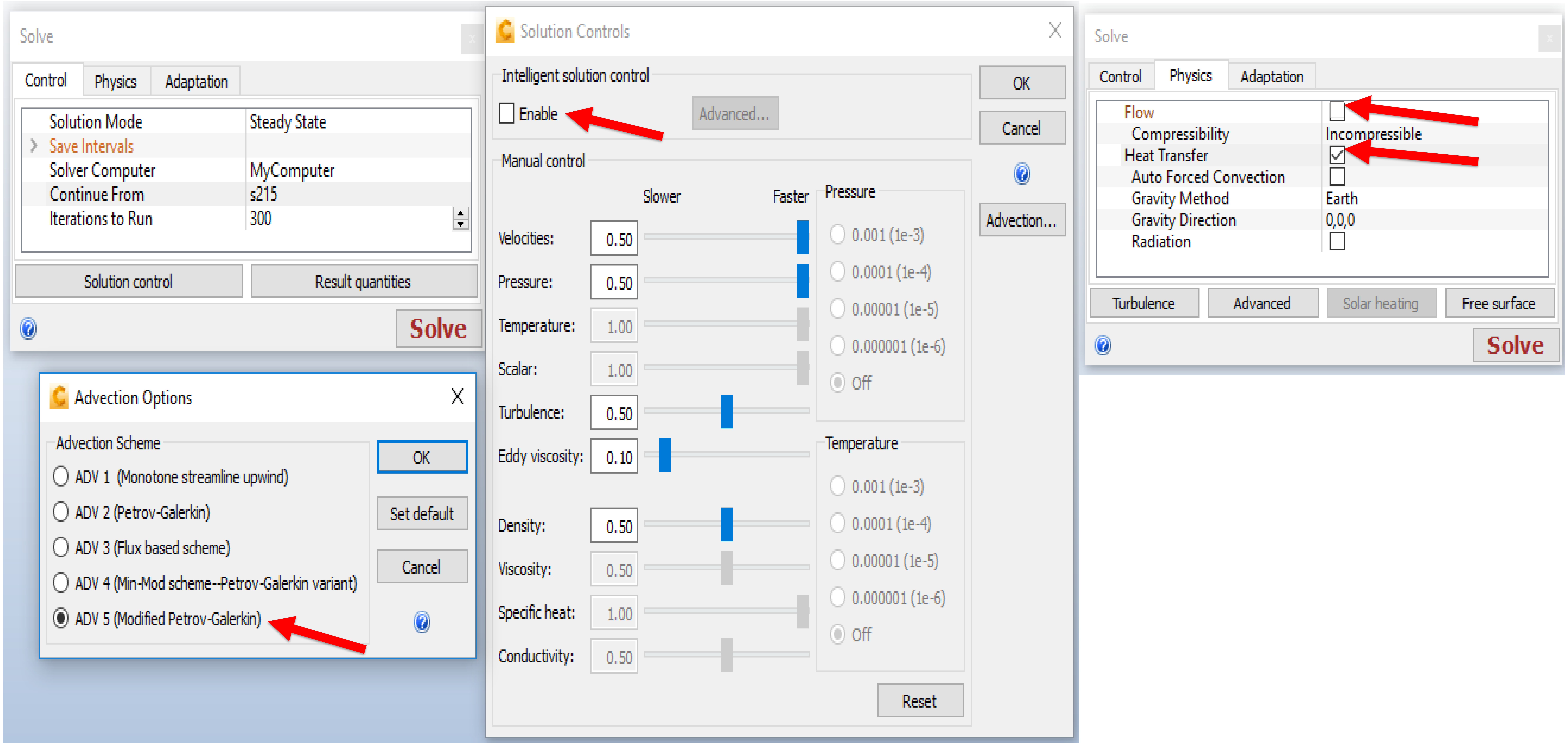
Overall mesh for the DG room CFD model

Solver Settings

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

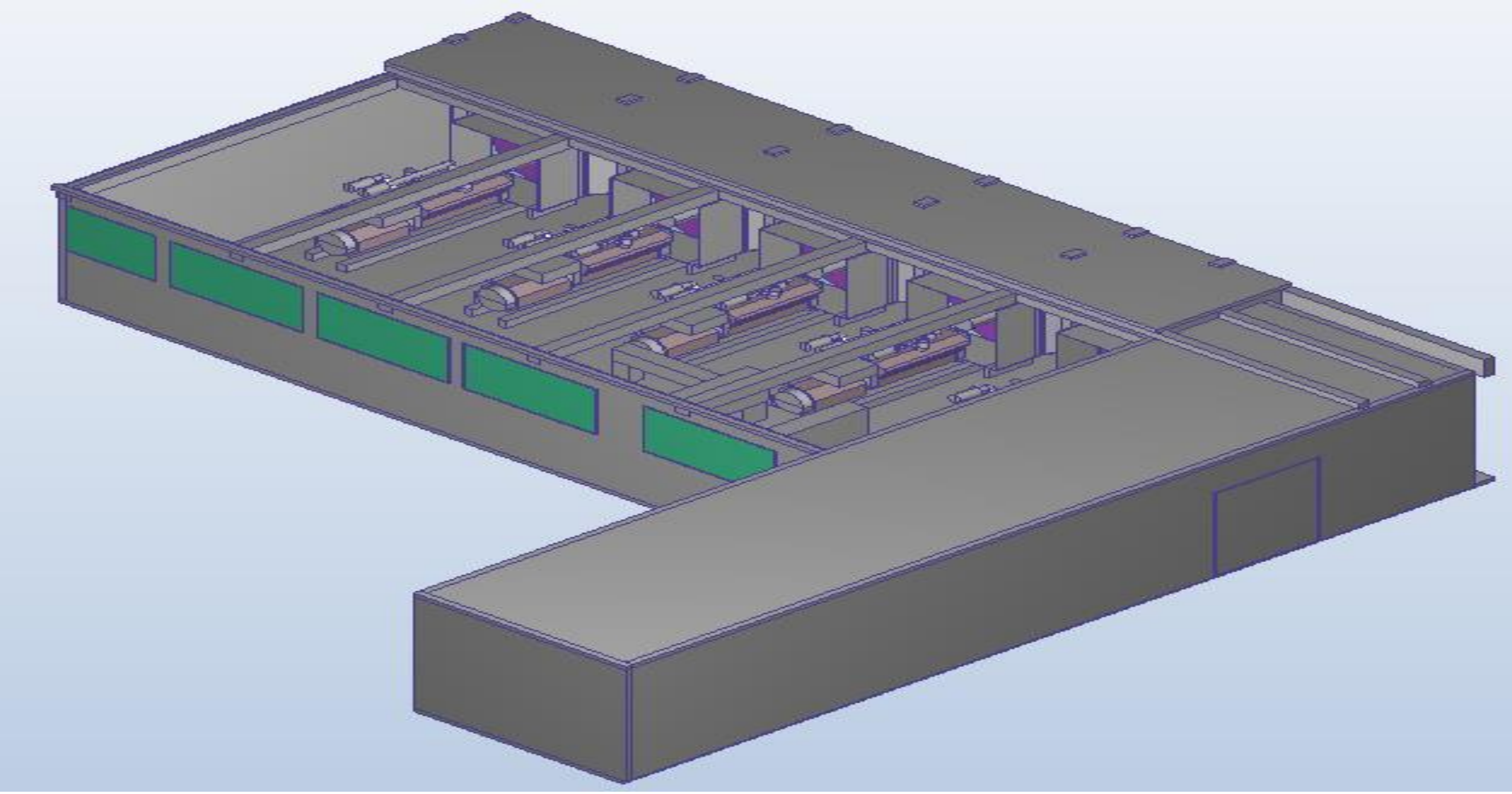


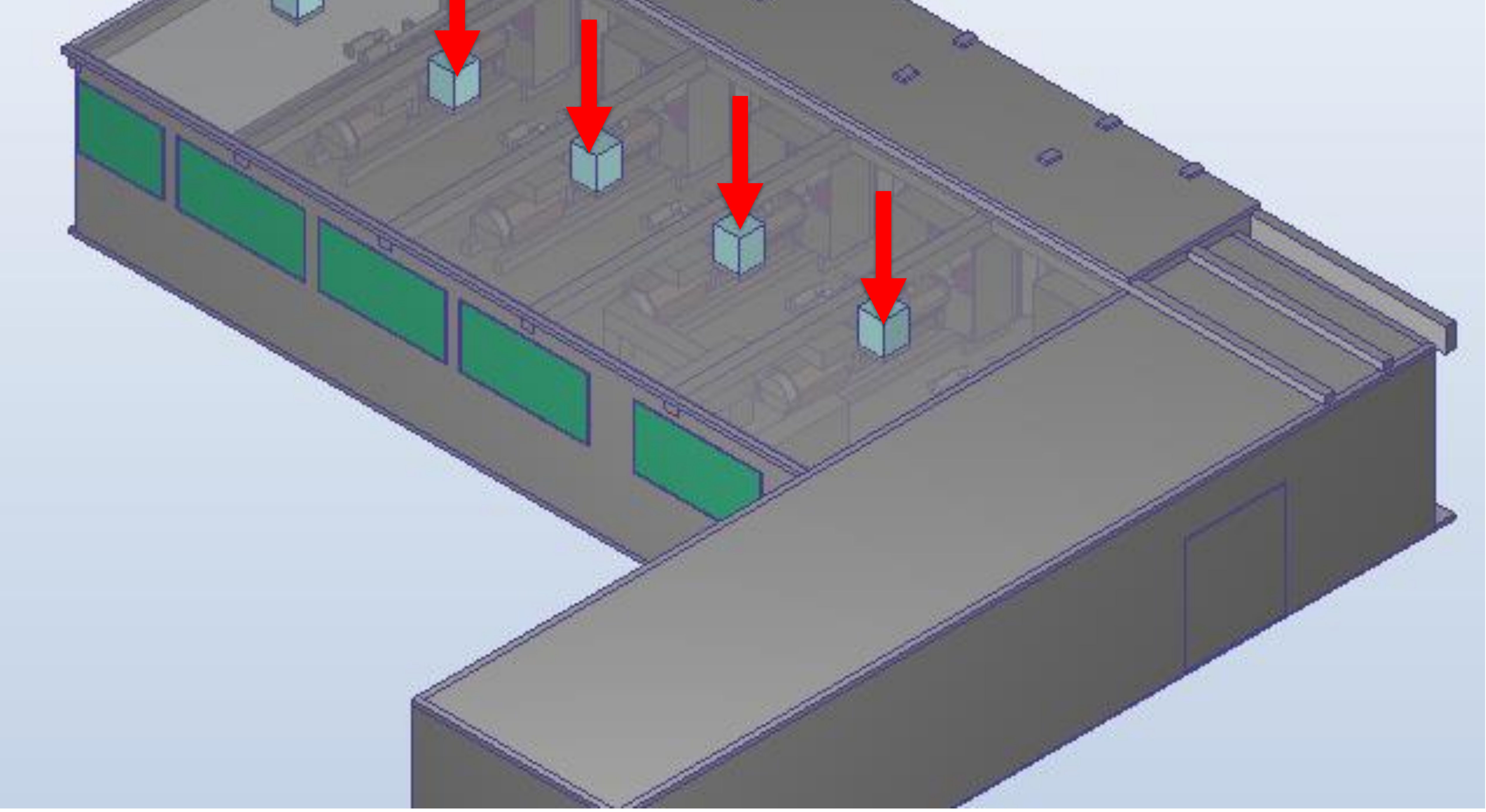
Solver settings for heat transfer analysis (Intelligent control-off, Advection 5)

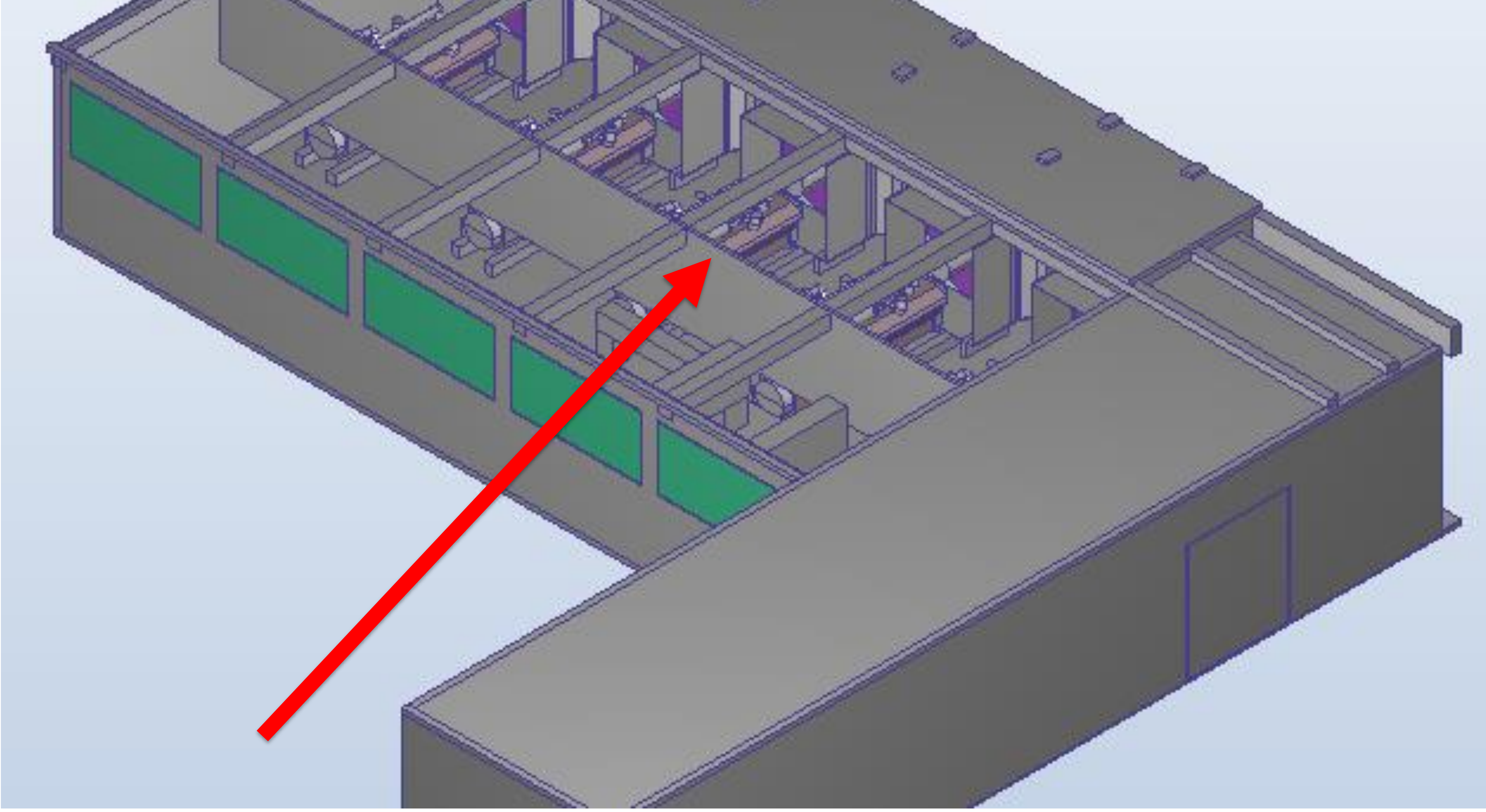


Key Results for Design Performance

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







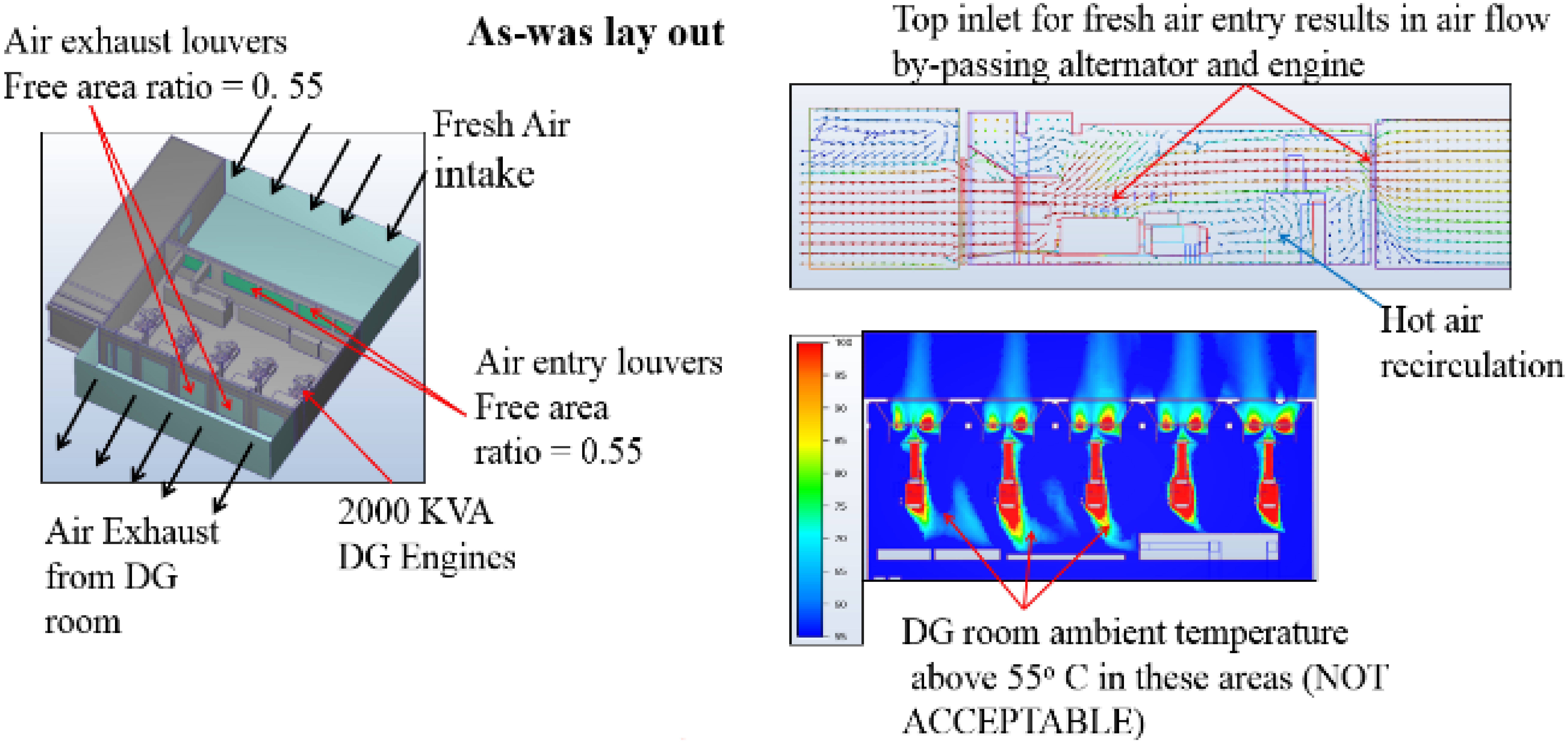
Design option 1

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

Design option 2

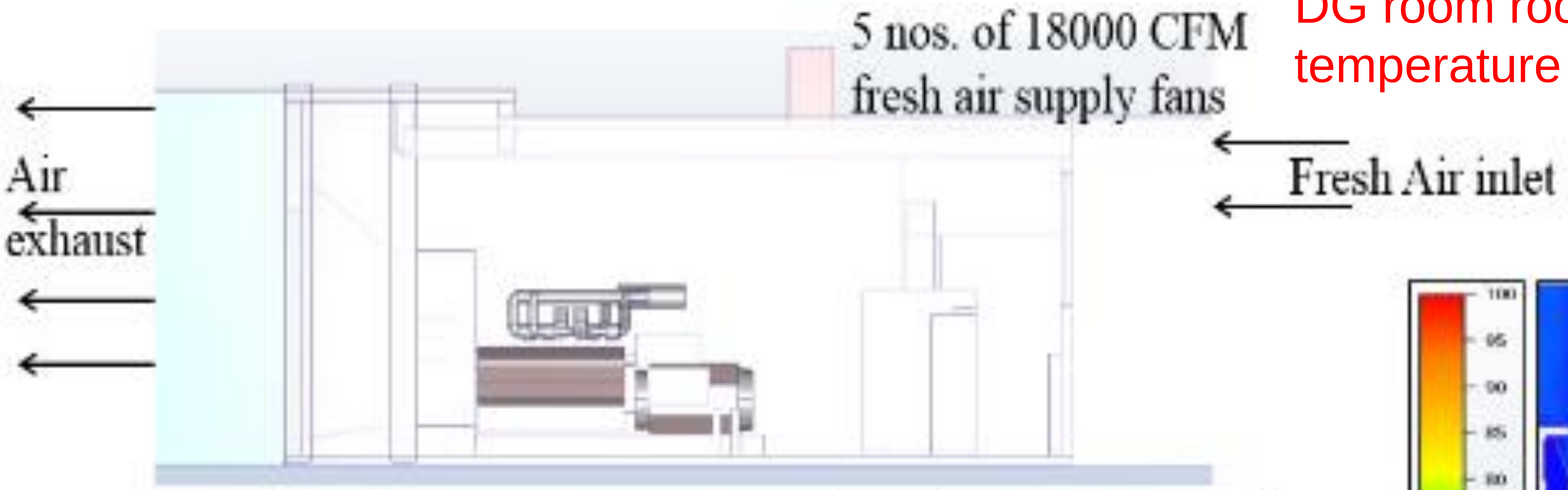
In this option an additional wall is considered containing the heat and airflow closer to the DG

CFD analysis results for as-was layout

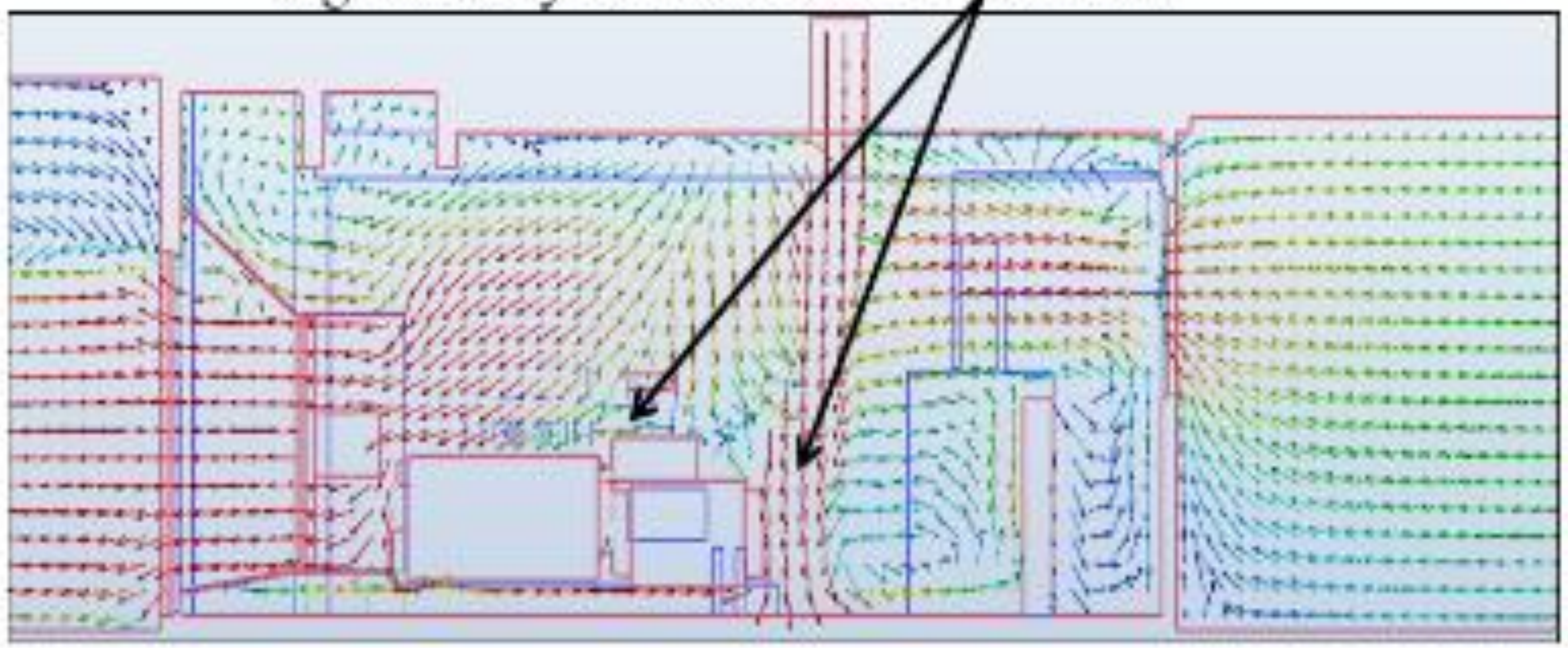


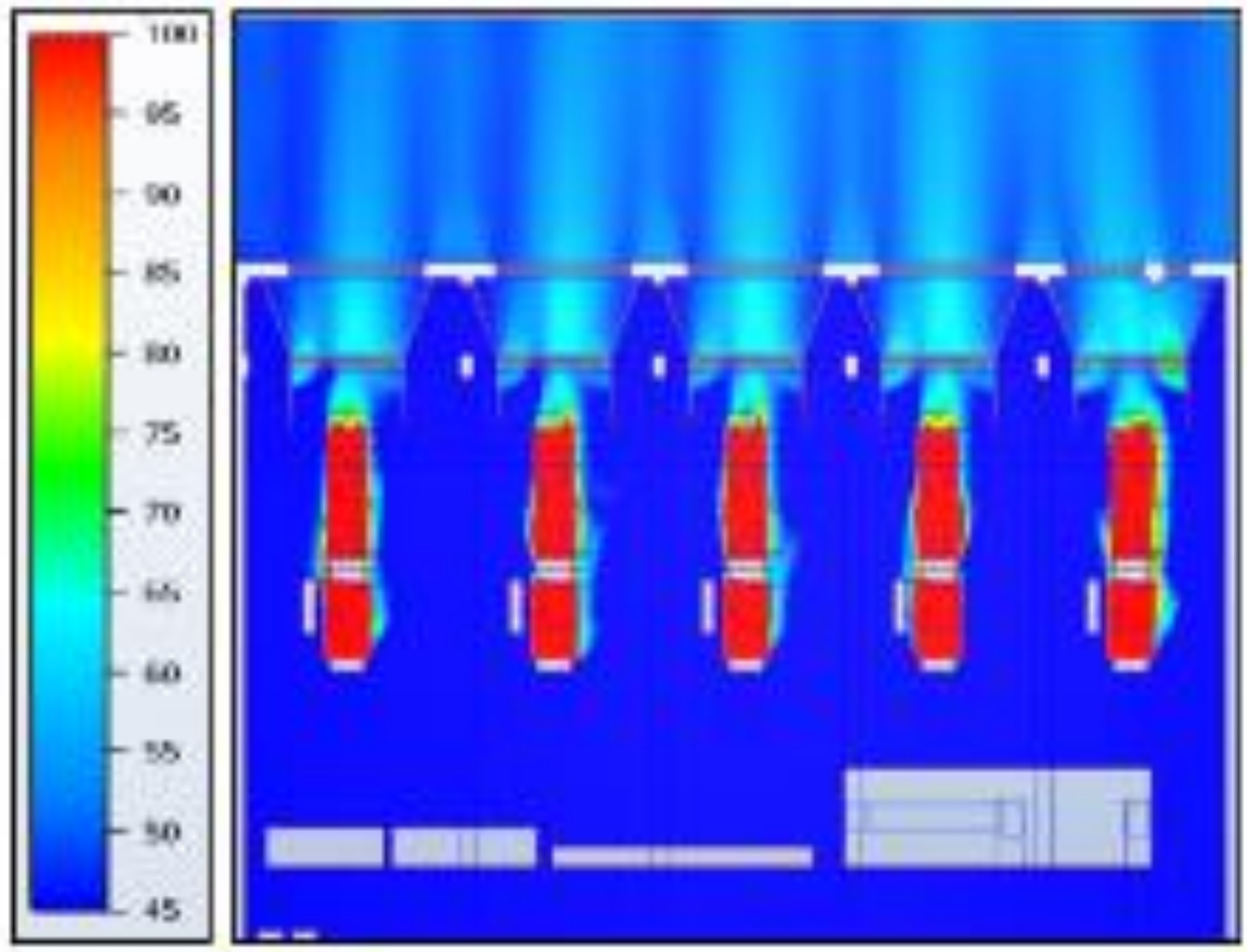
Design option-1:CFD analysis results for roof supply fans

Mechanical ventilation through supply fans at the DG room roof (expensive), though room ambient temperature was brought down to 45^0 C



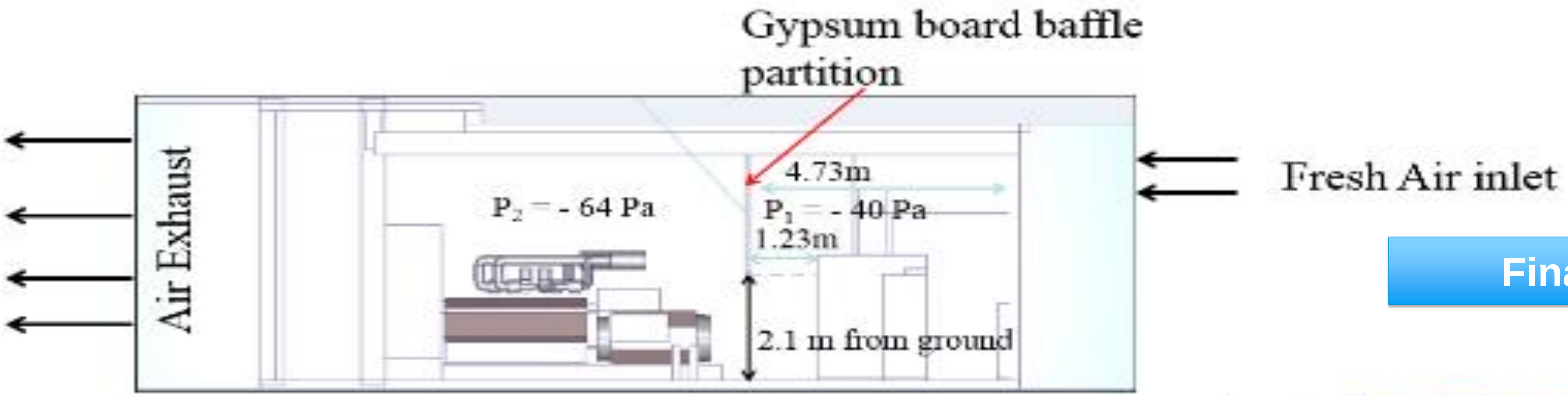
Air flow to the engine and alternator improved significantly due to fan flow from roof



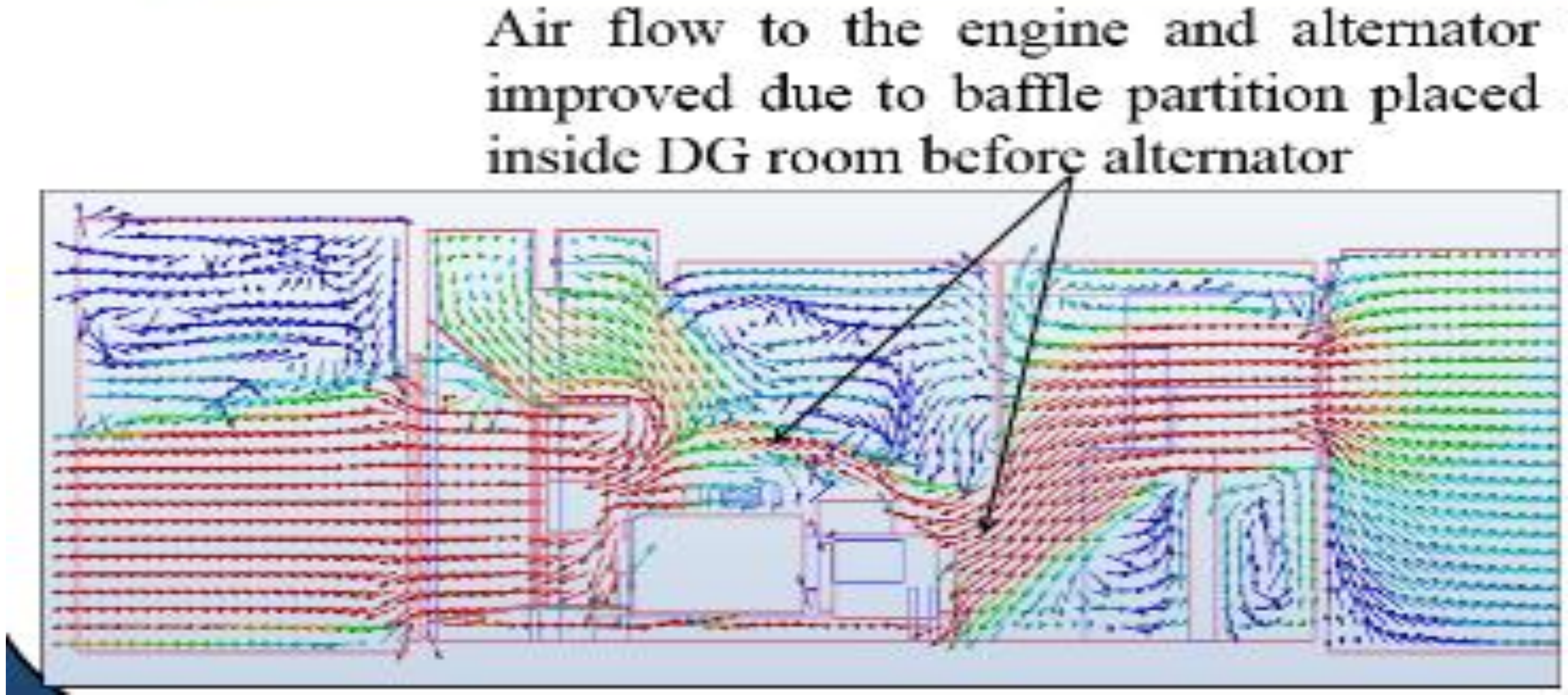


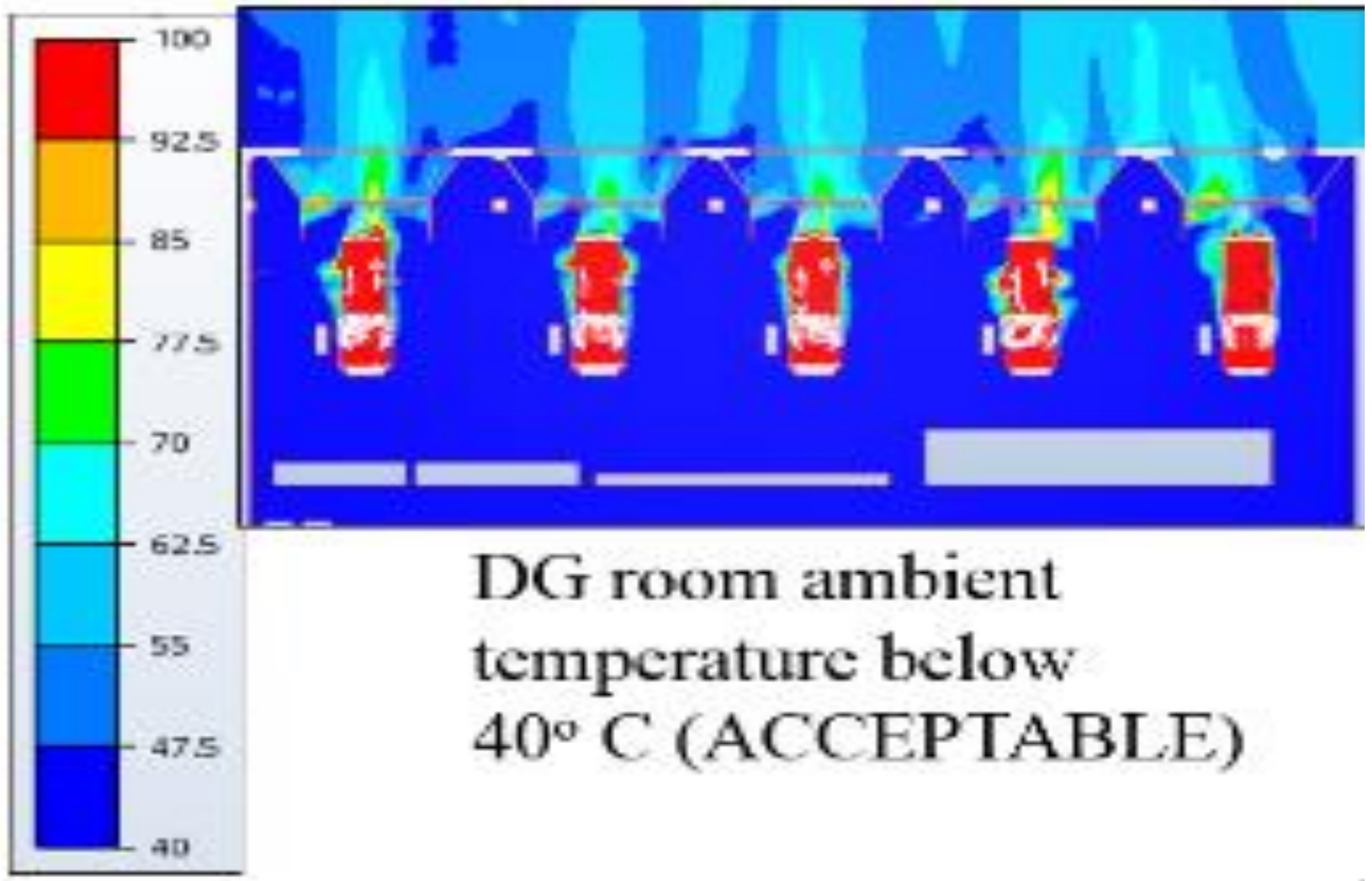
DG room ambient temperature below 45 Deg C

Design option-2: CFD analysis results with baffle partition



Final Design





Outcome- DG room

Design Option 2 results in:

- DG room ambient temperature below 40^0 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):

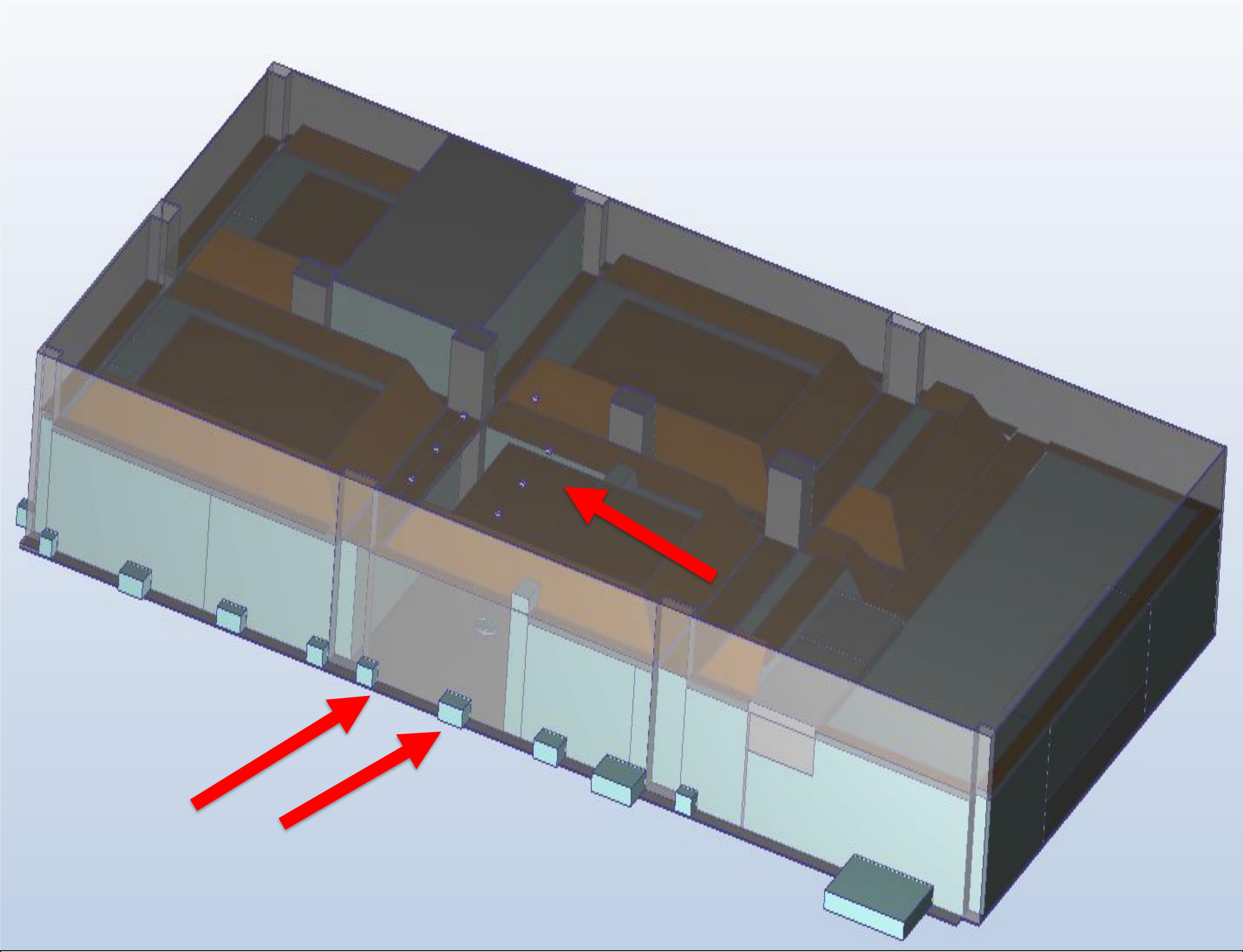
- Smoke visibility of minimum10 m
- Smoke temperature limit of 60^0 C

Convention Center Exhibition Hall

CAD Model

Create CAD model in Revit.

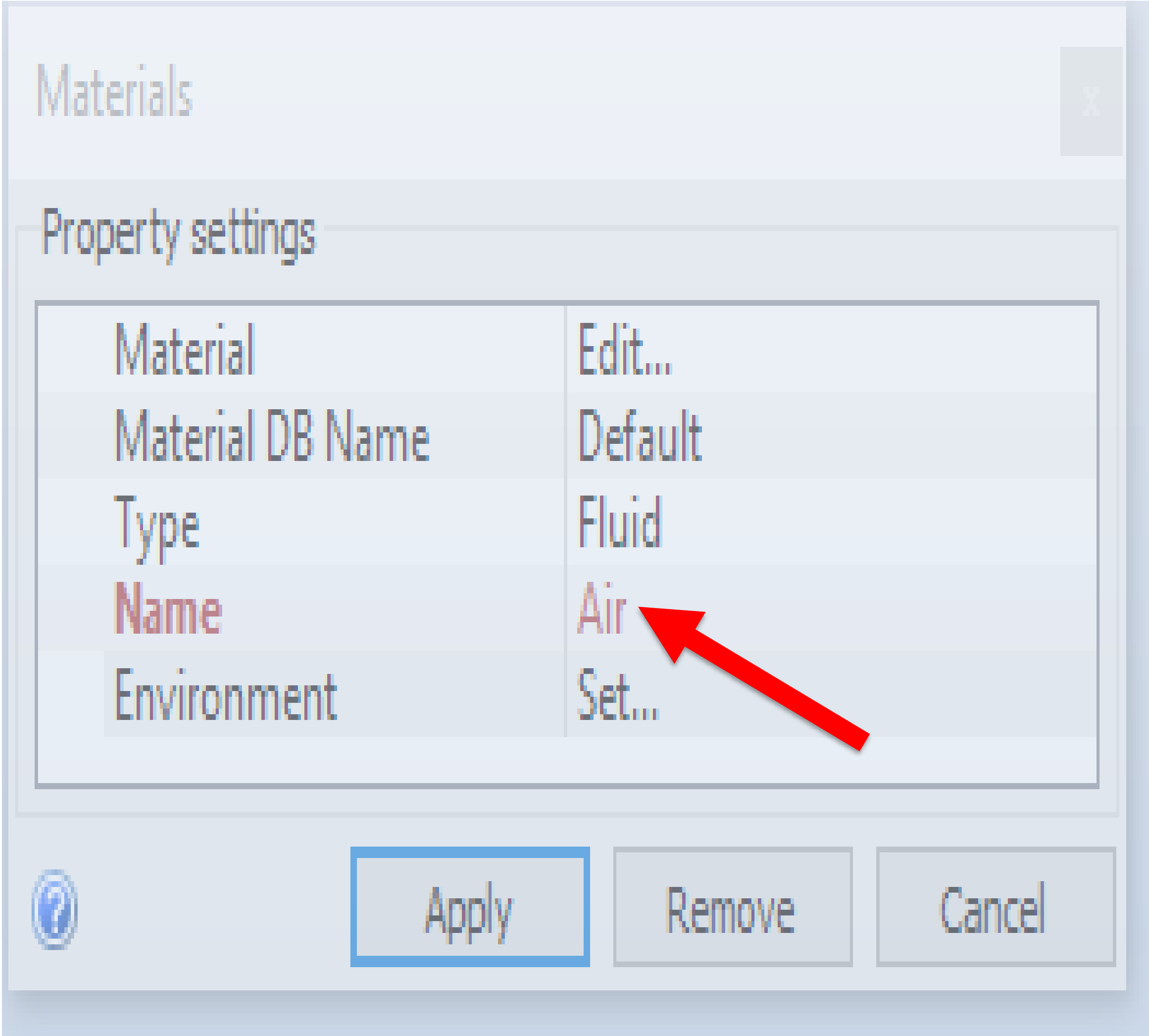
- Simplify geometrical details
- Add inlet and outlet extensions
- Create fire part

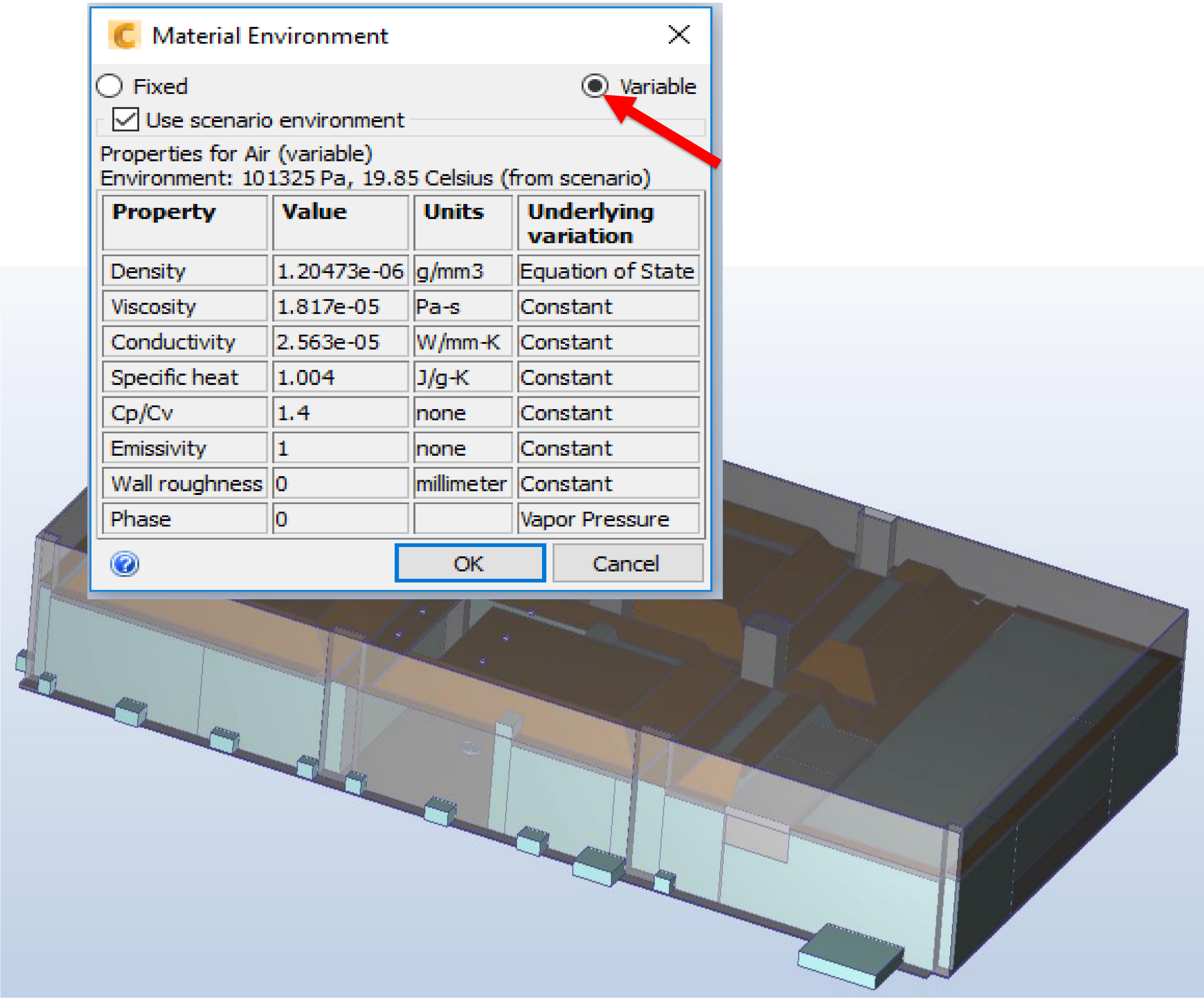


Exhibition hall size = 245L X160W X22.4H
No. of smoke zones = 10
Zone size for CFD = 80m X 40 m
Fire mode ventilation design= 12 ACH

Materials

Air domain material

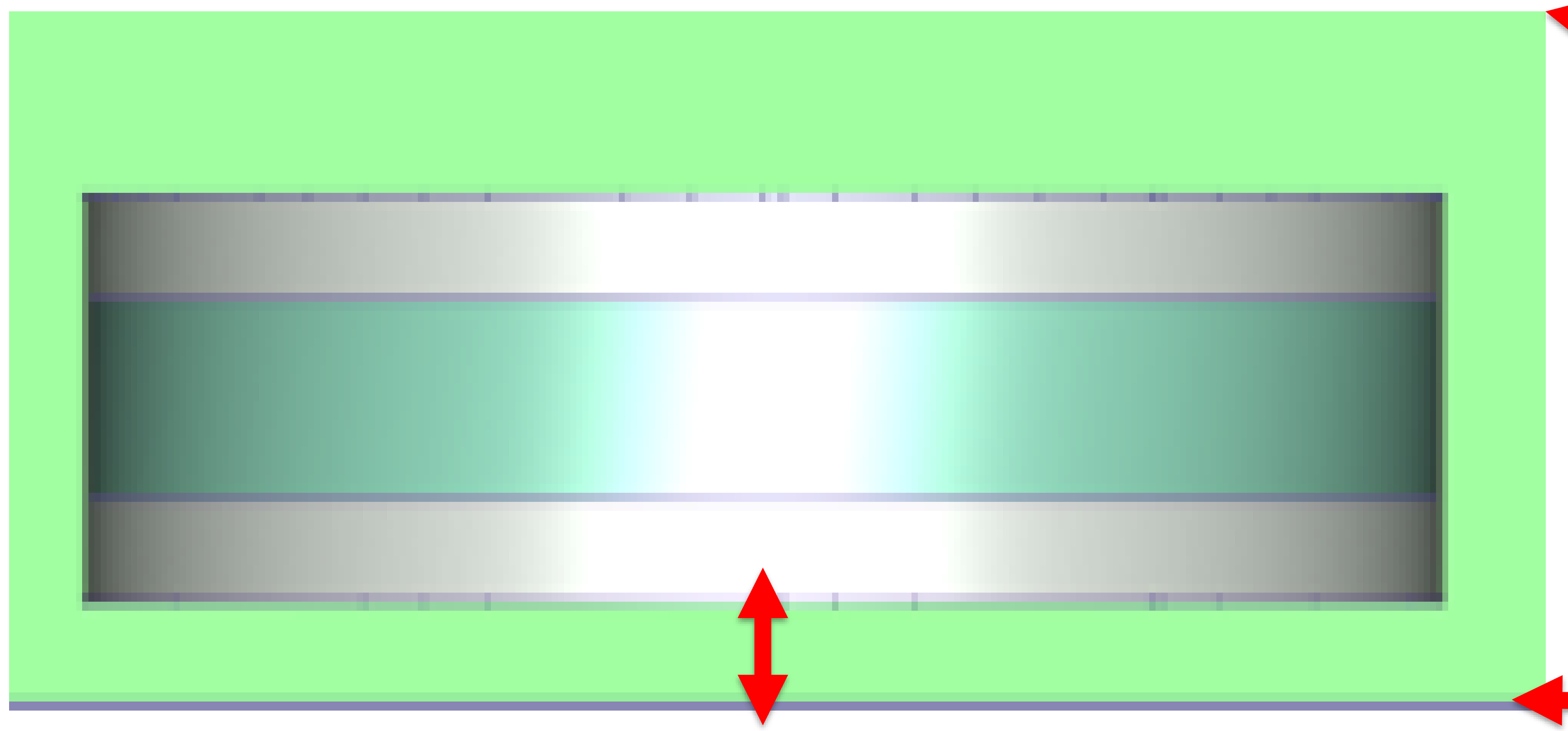




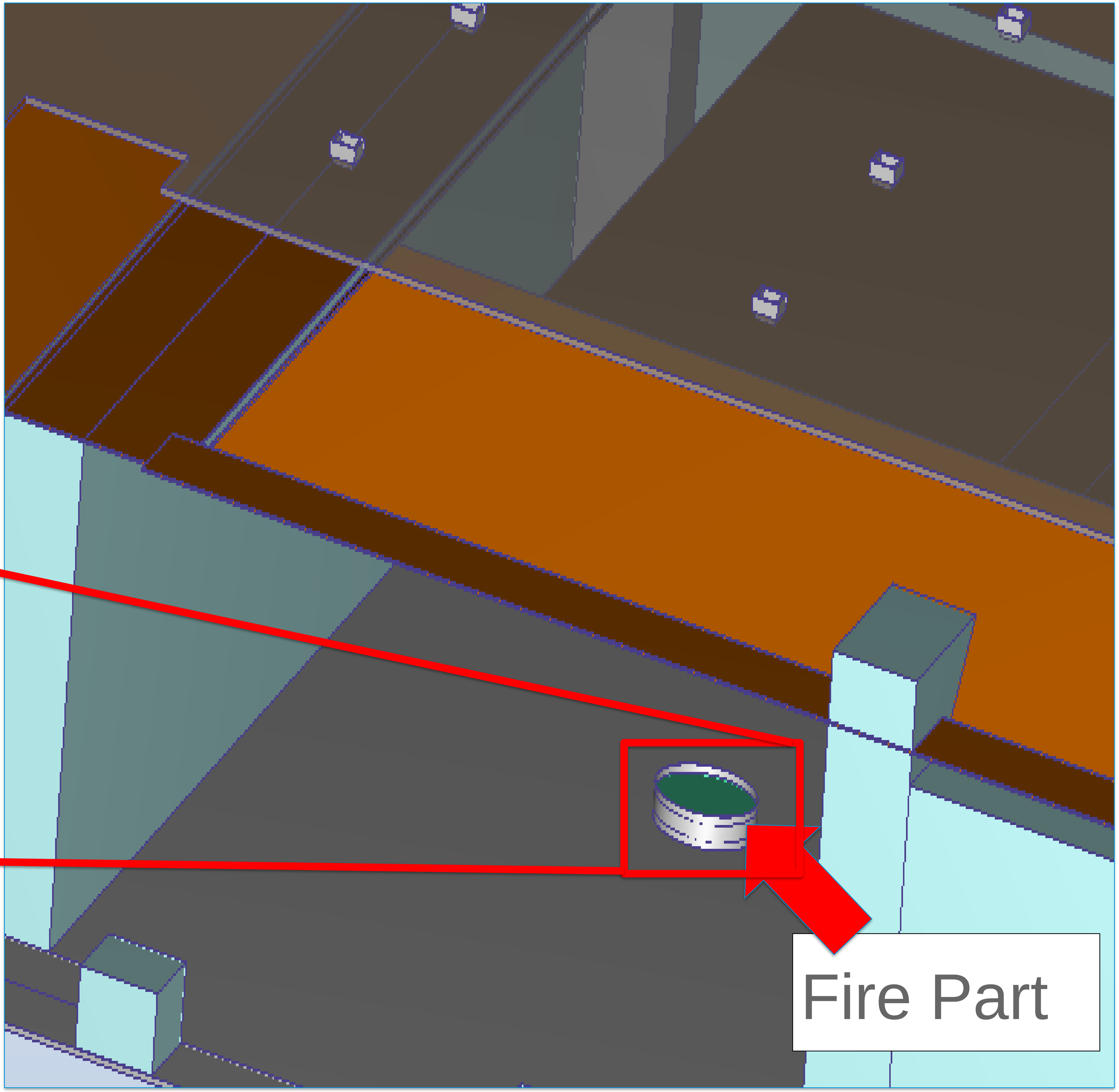
- All air volumes set to 'variable'

Fire Part material

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



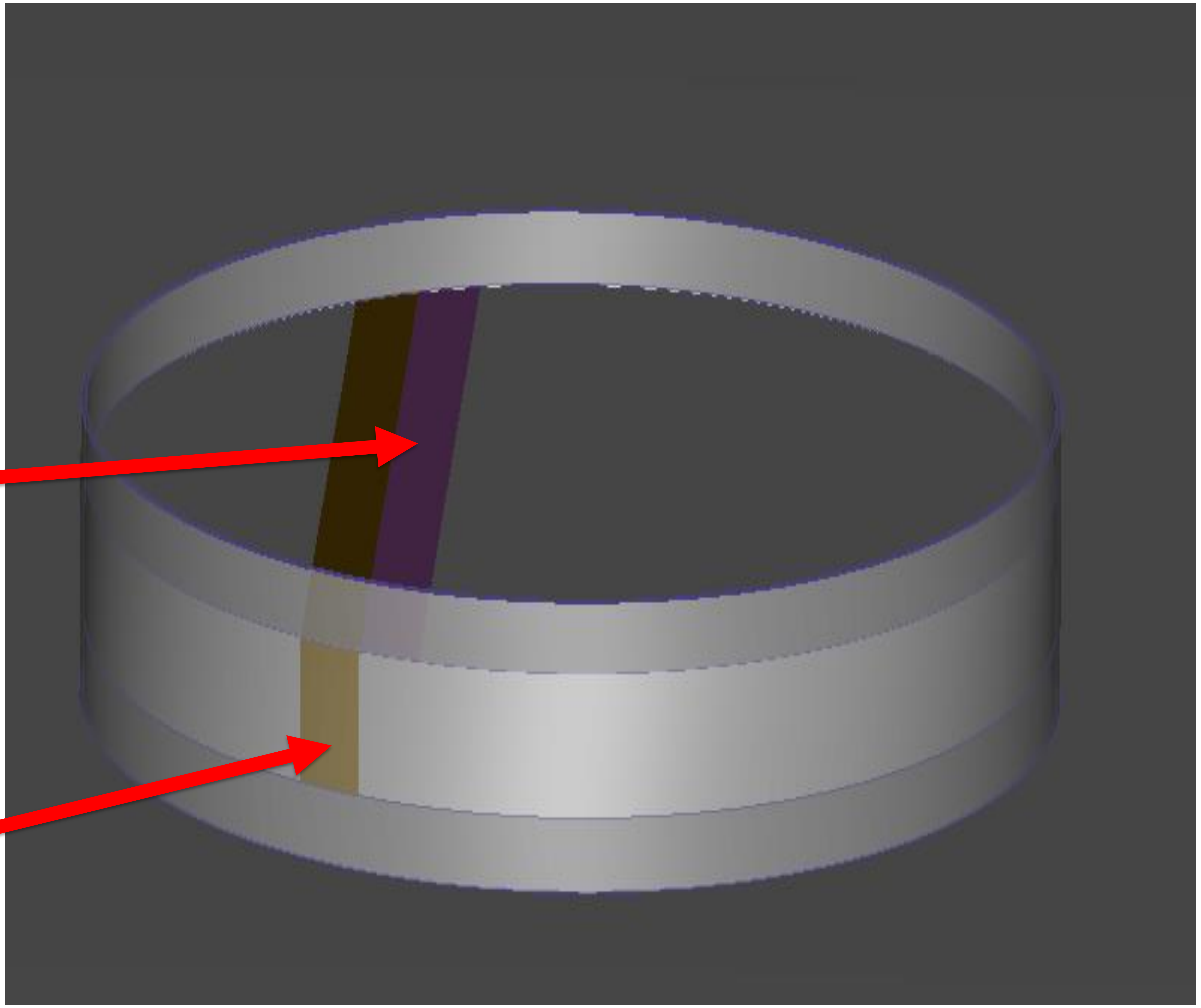
gap underneath the fire for cool air to be drawn in

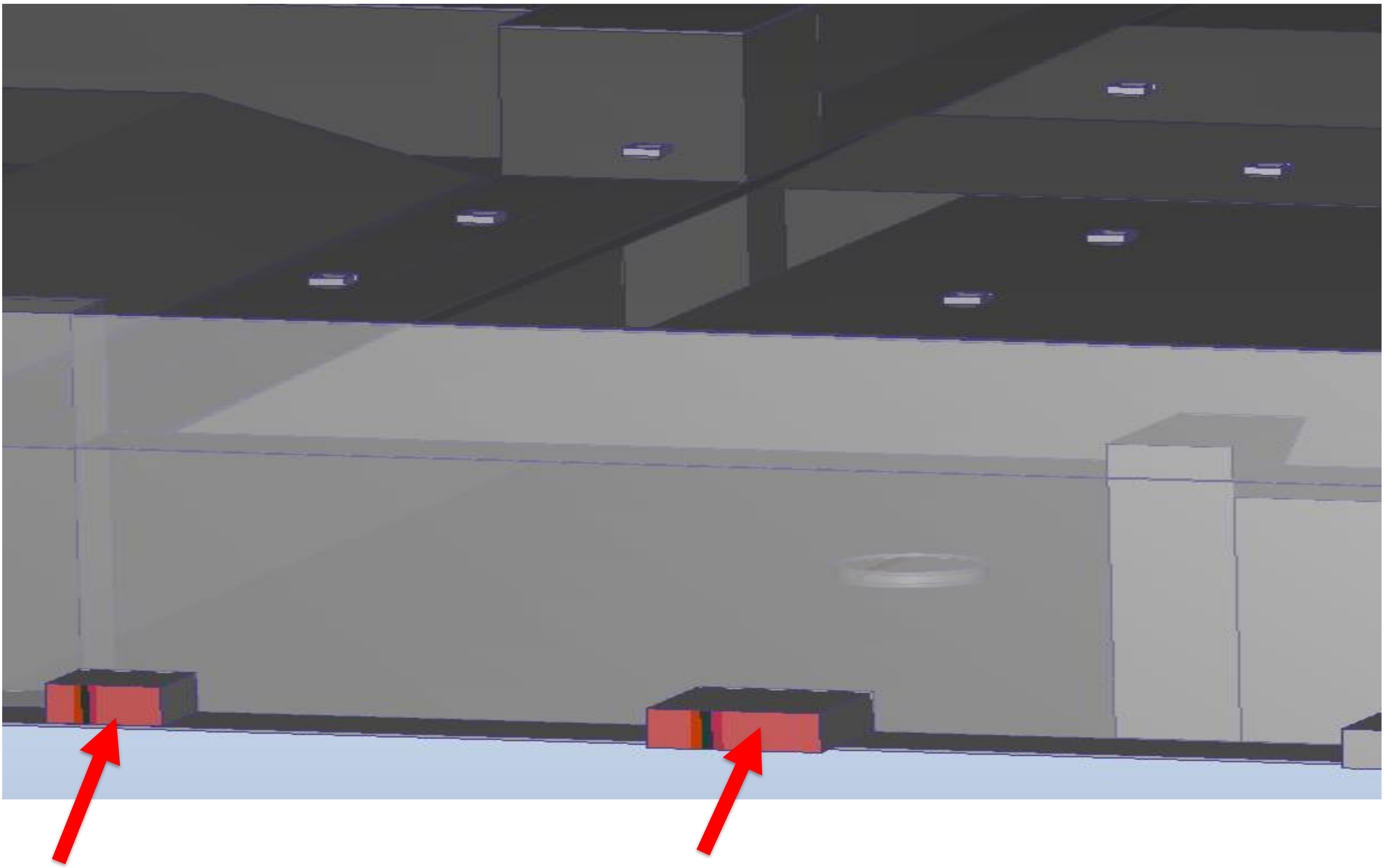


Boundary Conditions

Heat source and scalar BCs
Fire part

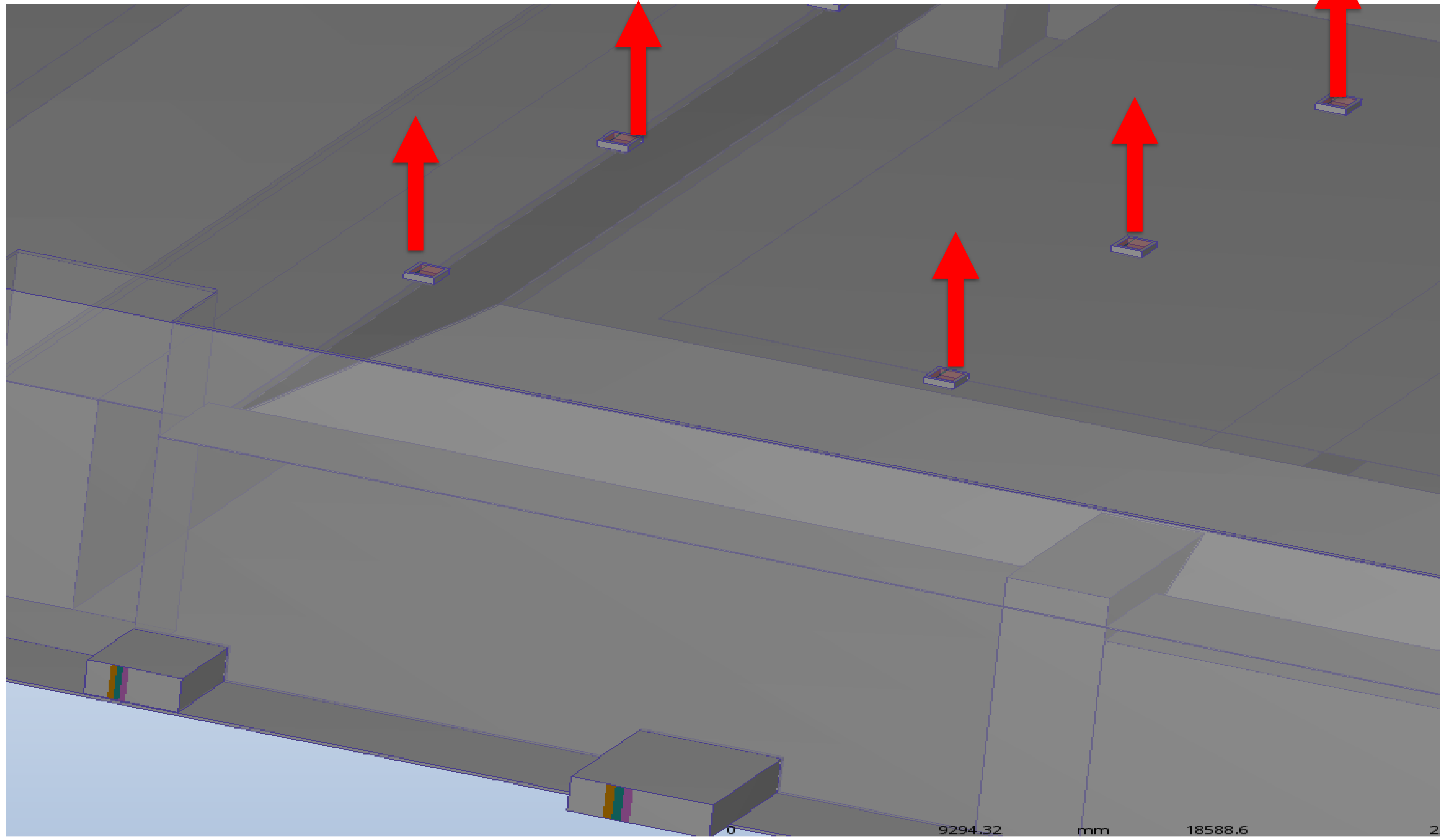
- Scalar of 1 applied to it's top surface. Scalar 1 represents 100% smoke
- Total heat generation to it's entire volume,





Inlet BC

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



Outlet BC

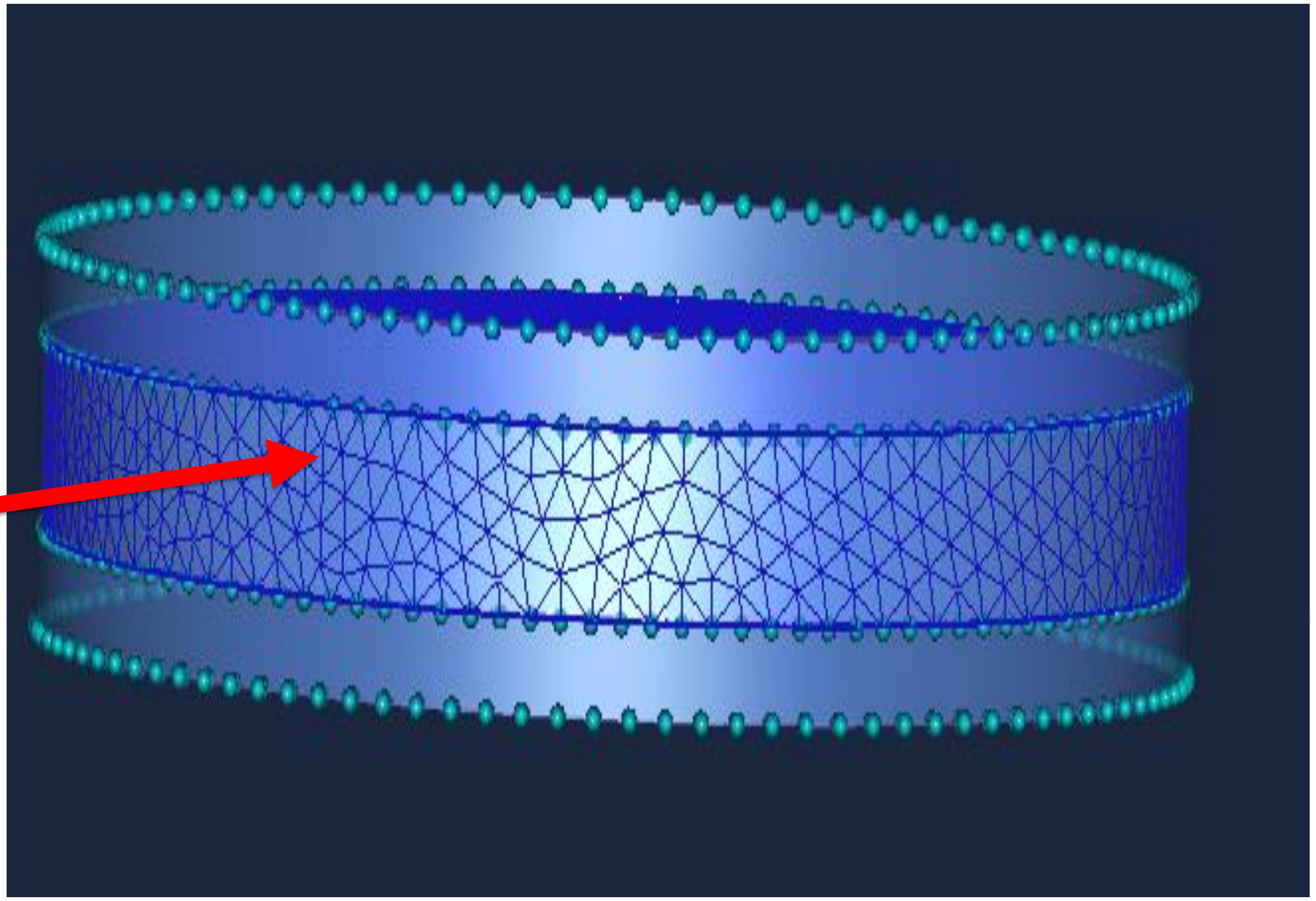
- Volume Flow Rate (ft3/min)

Meshing

Meshing of fire part

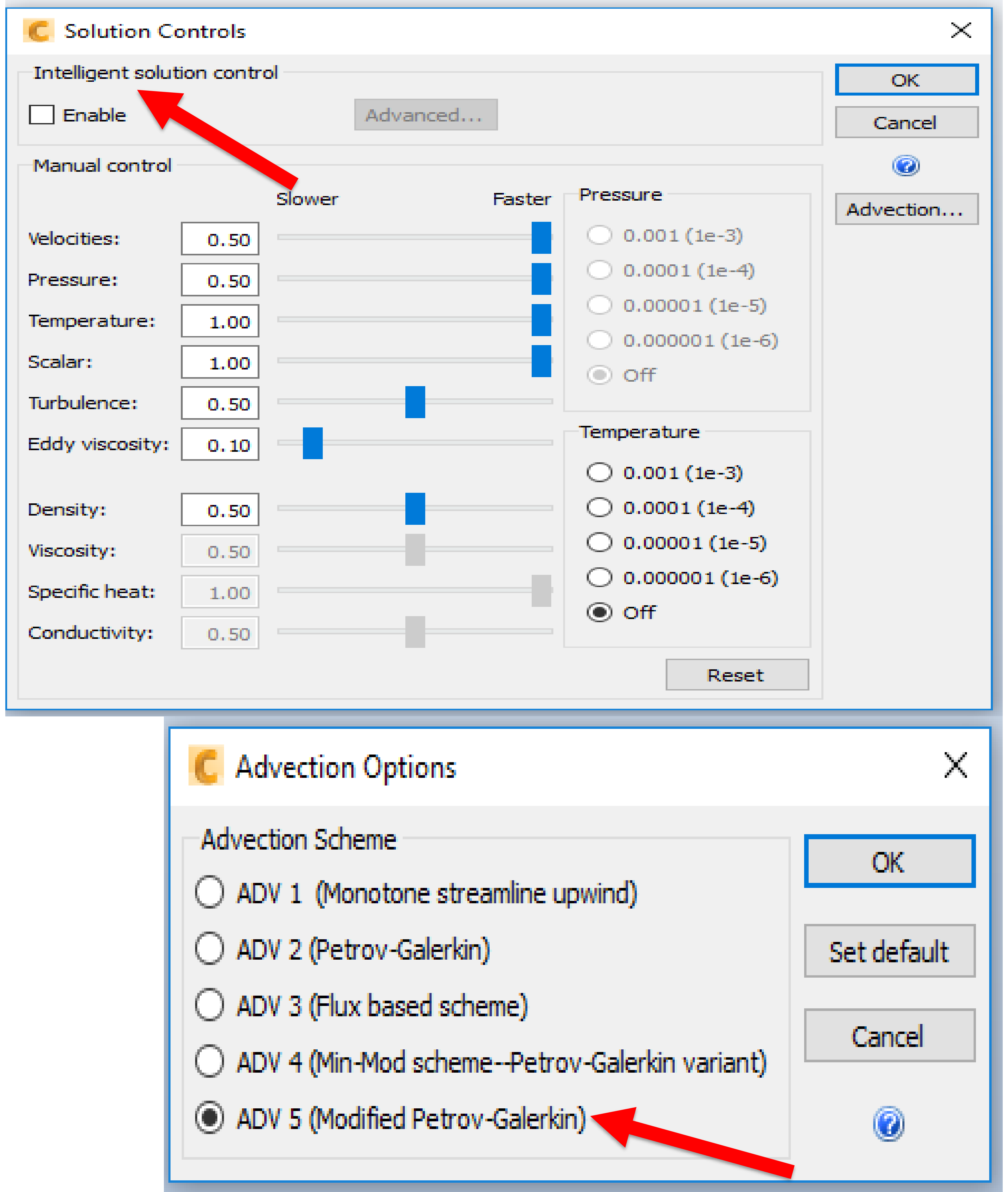
To capture smoke flow and heat release from fire source

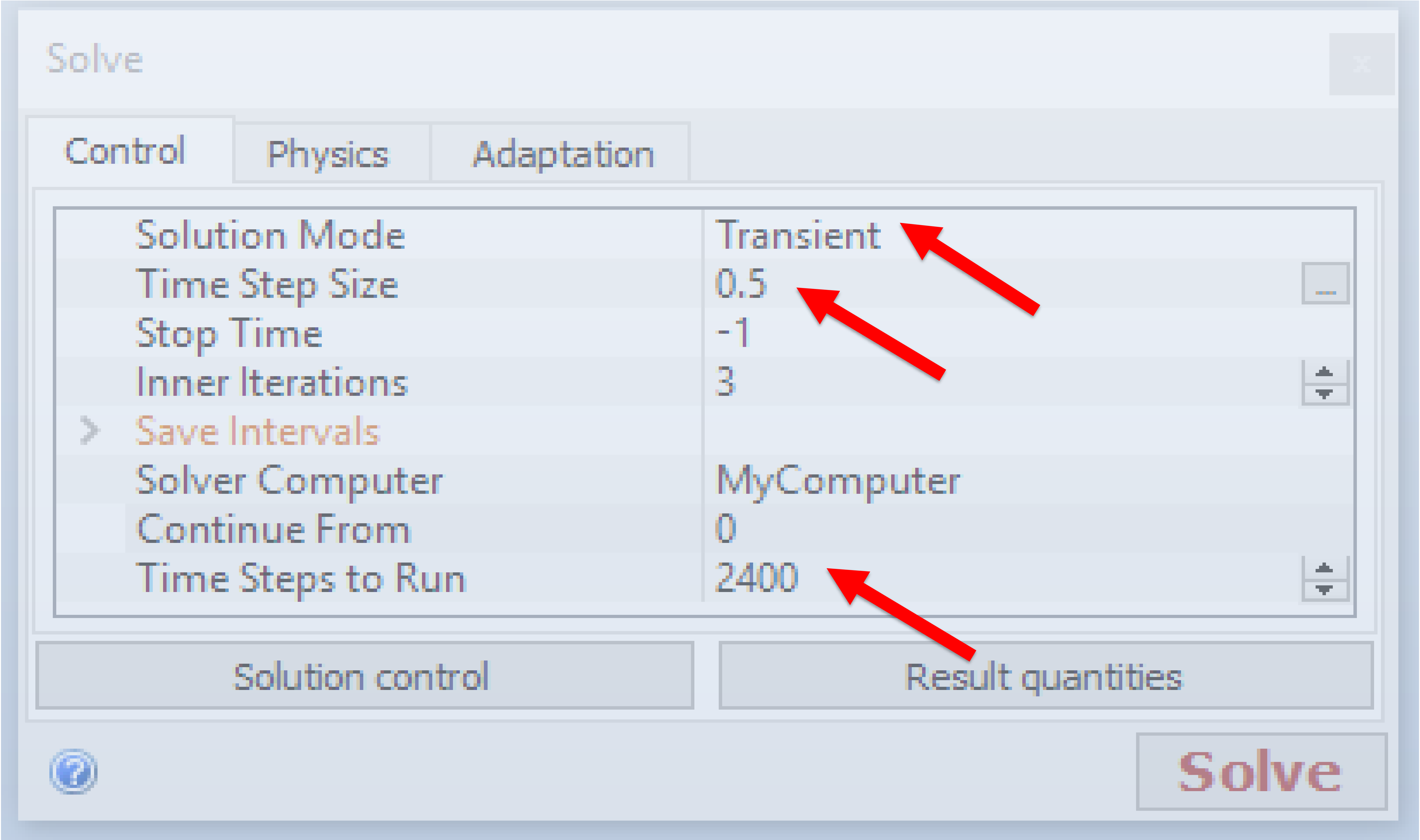
- Fire part - uniform mesh 4-5 elements
- Air above and below - a good uniform mesh



Solver Settings

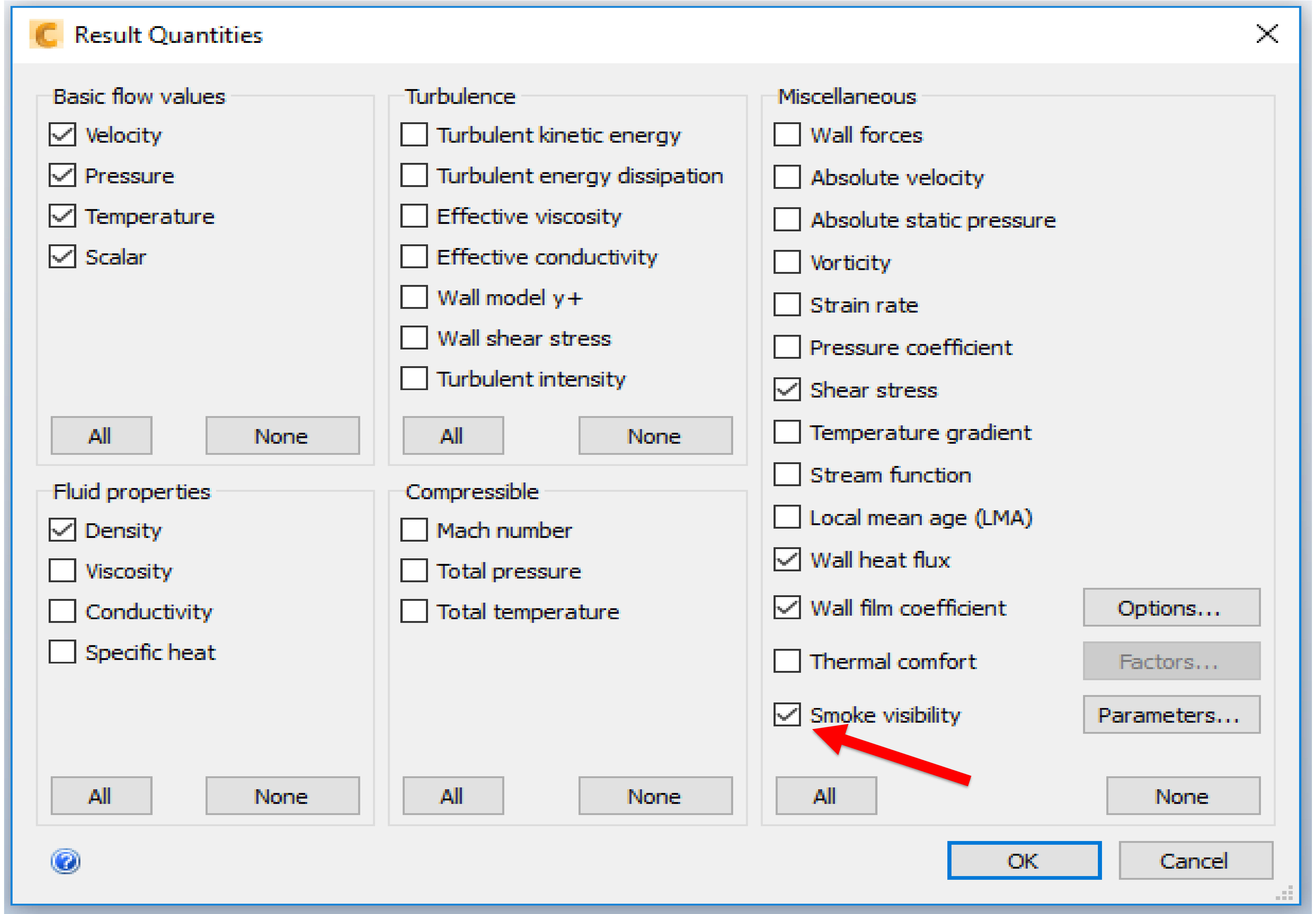
Solver settings for smoke analysis (Intelligent control-off, Advection 5)





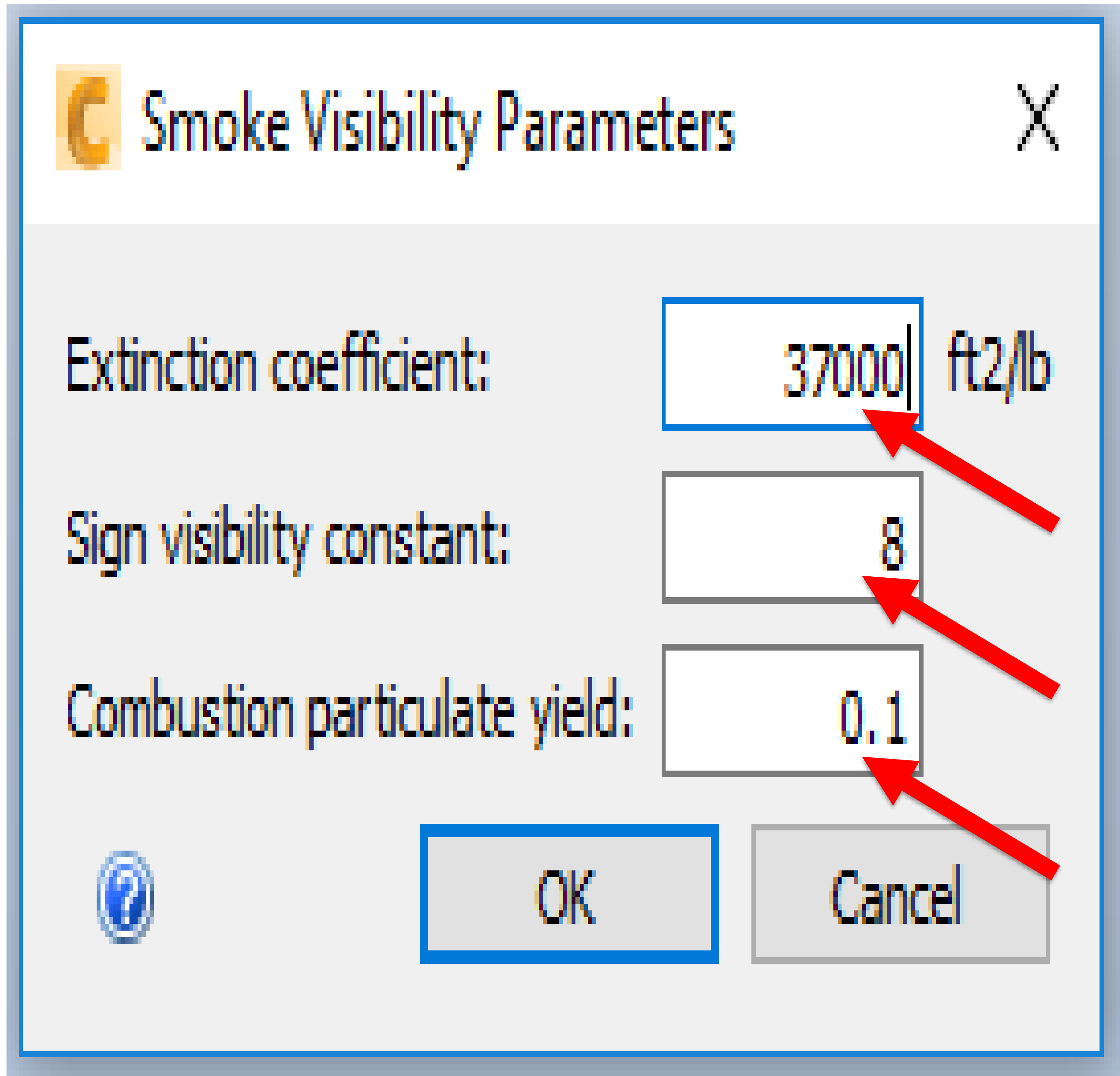
Transient analysis with time step size specified

Solver settings for smoke analysis (Smoke visibility parameters)

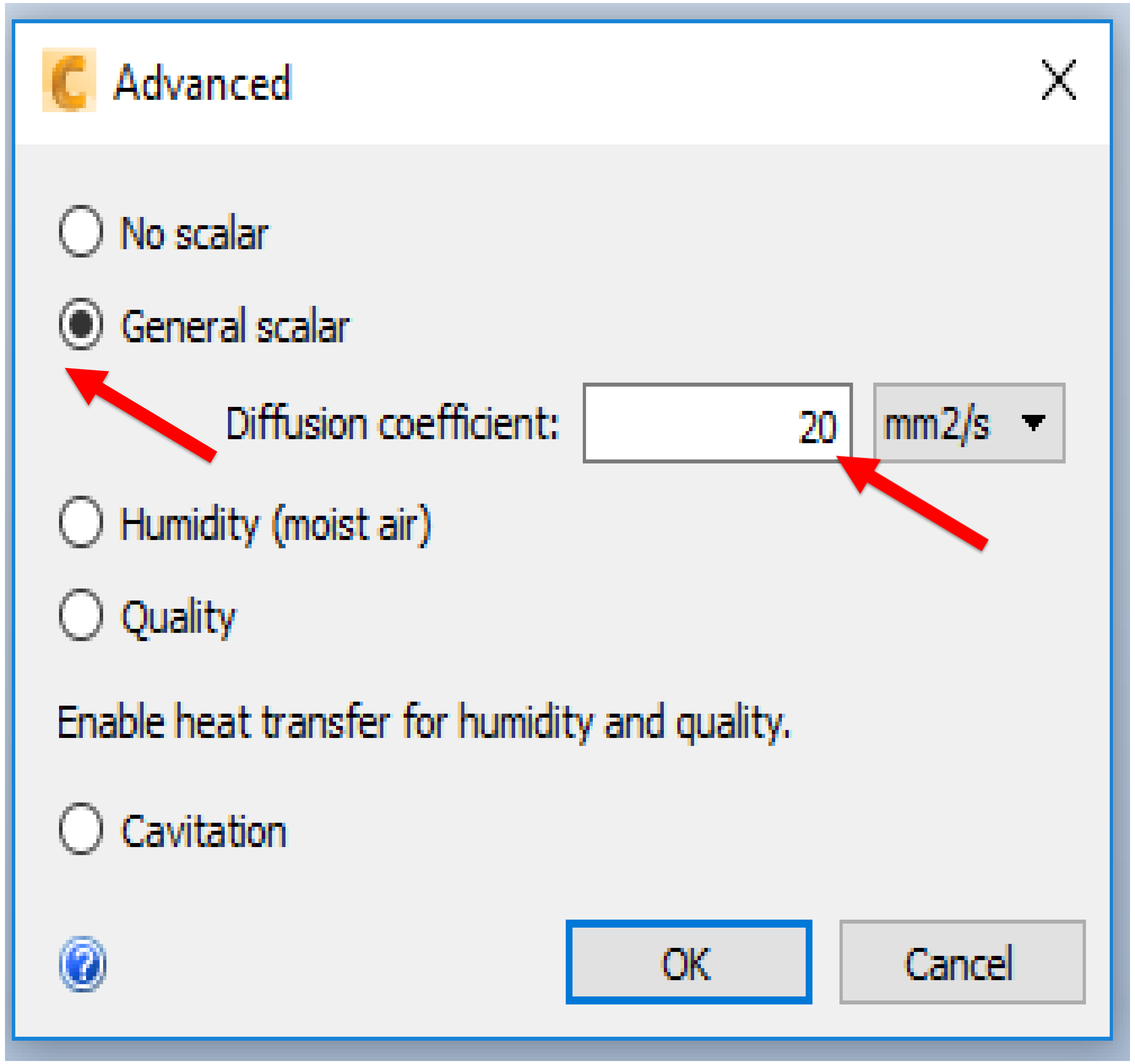


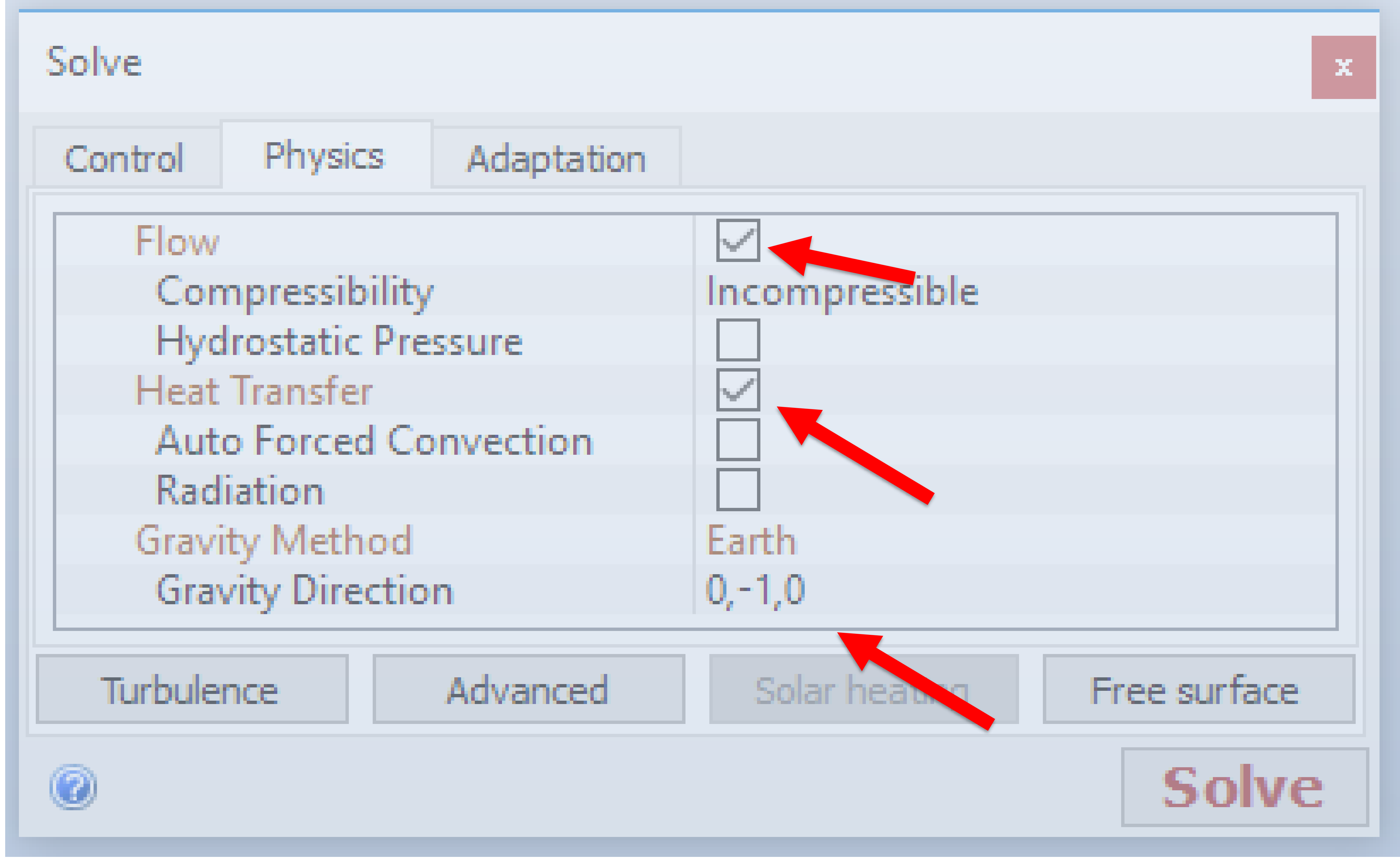
Results Quantities

- Visibility On



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





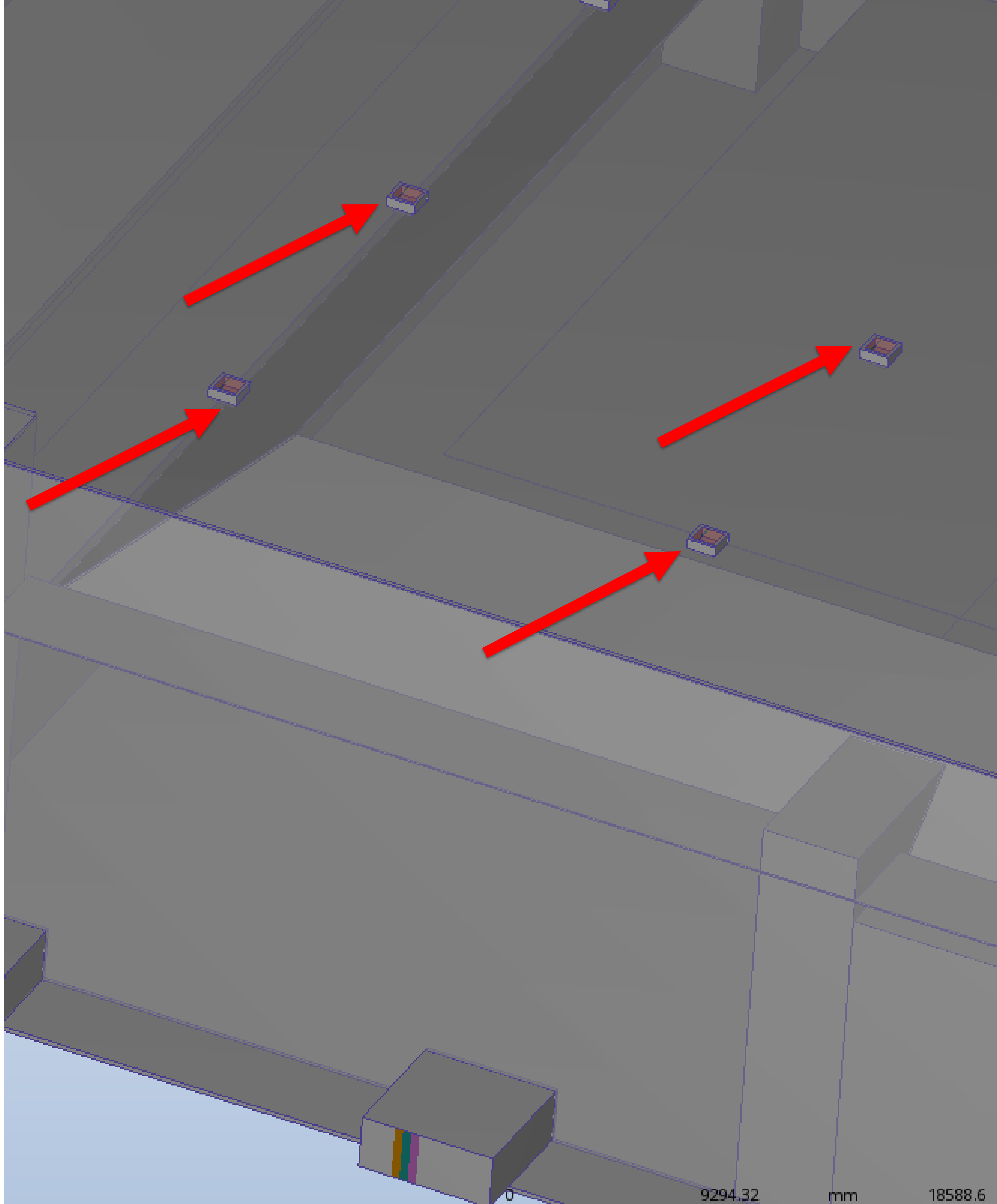
- Run Flow and Thermal together.
- Gravity set in downwards direction
- General Scalar ON .
- Non-zero value for the diffusion coefficient.

Key Results for Design Performance

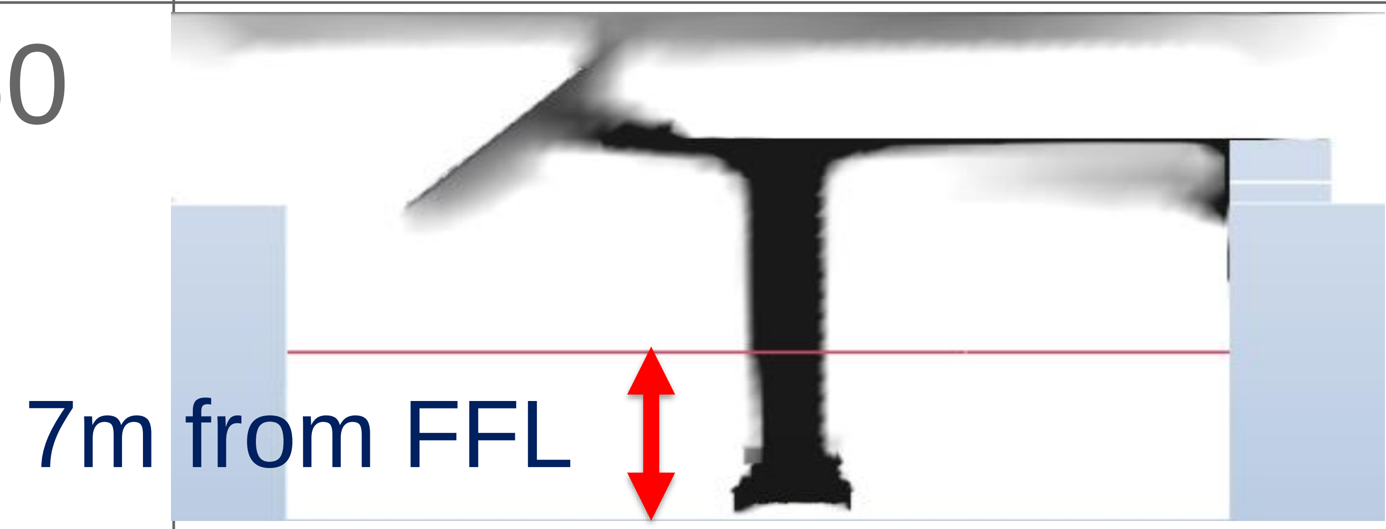
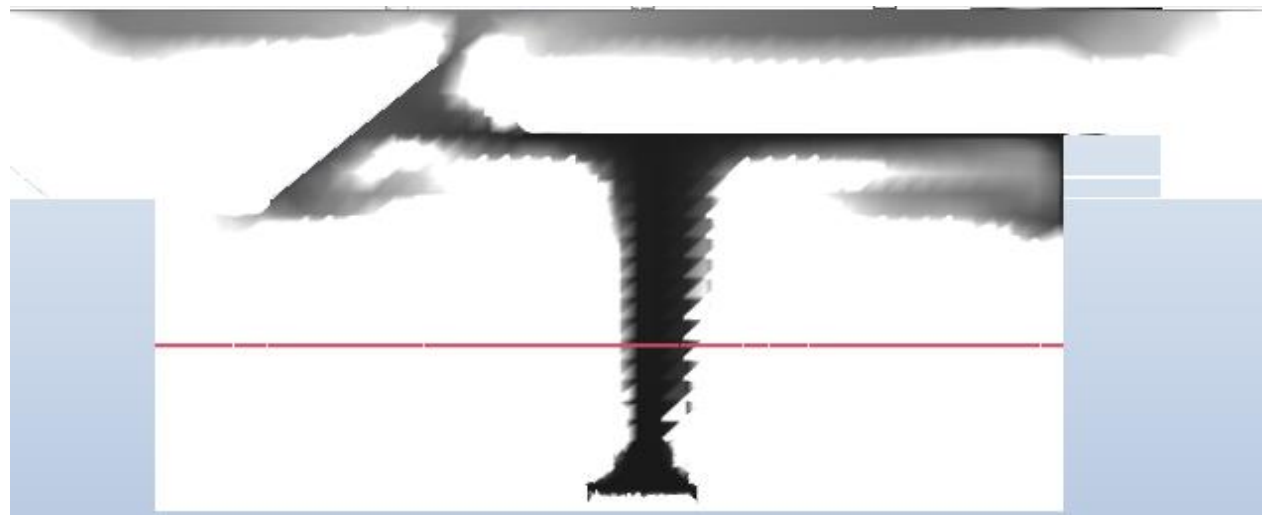
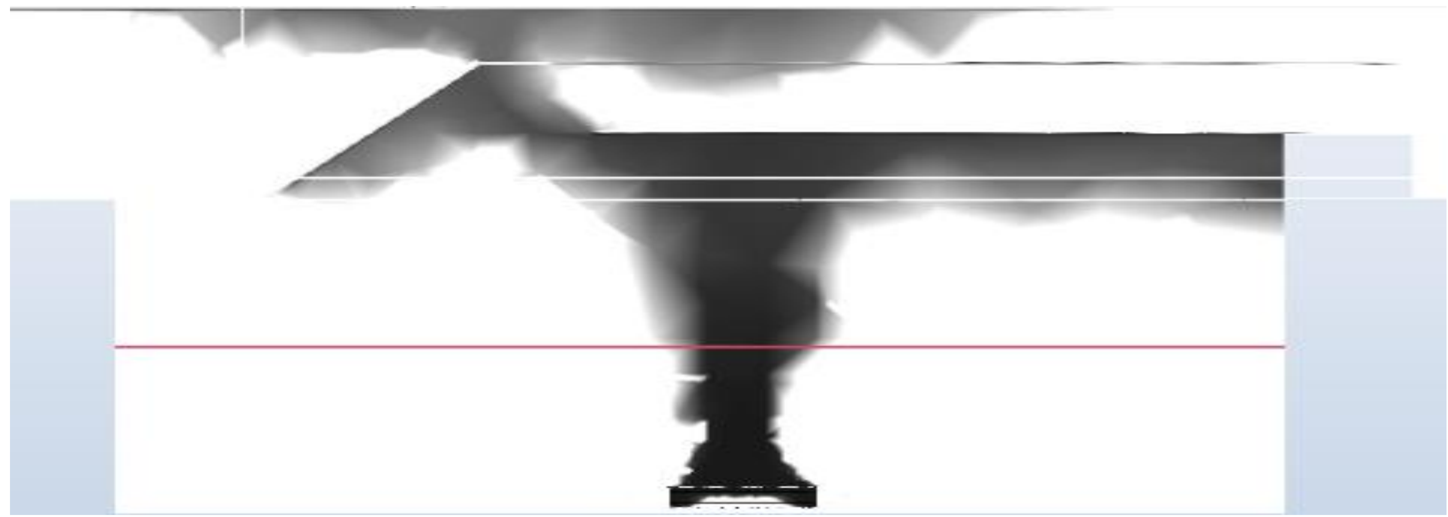
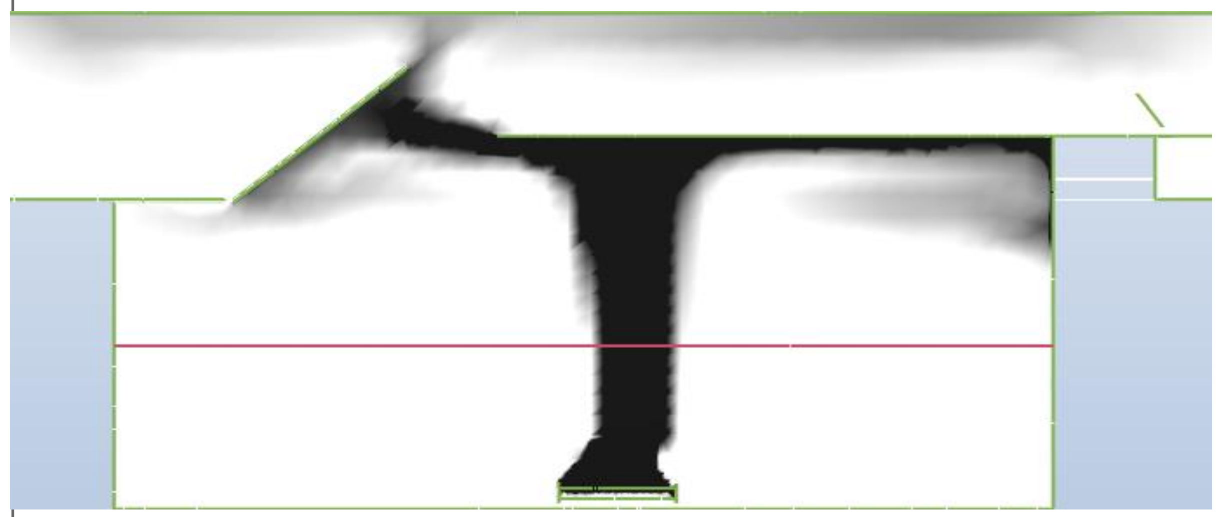
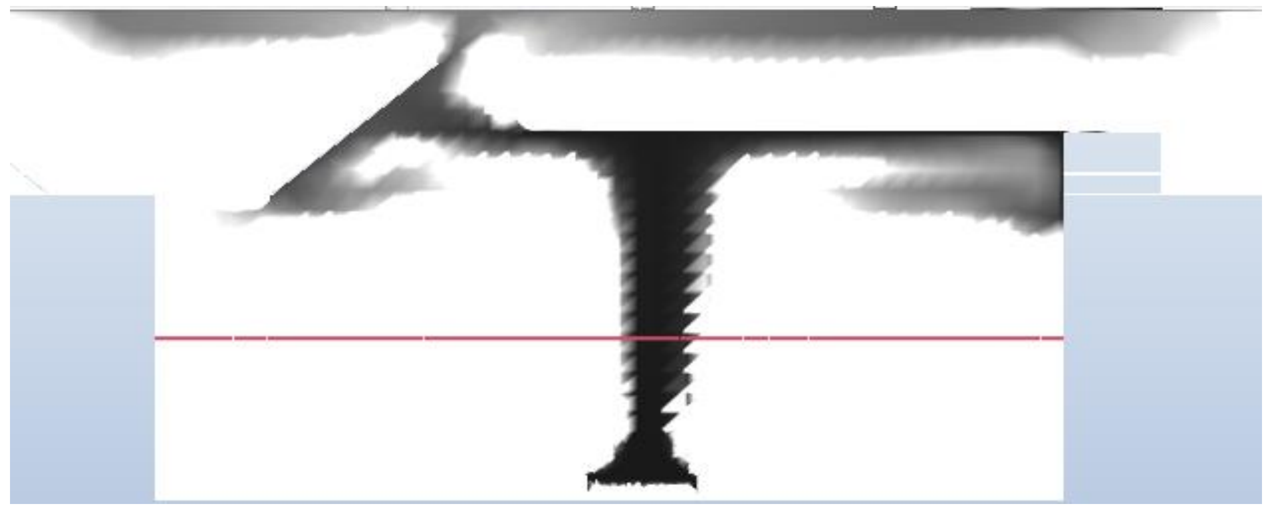
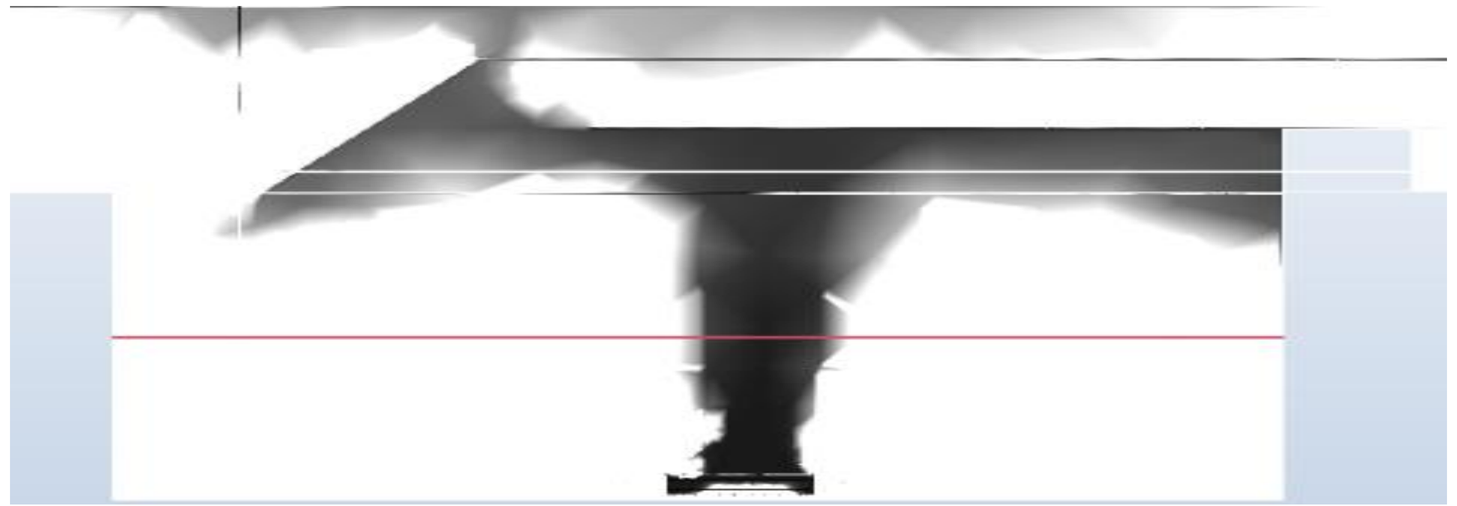
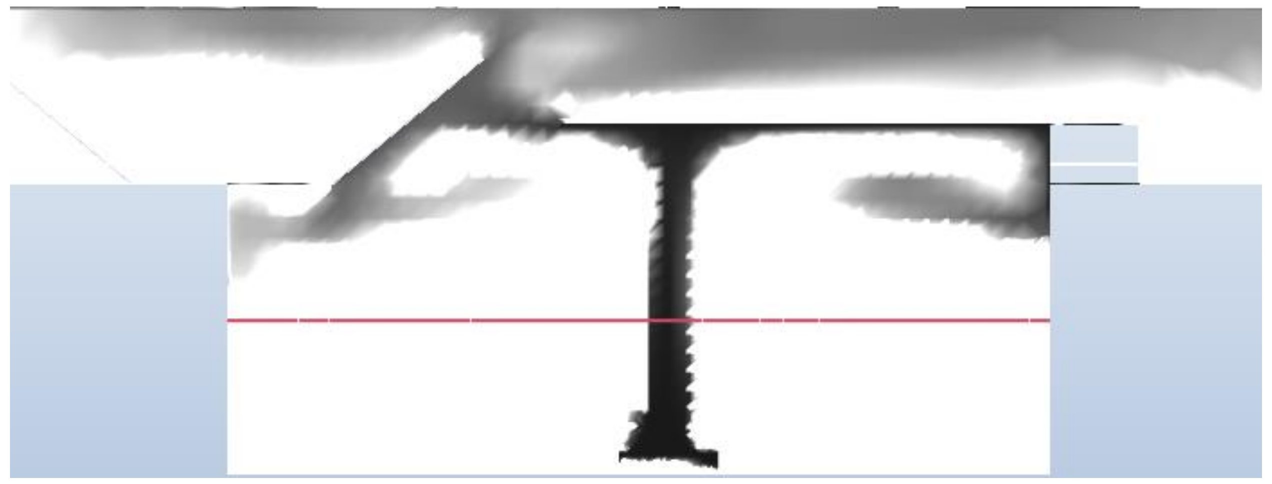
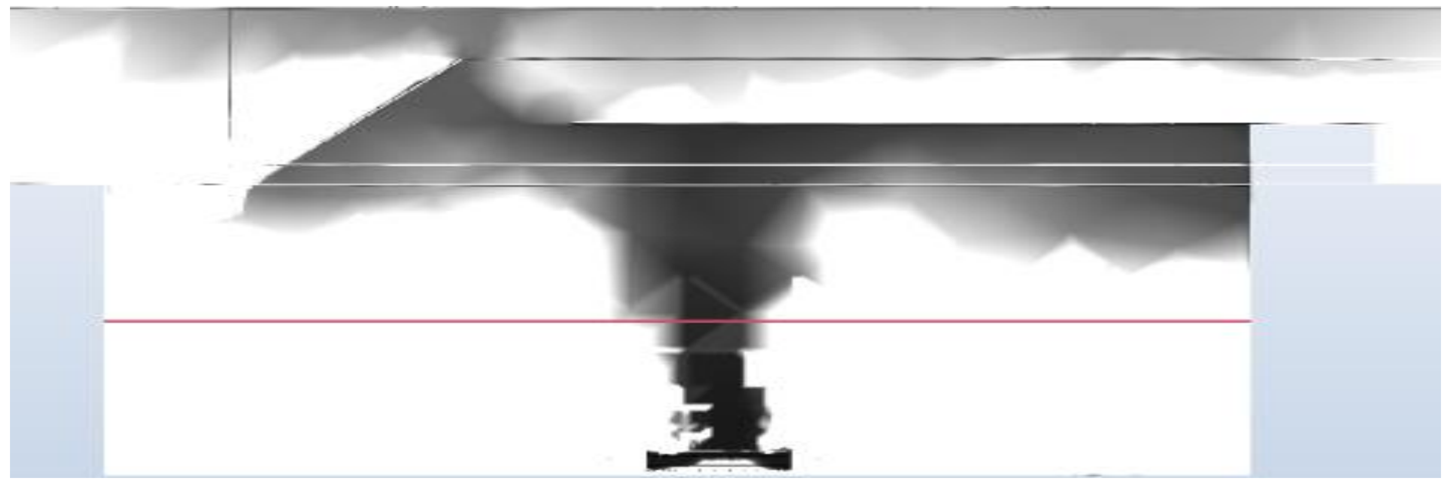
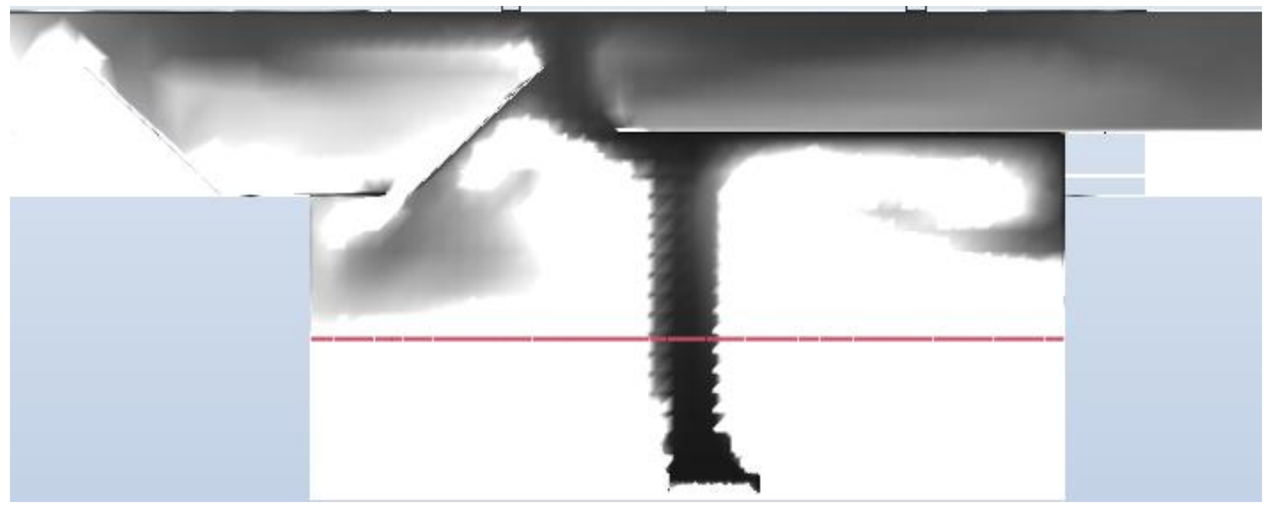
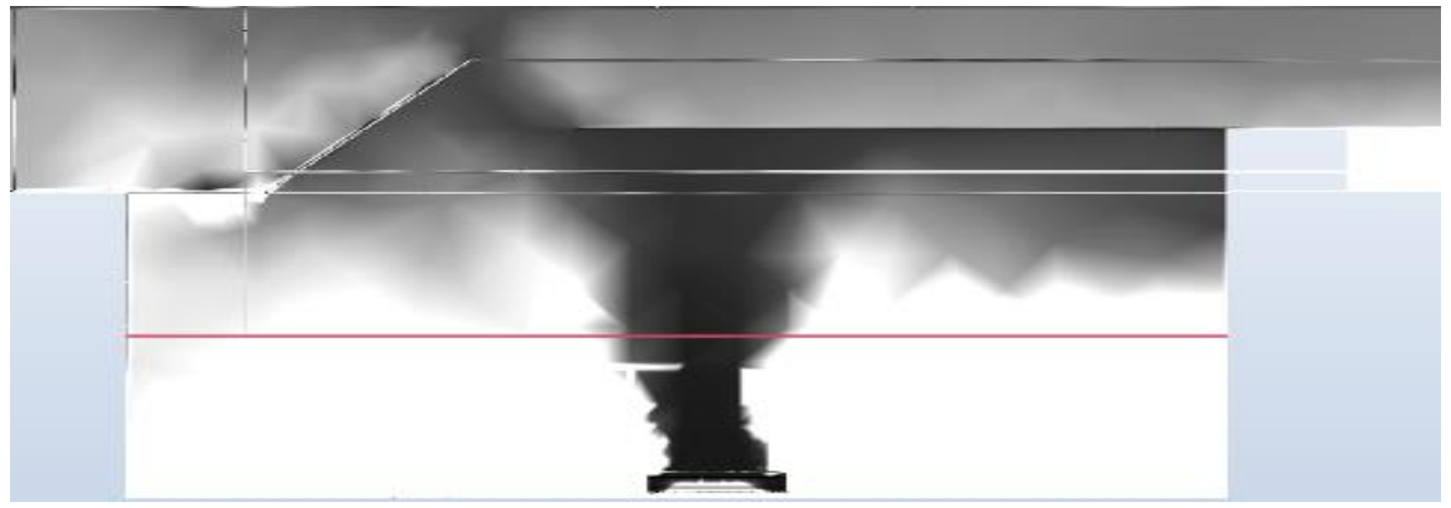
Smoke Simulation
Performance Evaluation

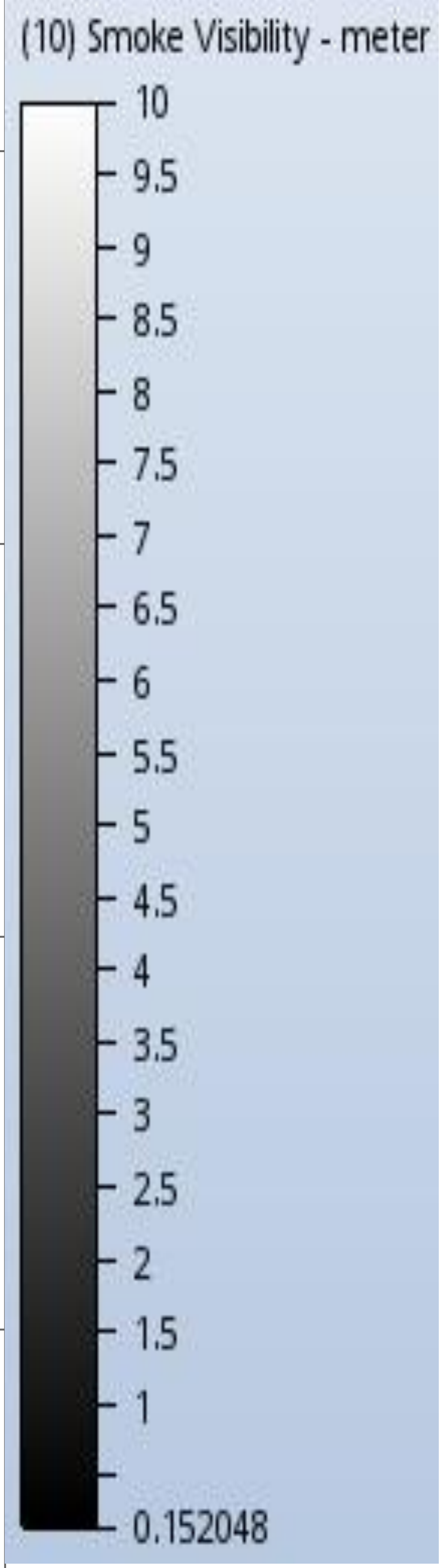
Design Options based on Outlet BC:

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

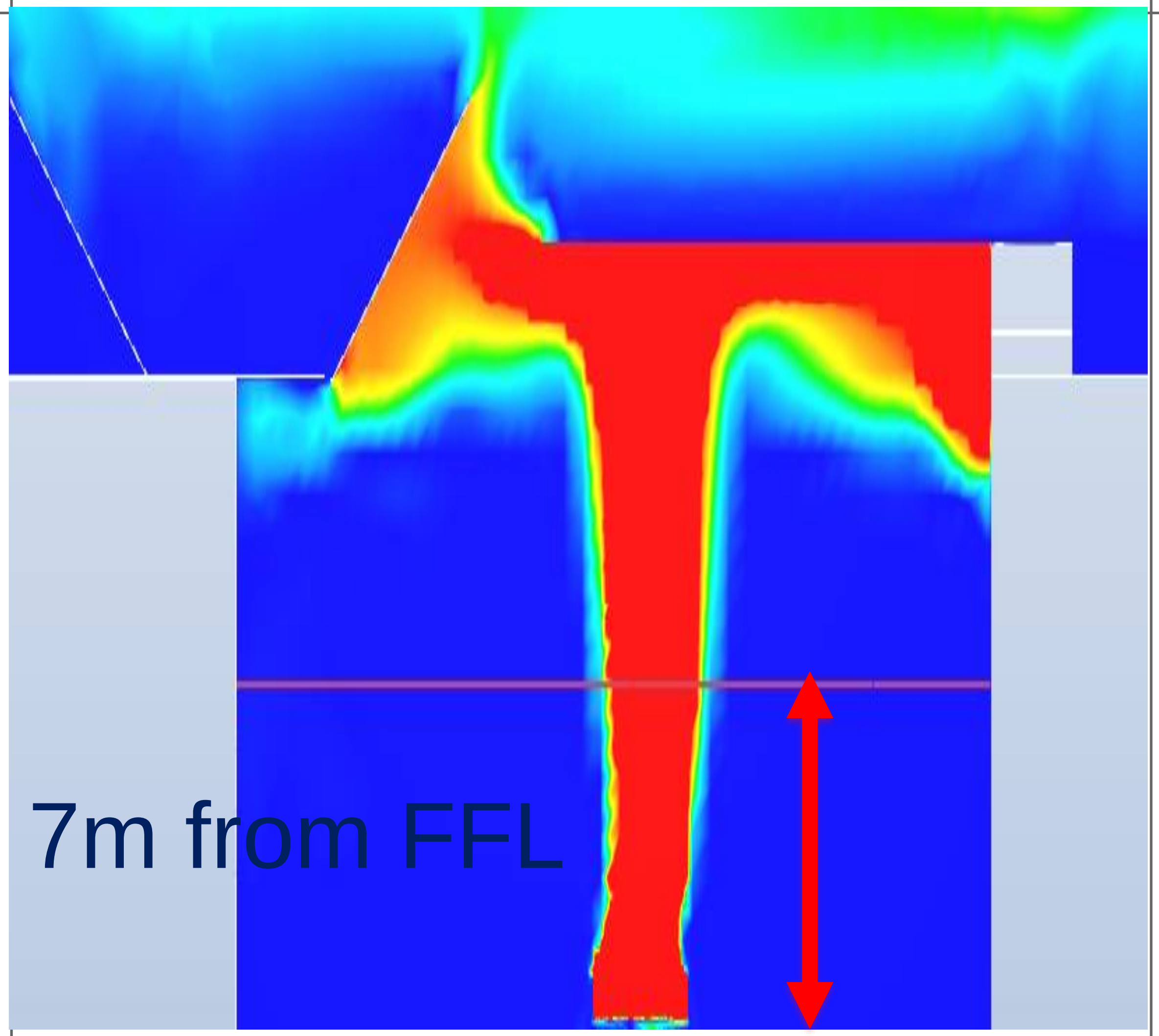
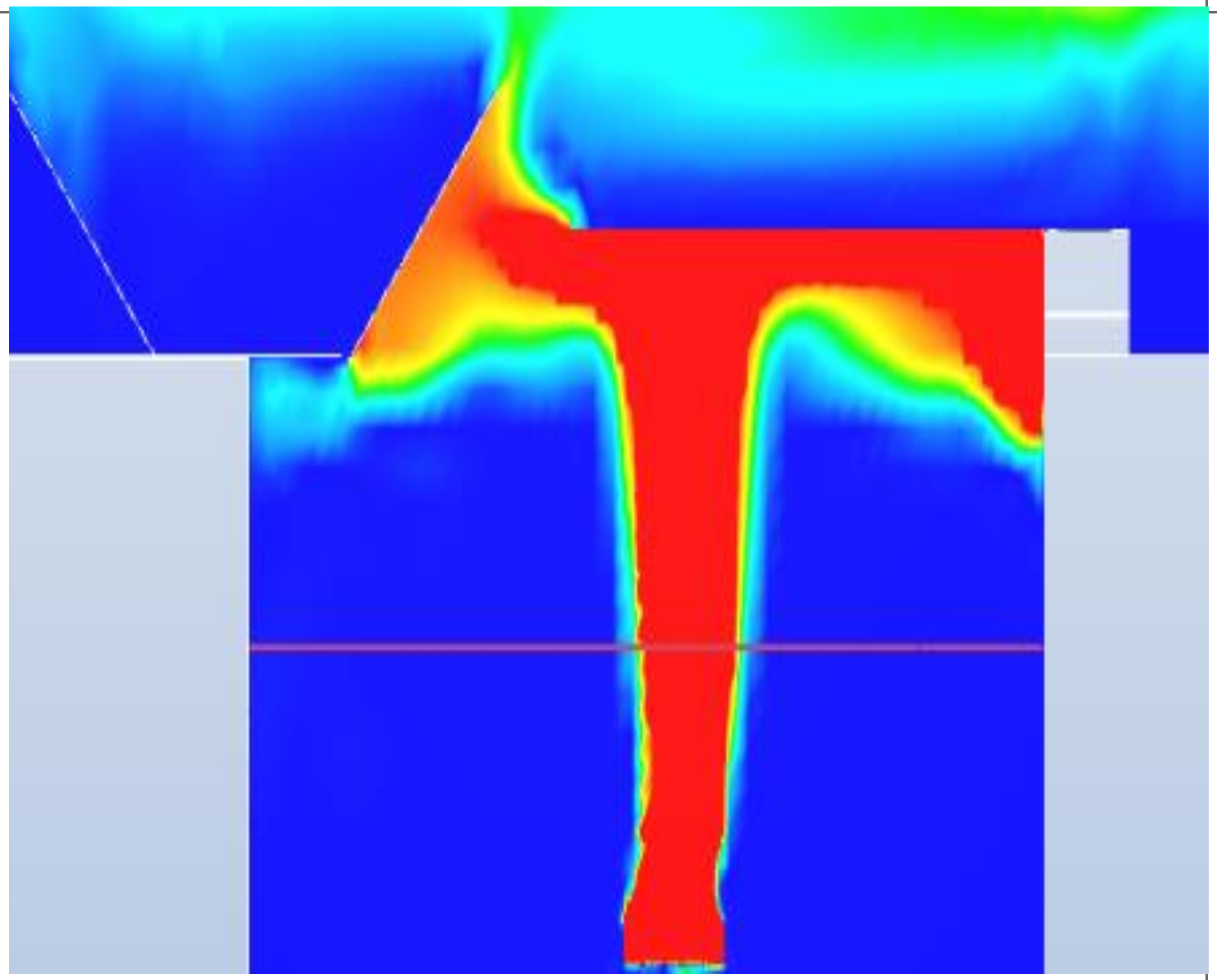
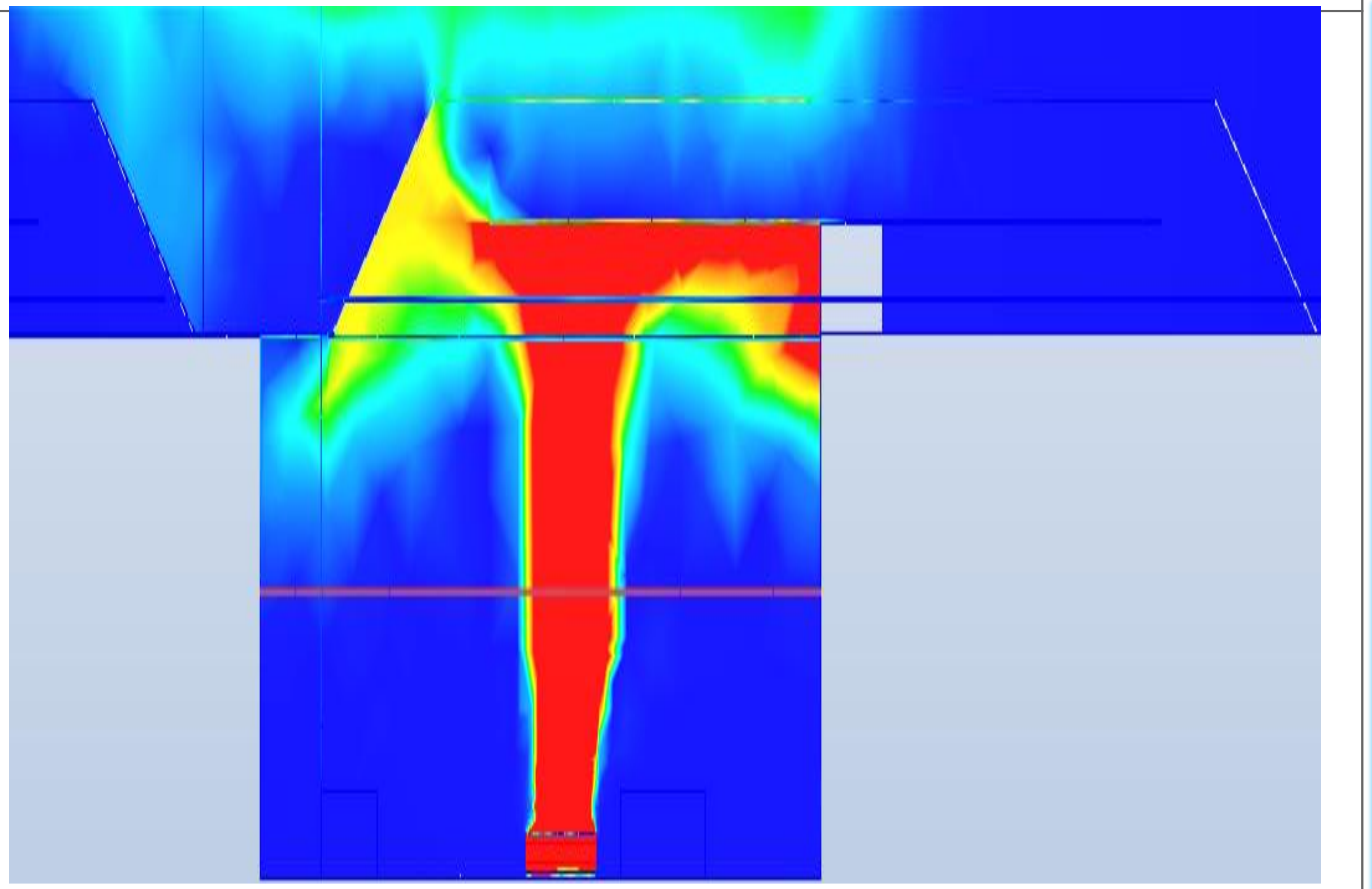
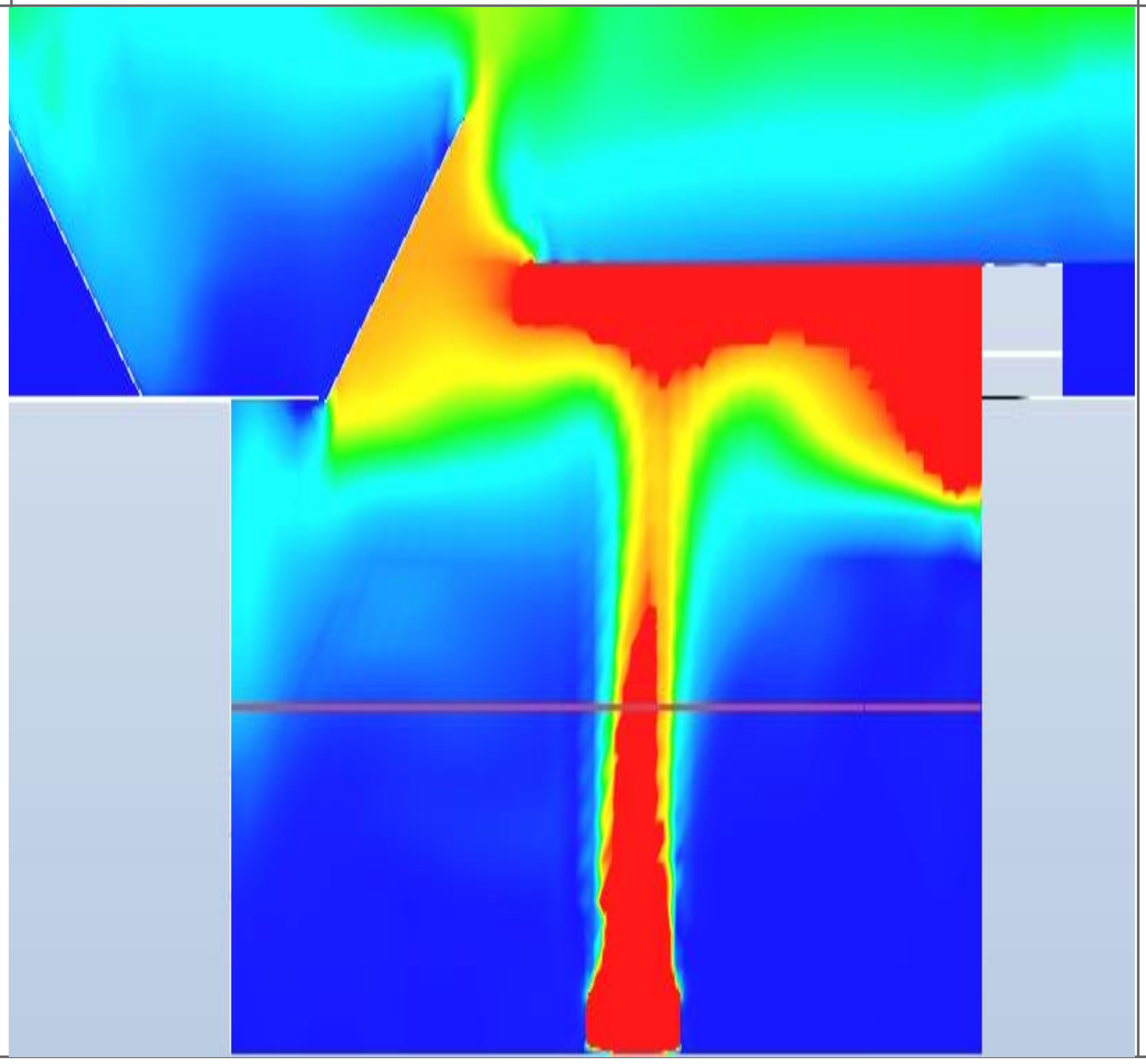
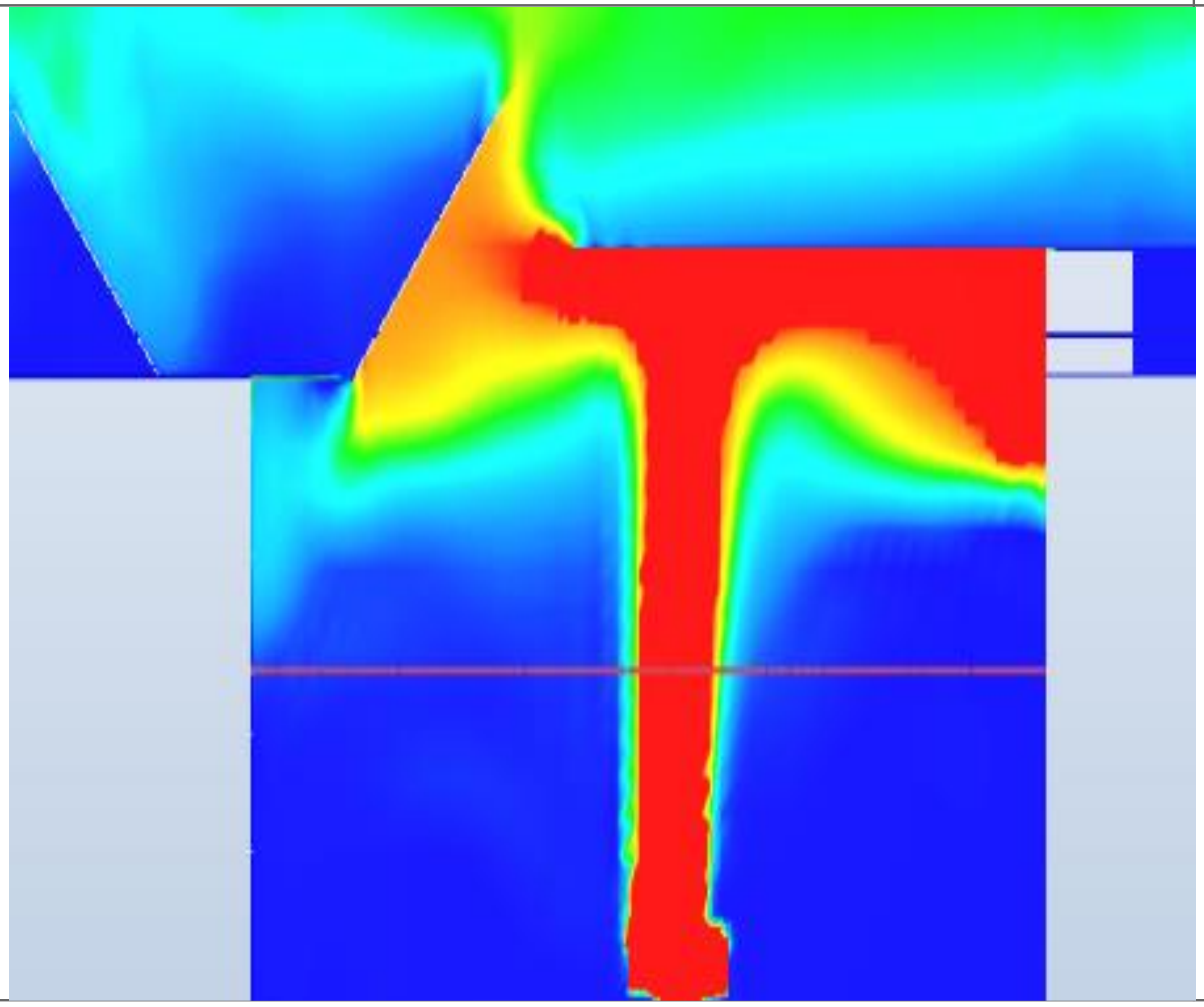
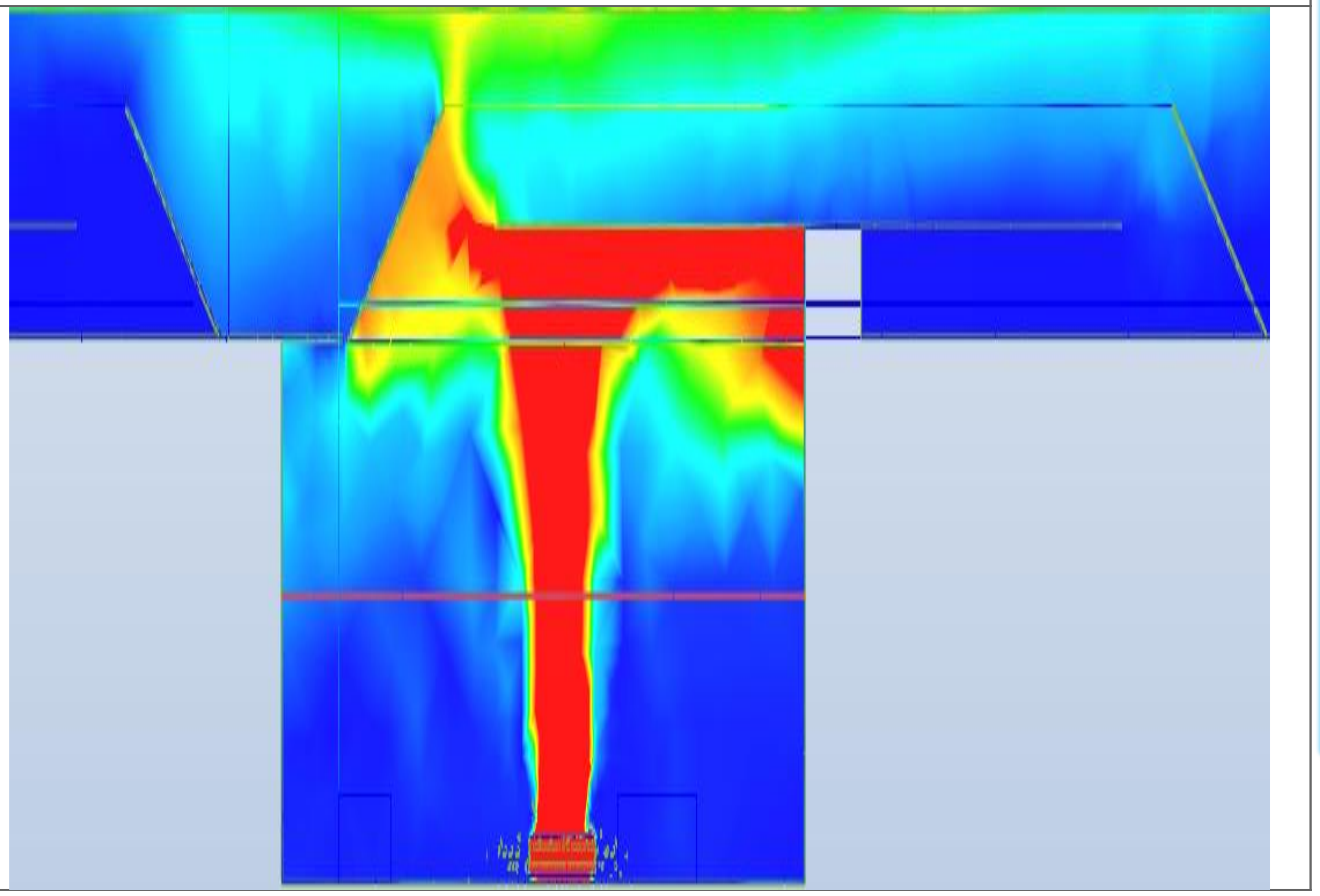


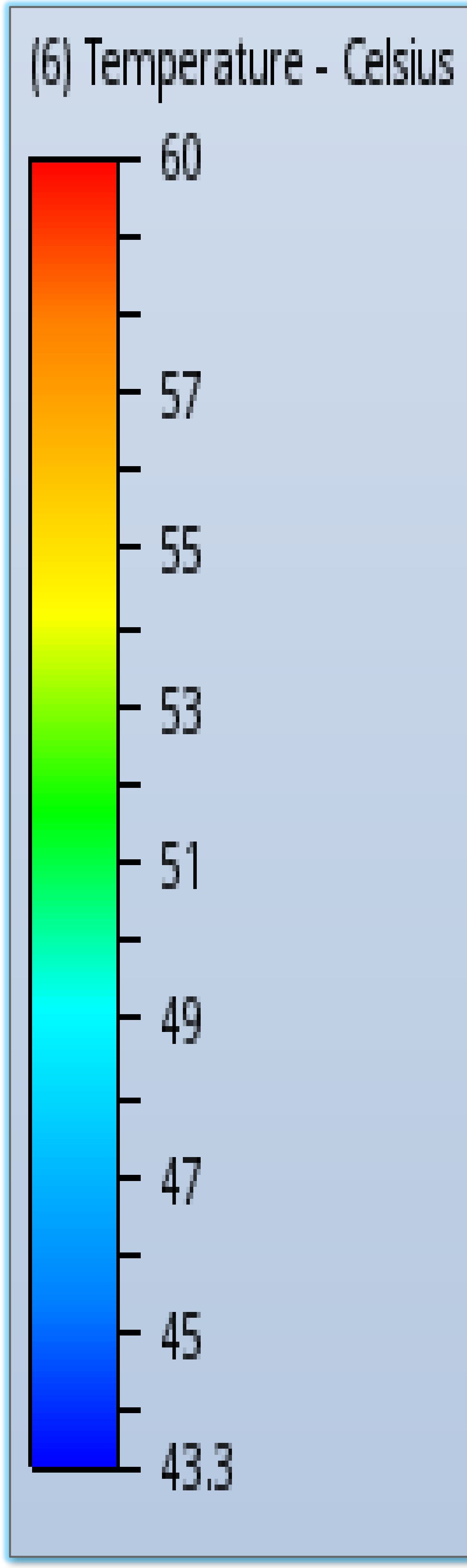
CFD analysis results –Smoke Visibility

Time (s)	Initial Design	Design Option1	Design Option 2
60			
90			
180			
360			

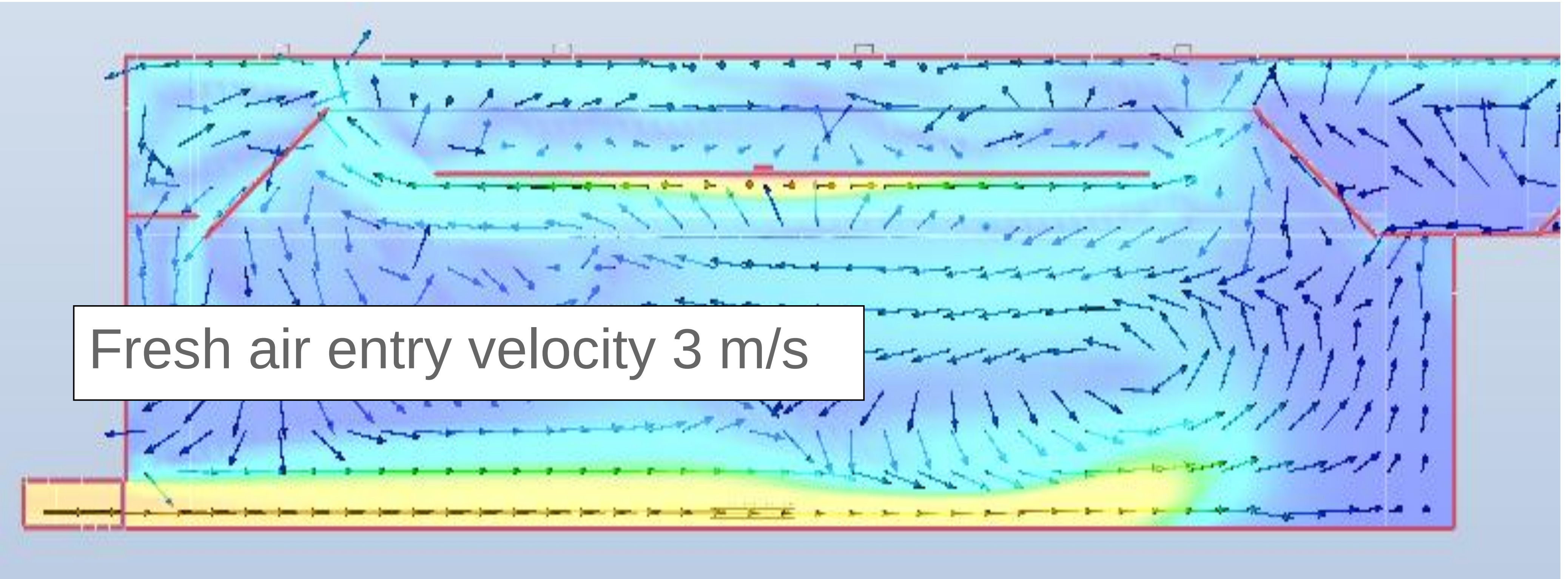
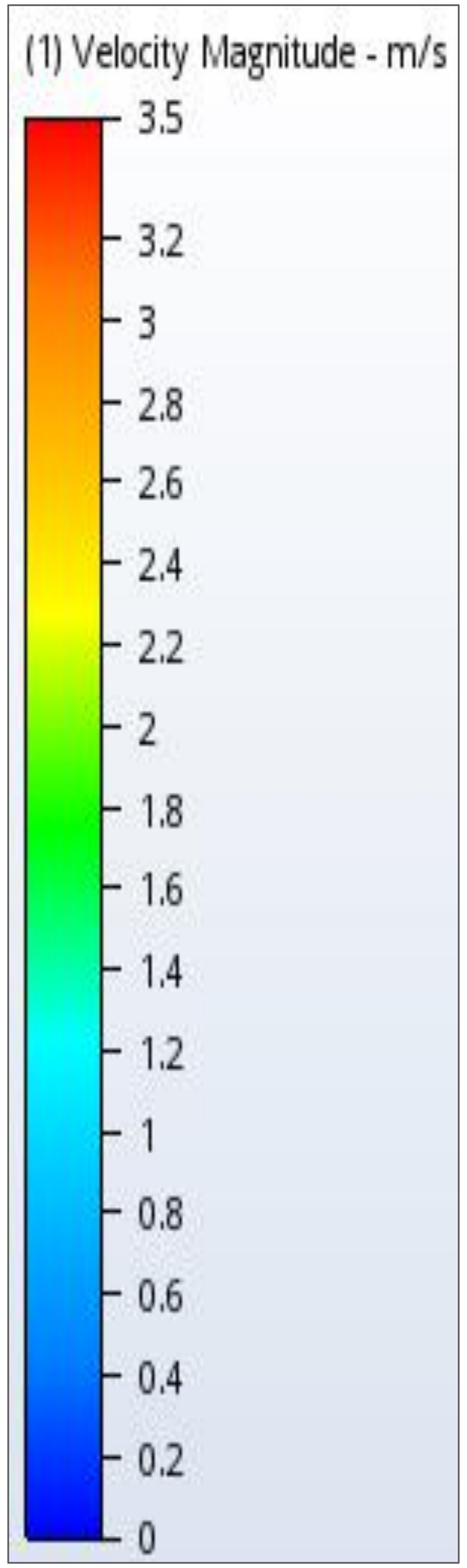
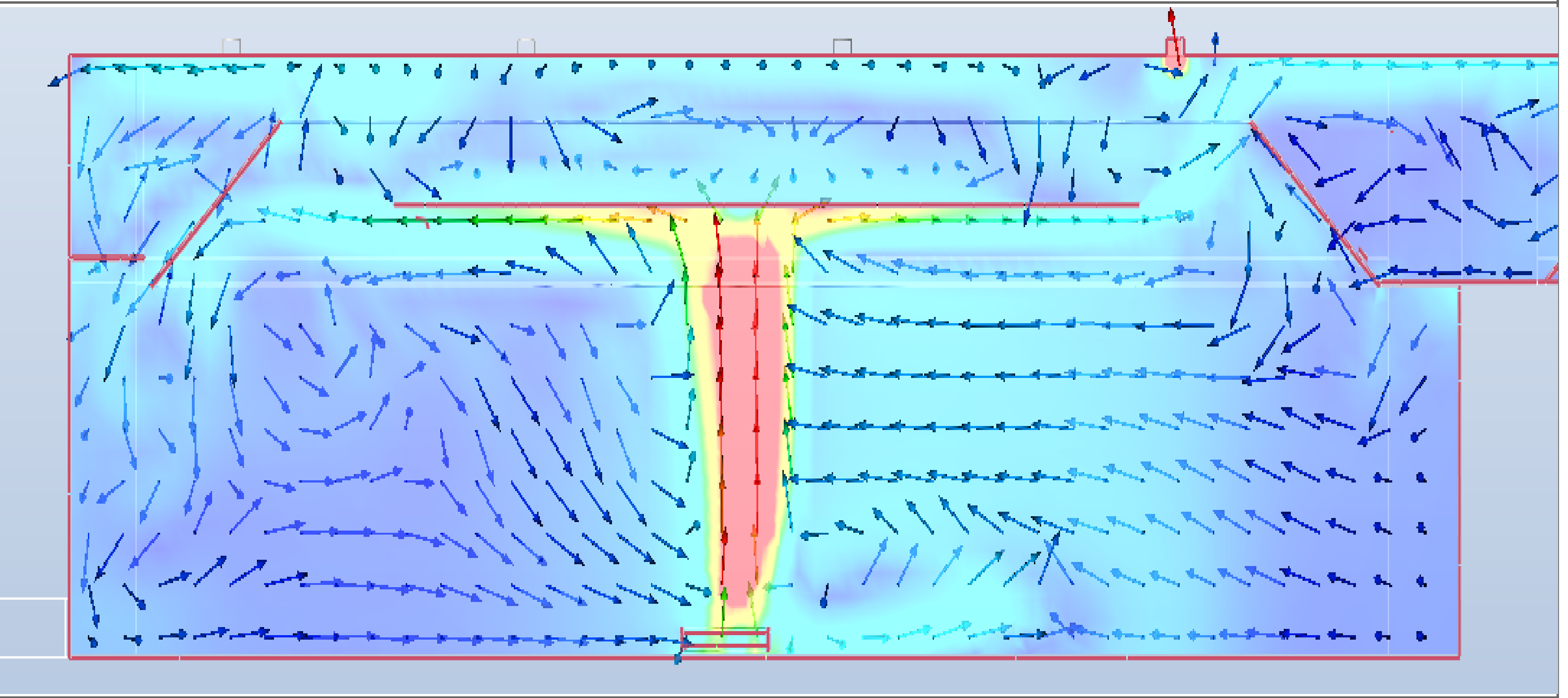
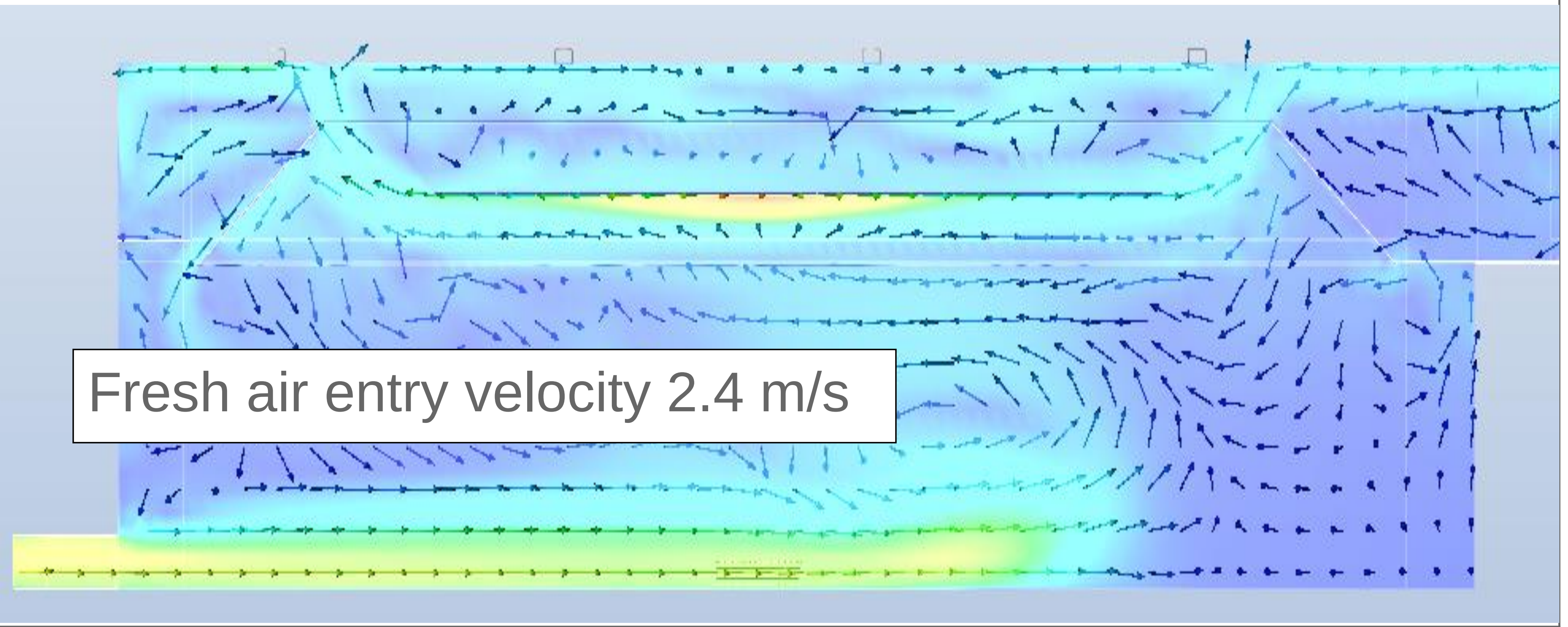
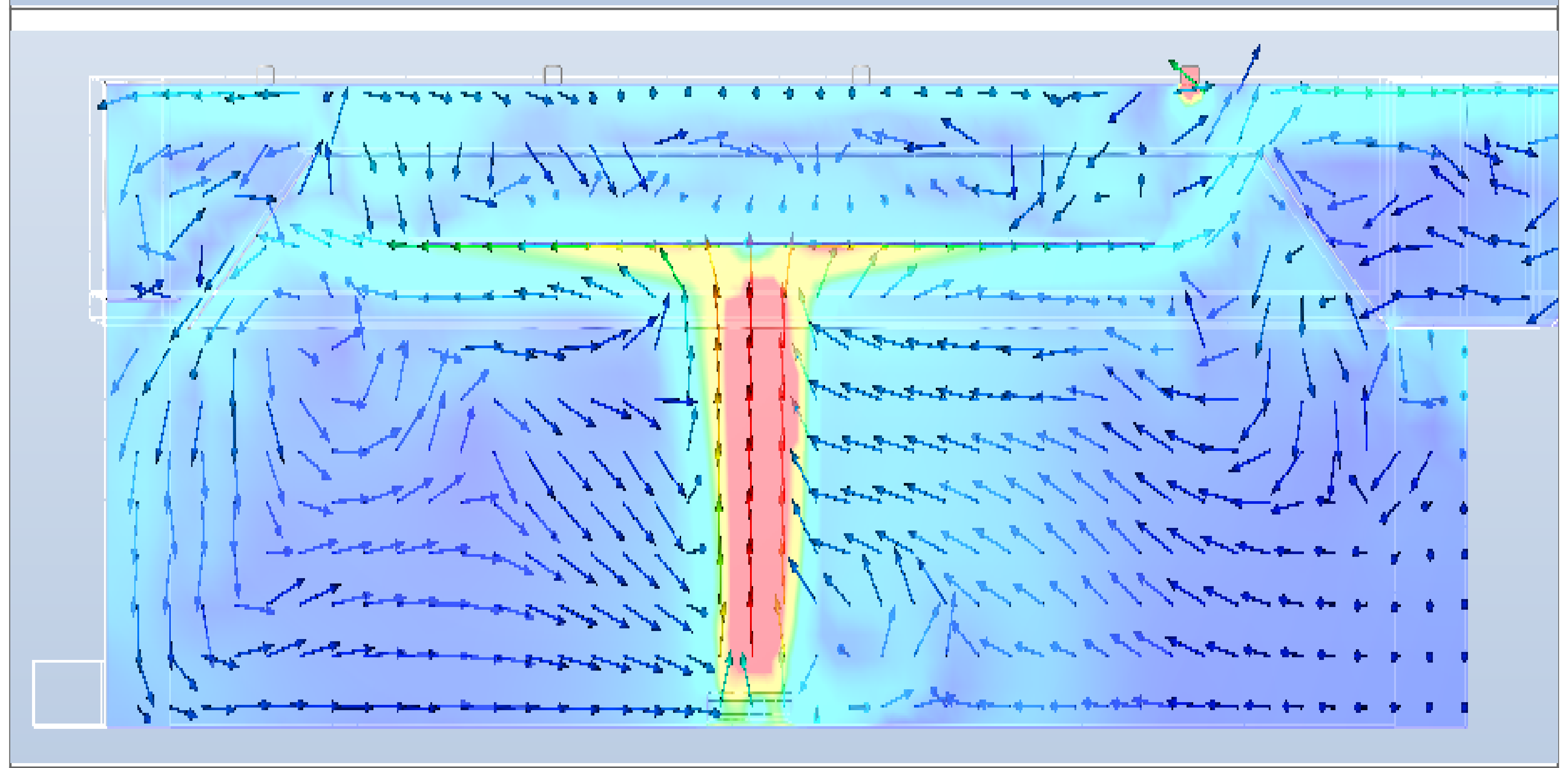
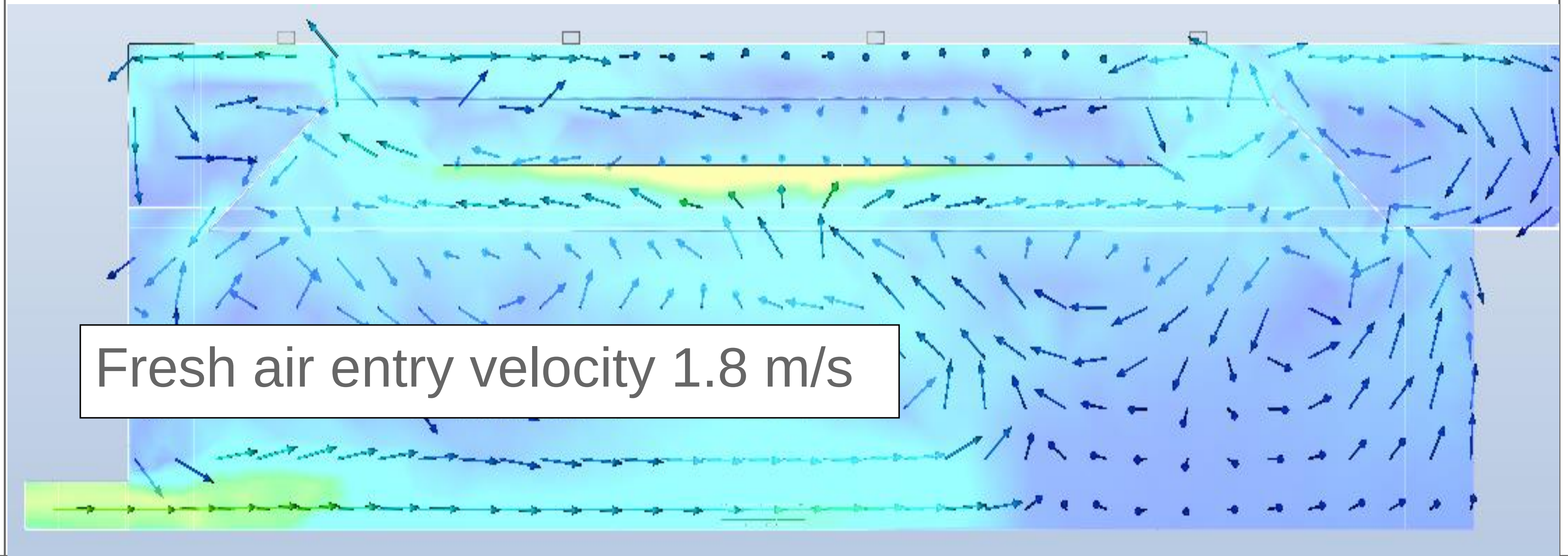
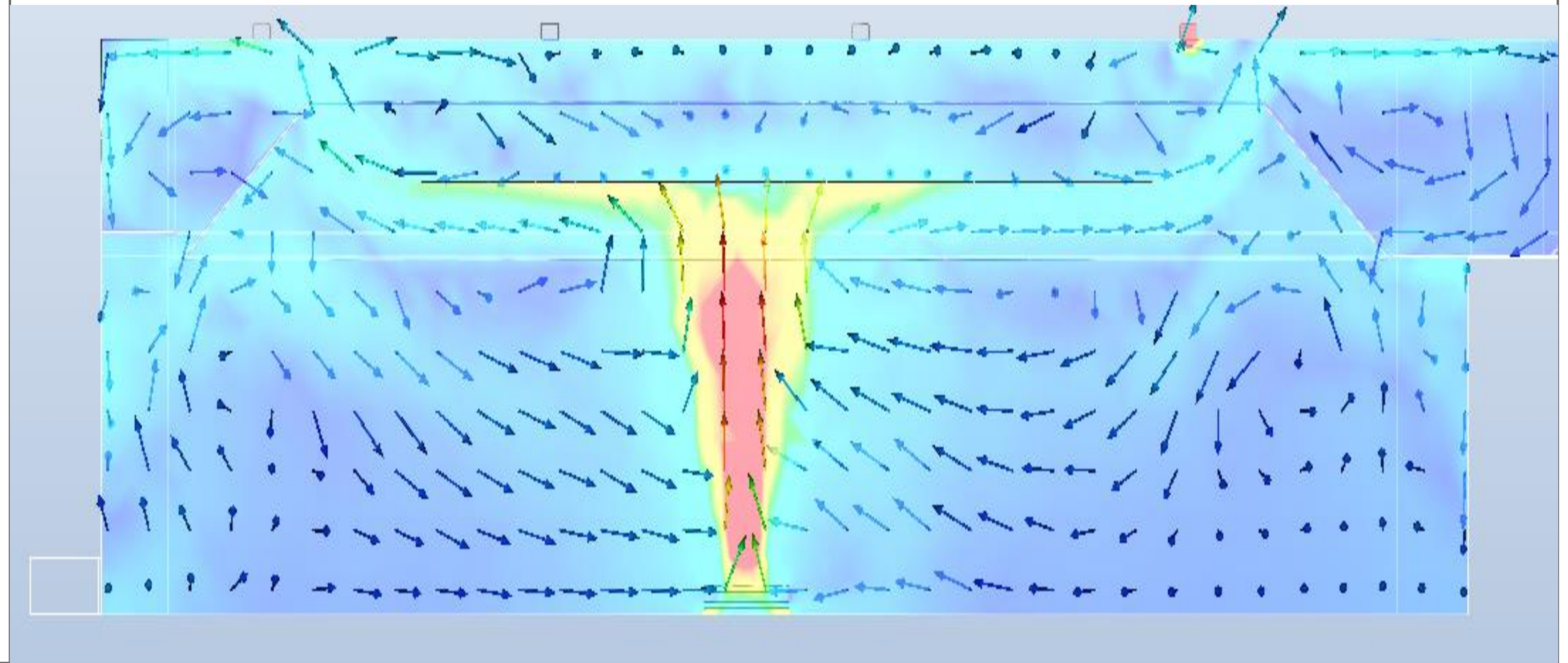


CFD analysis results –Smoke Temperature

Time (s)	Initial Design	Design Option 1	Design Option 2
90			
180			



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

	Thru door		Thru Fire
Initial Design			
Design Option1			
Design Option 2			

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 60^0 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

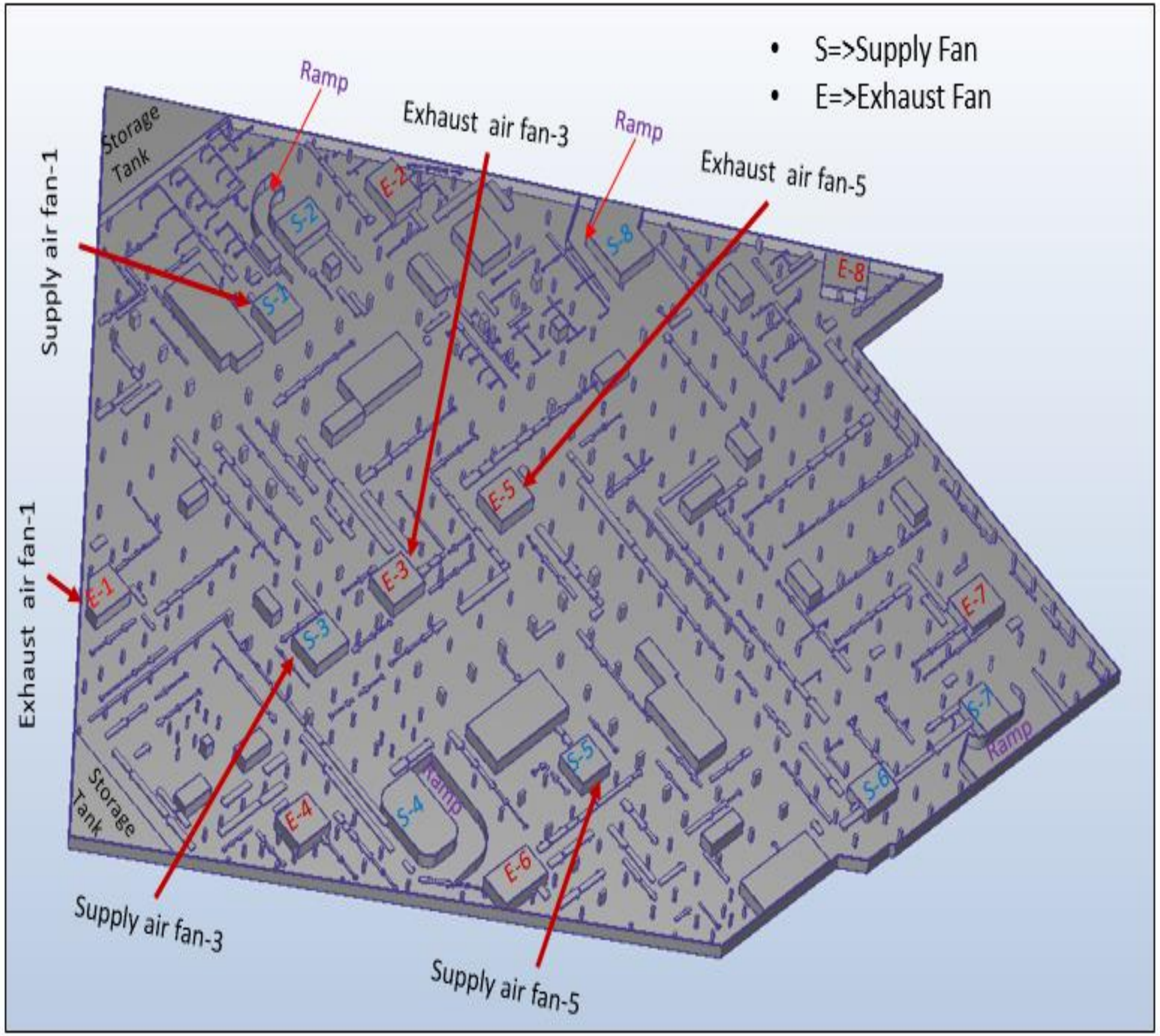
Design performance criteria:

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

CAD Model
Basement Car Park

Basement car park details

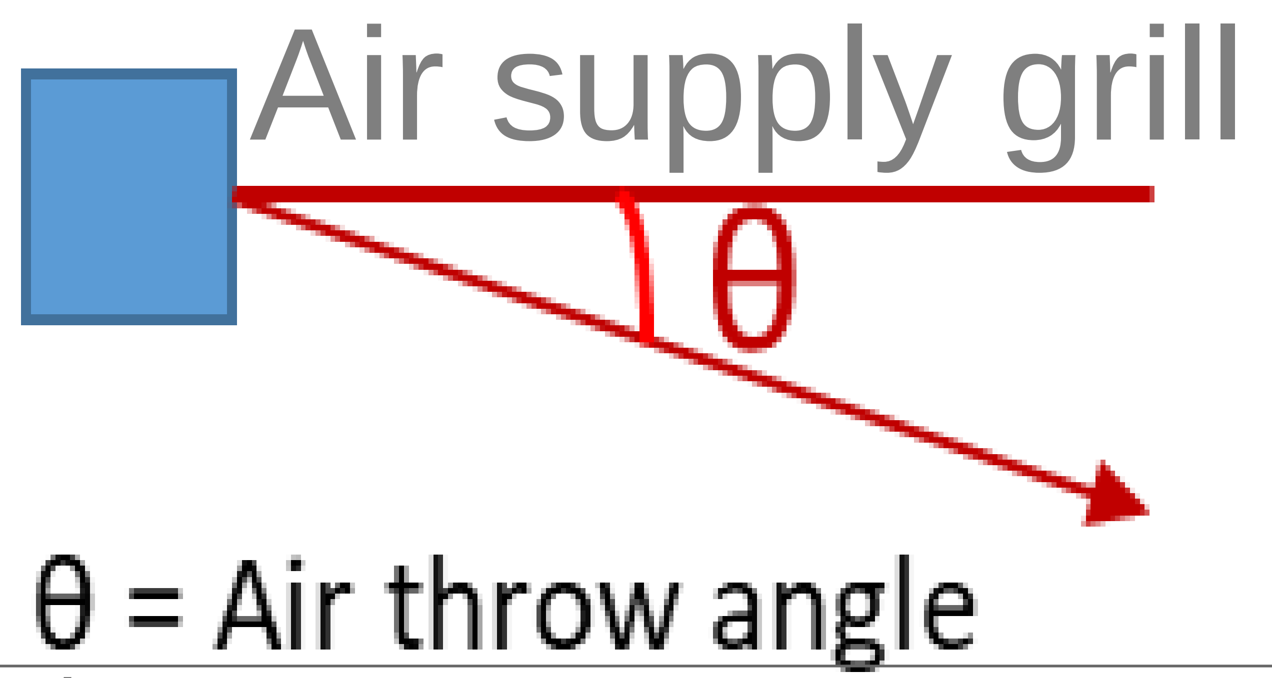
- Basement area = 46500 sq. m
- Height = 3.15 m
- No. entry/exit ramps = 4
- Normal mode ventilation= 6 ACH
- Total no. of fans = 8



Key Results for Design Performance

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

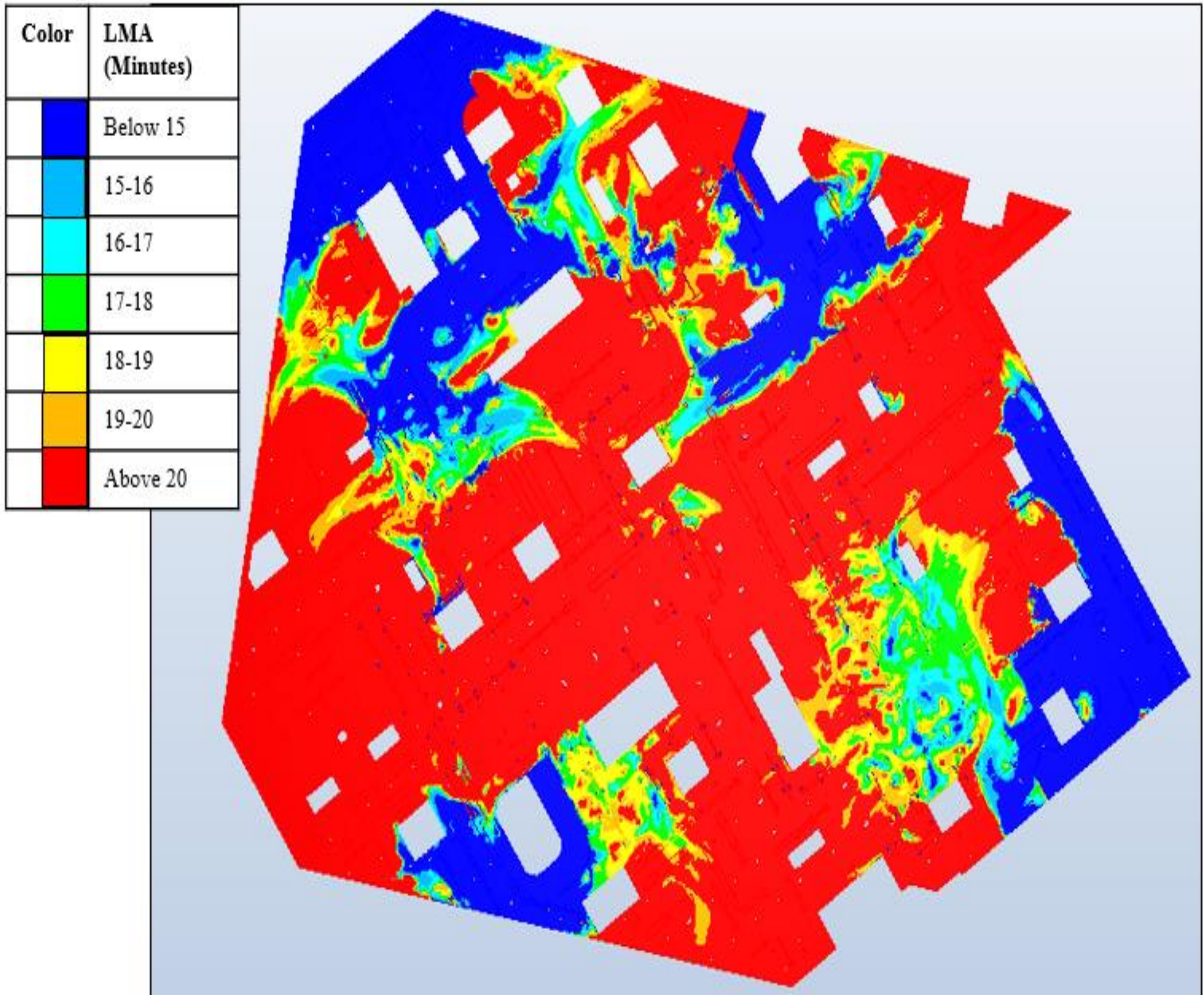
Basement Ventilation
Performance Evaluation

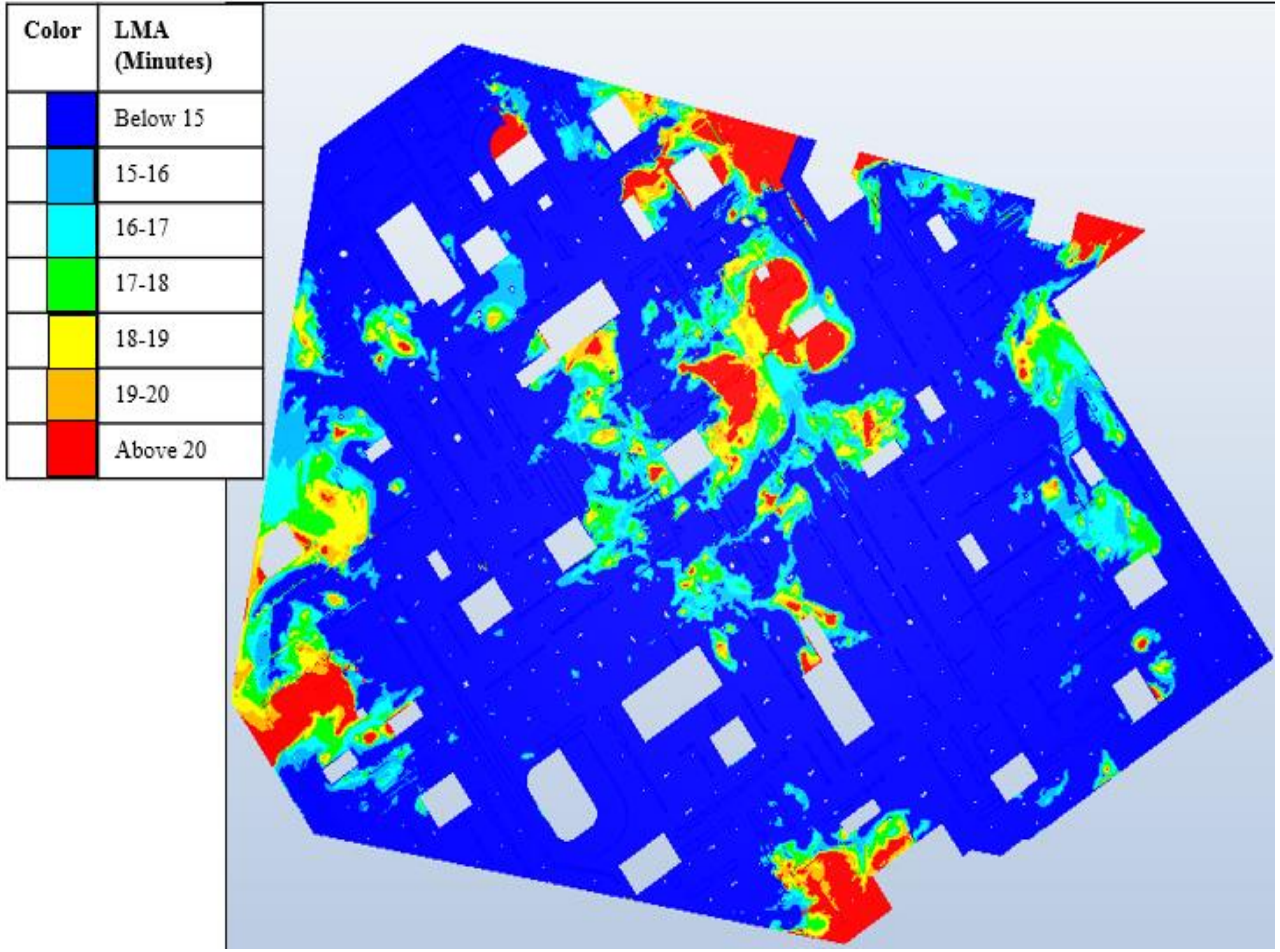


	Supply air flow direction (θ)	Exhaust air flow direction (θ)	Flow rate
Initial Design	0 degrees (horizontal)	0 degrees (horizontal)	Based on 6 ACH from 8 fans
Design Option 1	90 degrees (vertically downwards)	0 degrees (horizontal)	30% extra CFM from all fans (equivalent to 7.18 ACH)
Design Option 2	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

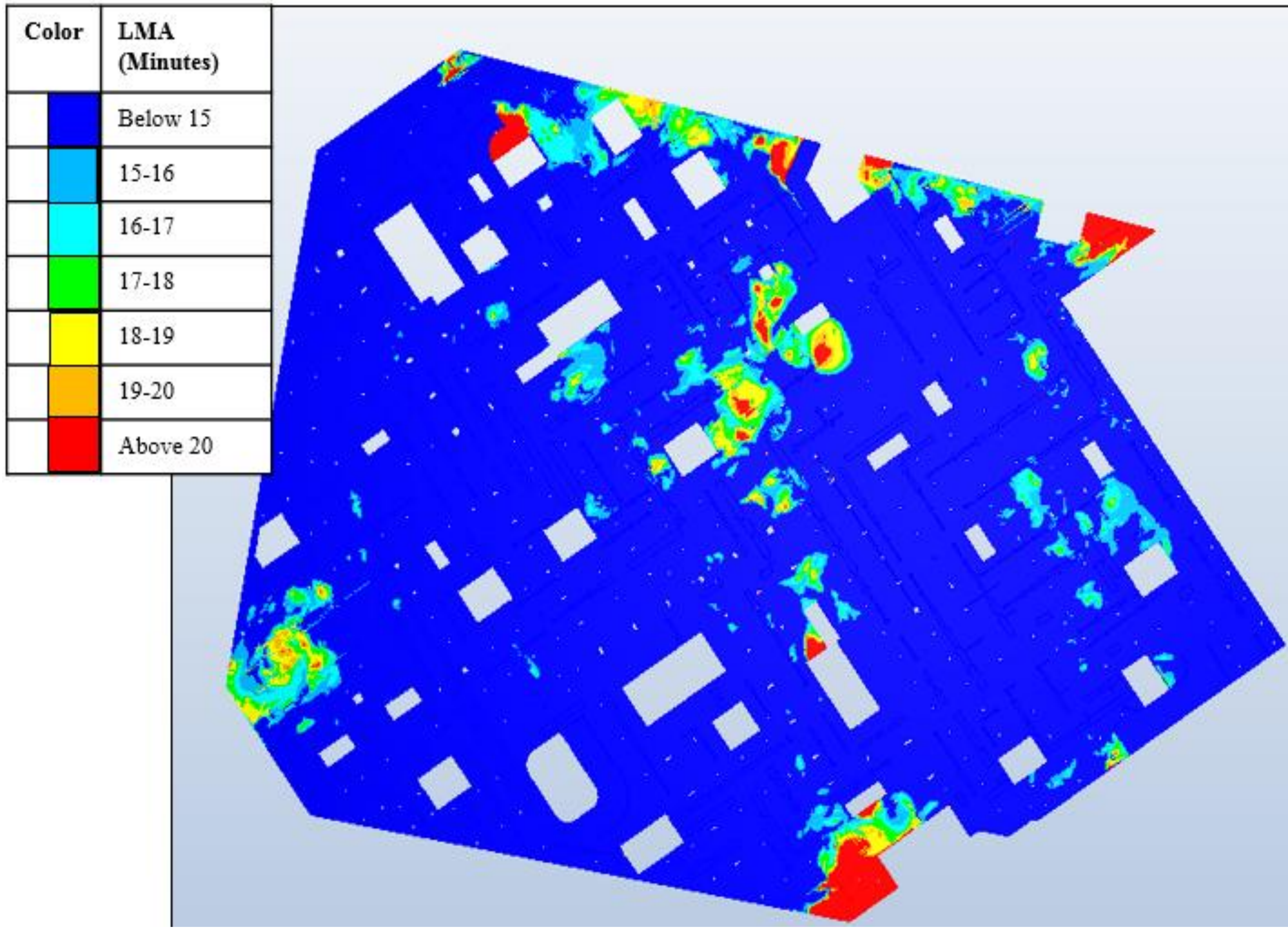




Design Option 1:

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

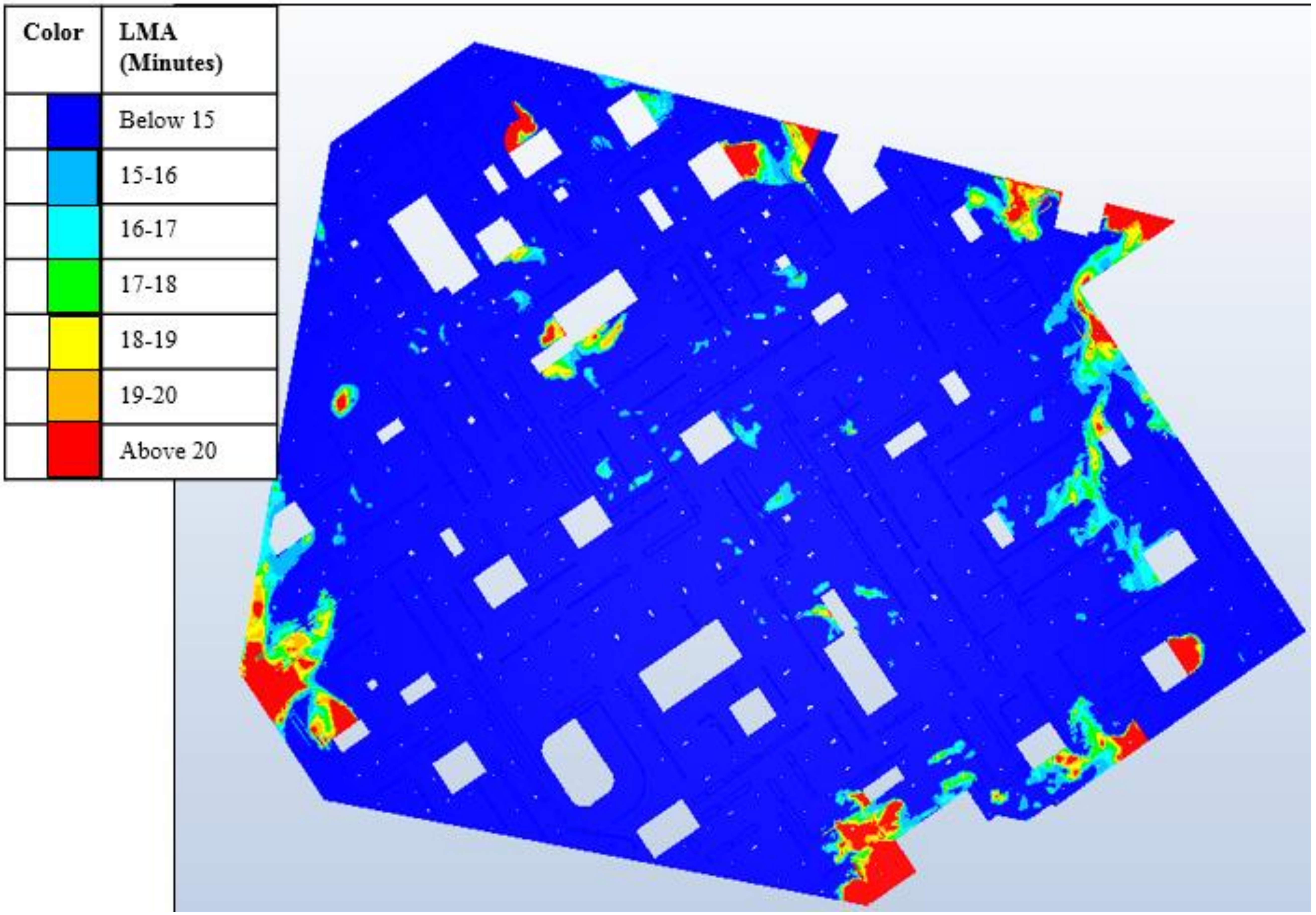
LMA values<15 minutes for ~73% of area



Design Option 2:

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

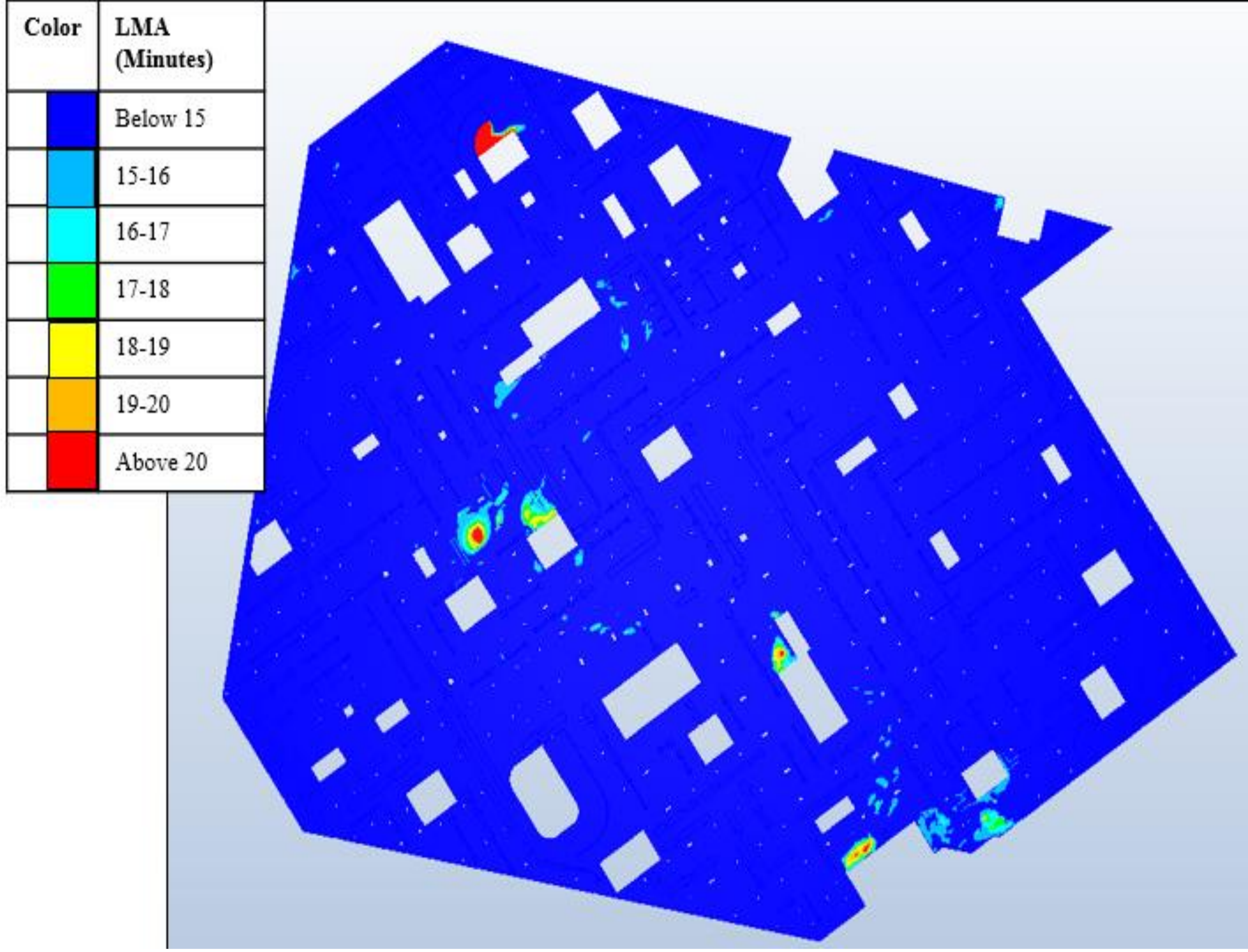
LMA values <15 minutes for ~ 89% of area



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:

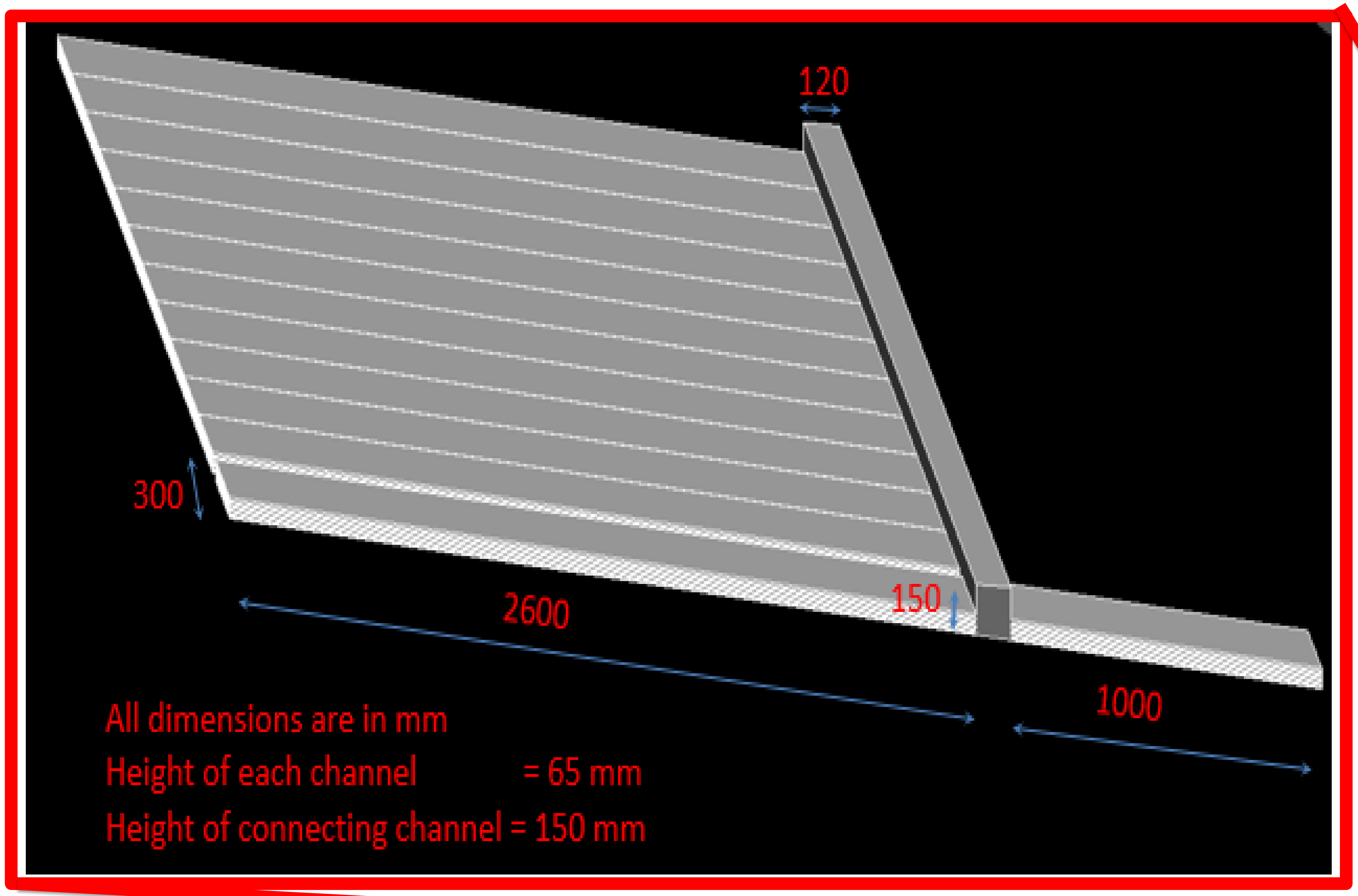
- Predict water levels at Skylight obstructions
- Recommend corrective action

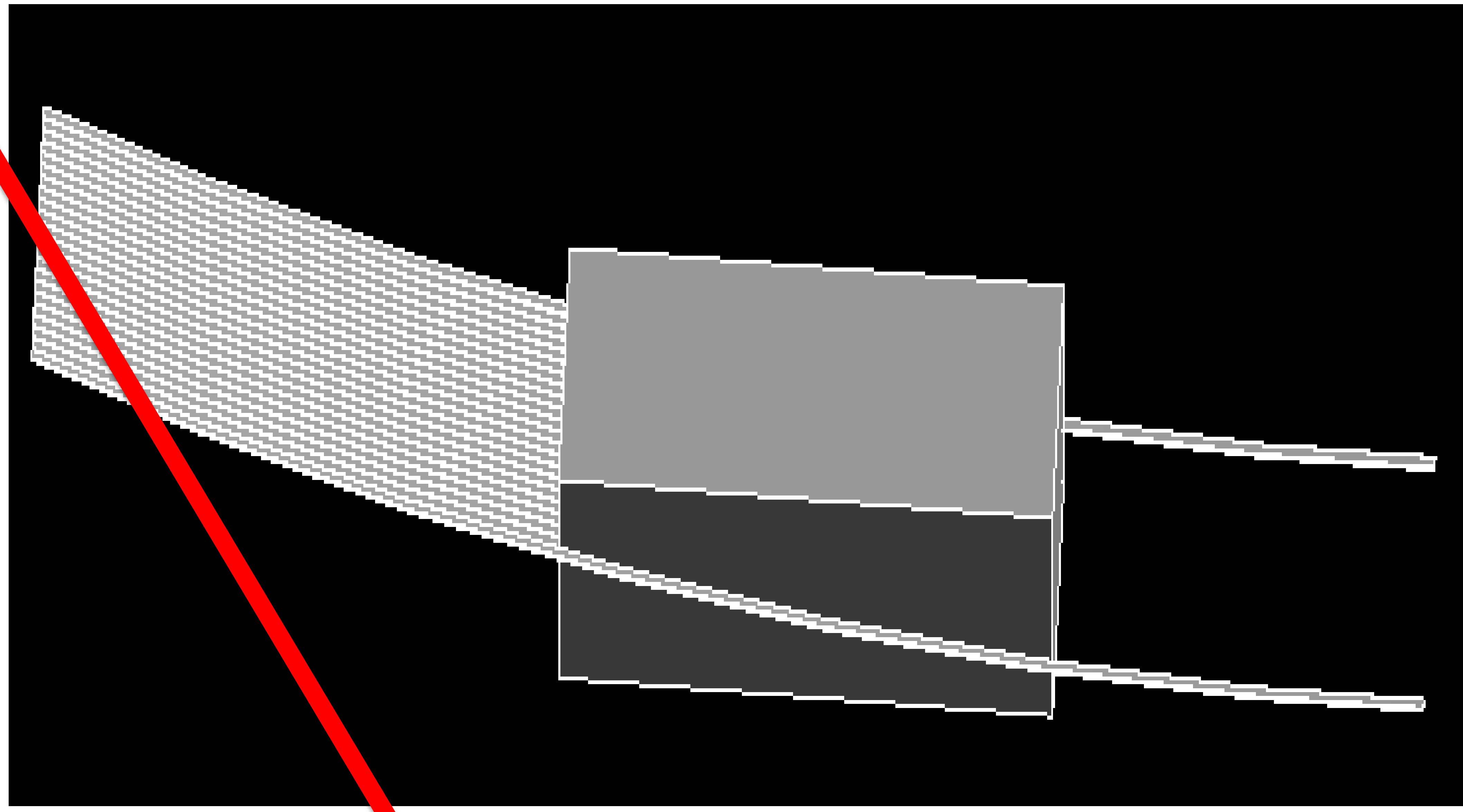
Design performance criteria:

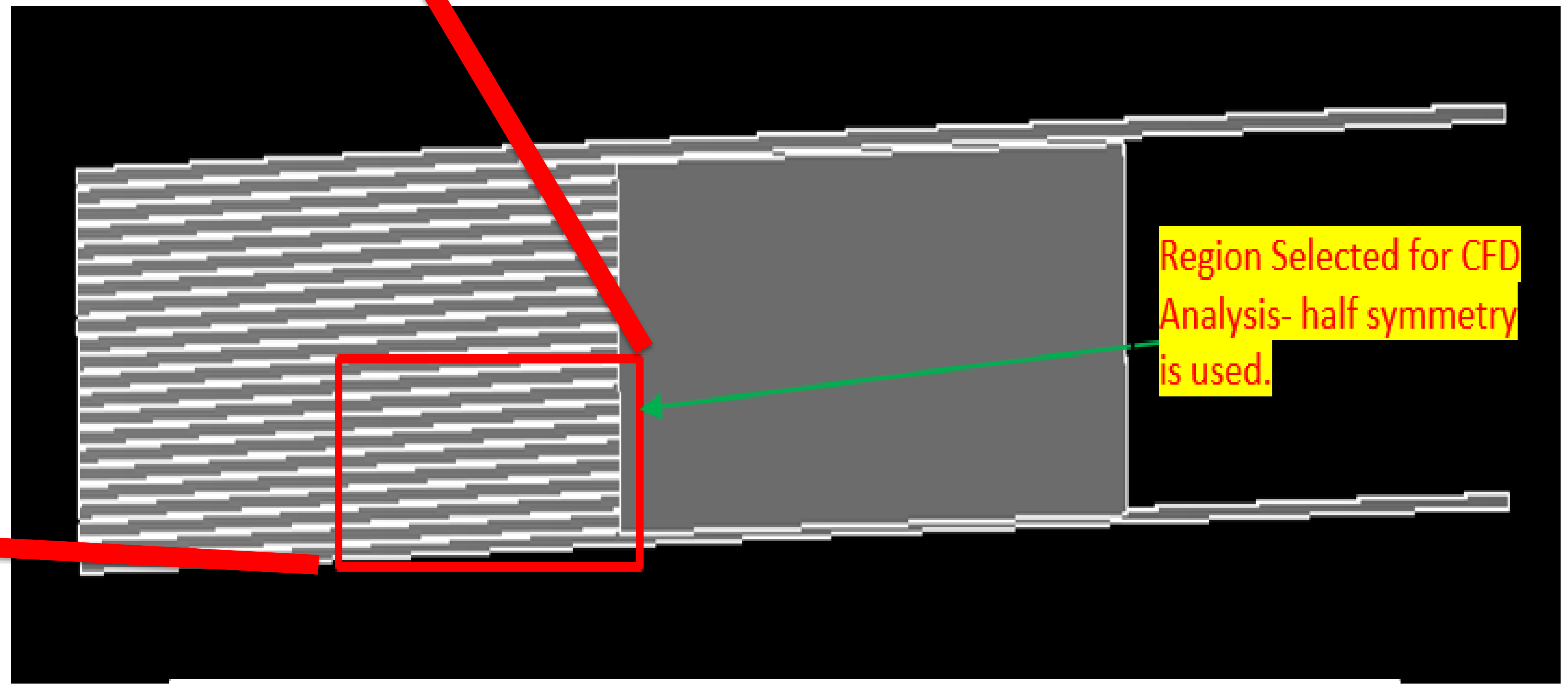
- Water level in roof channels below 65mm

CAD Model

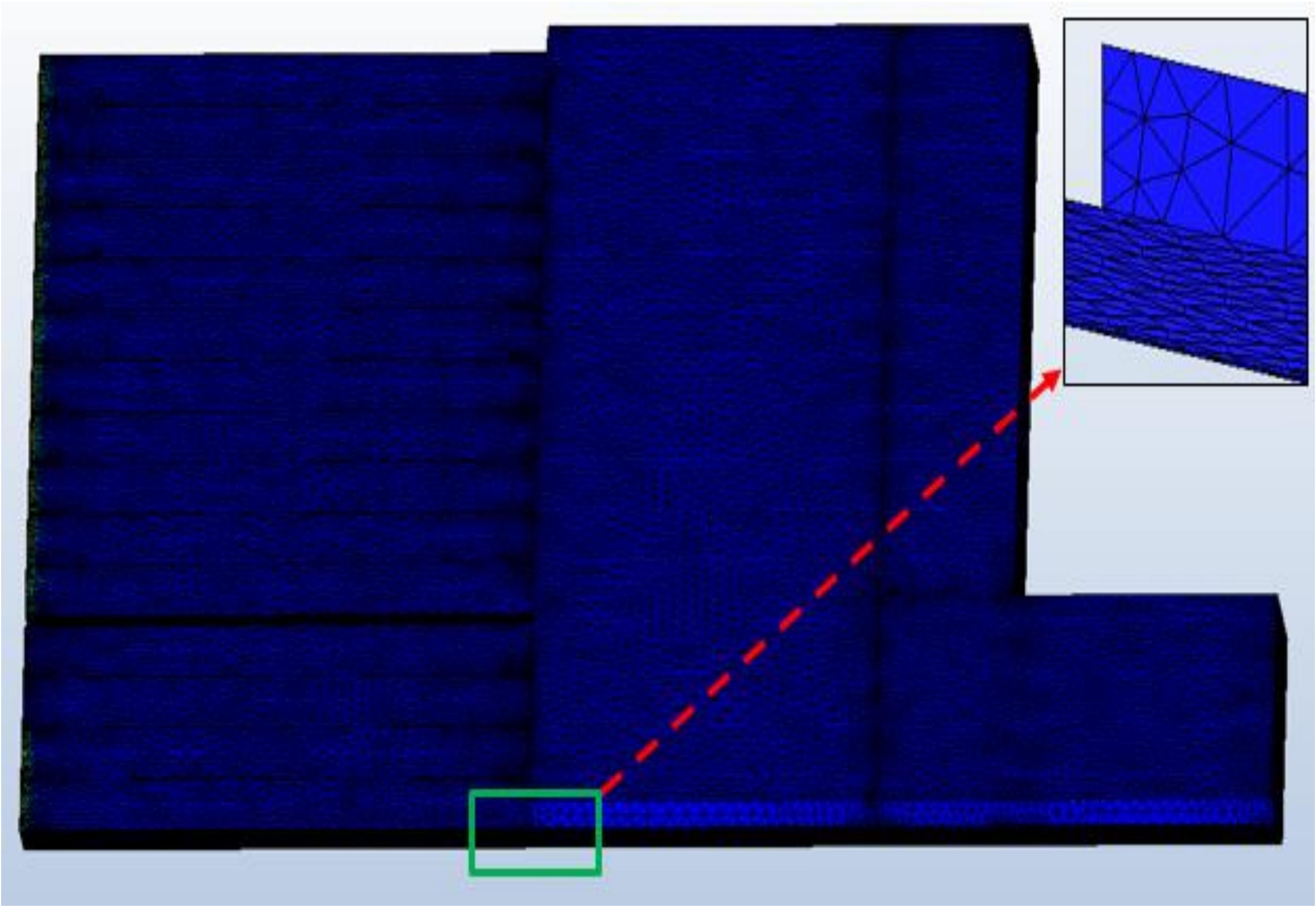
Airport roof with skylight portion







CFD modeling
Free surface flow



Material Properties:

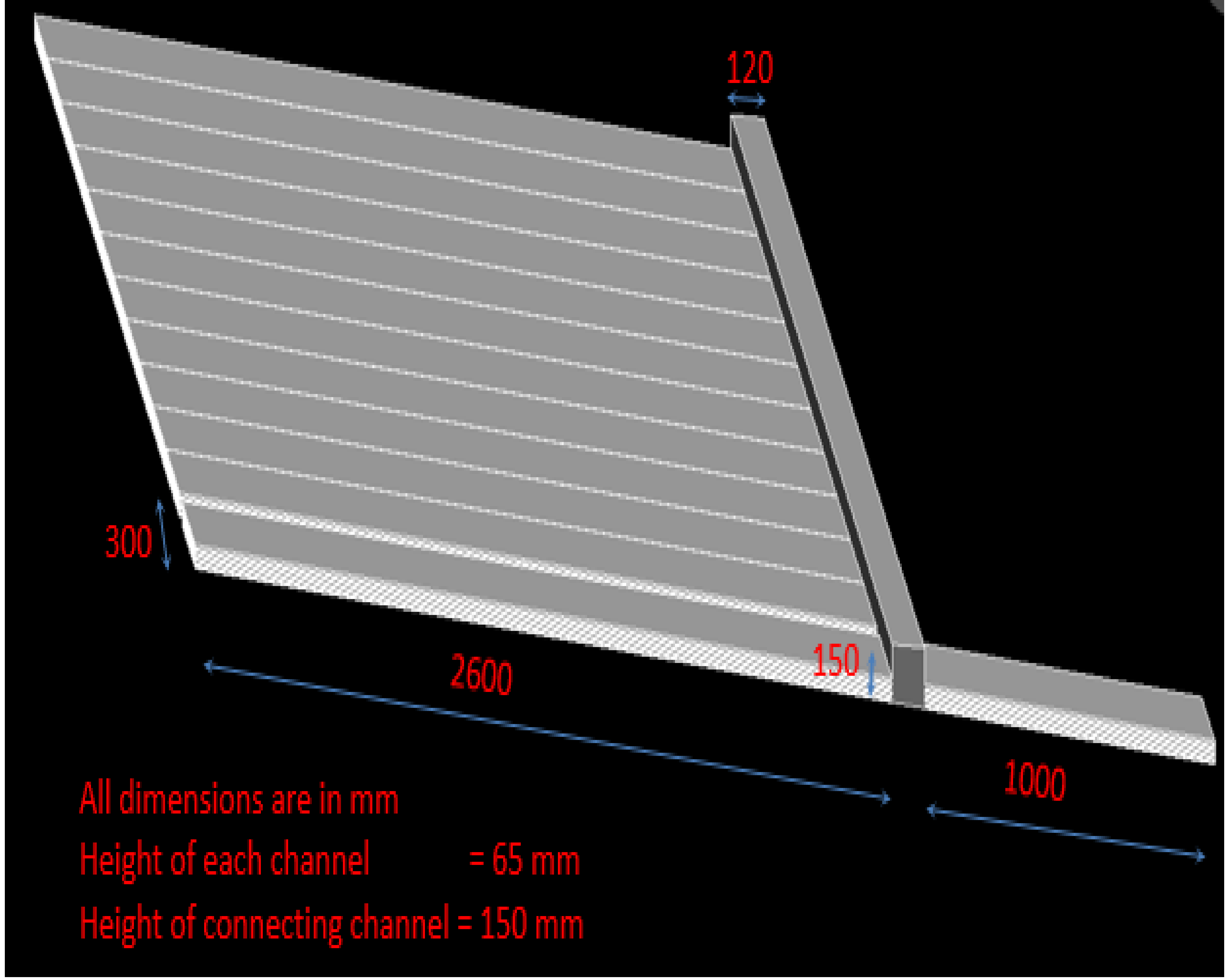
- Water : Fixed Quantity

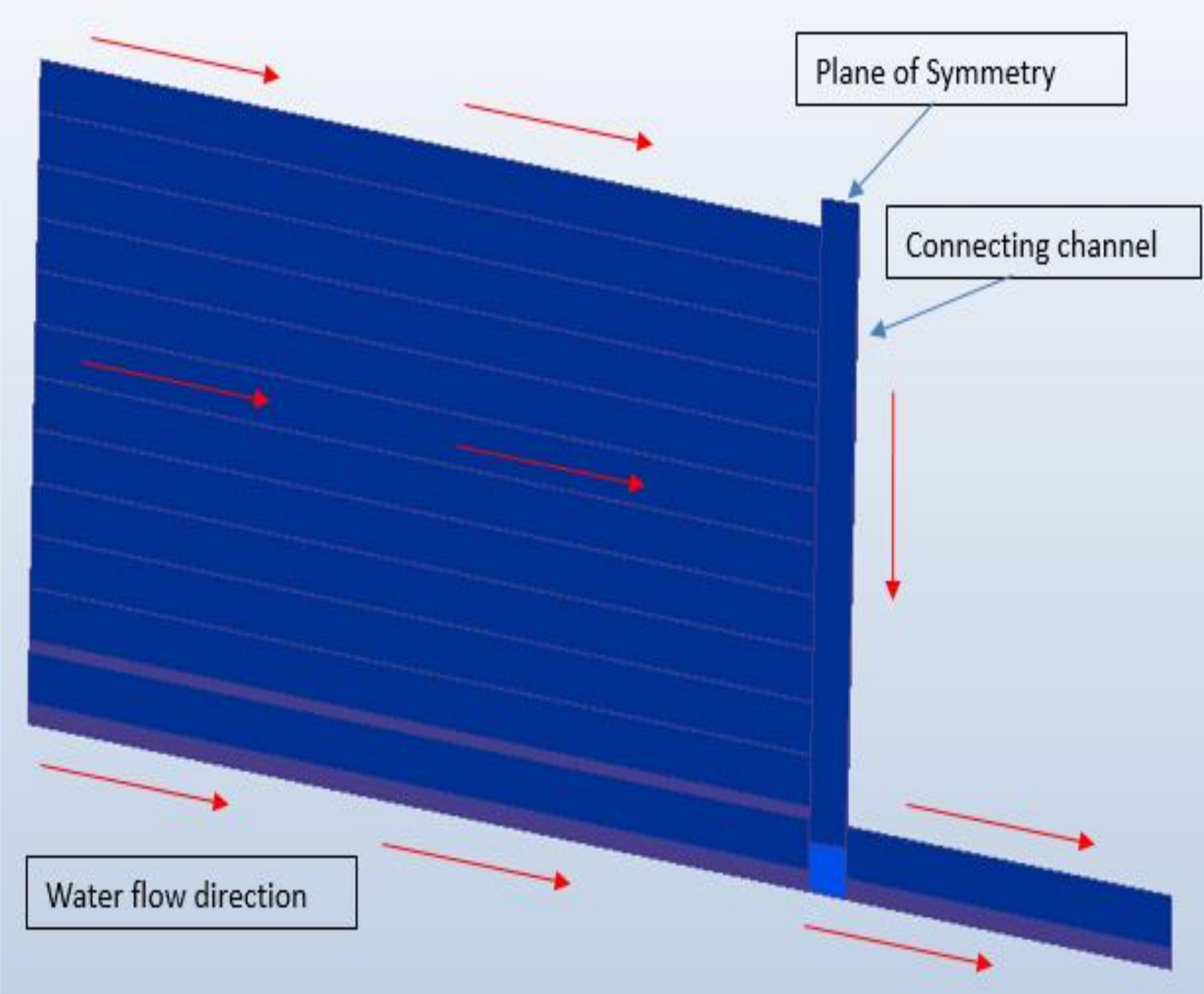
Boundary conditions:

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

Solution Parameters:

- Default values in CFD S/W

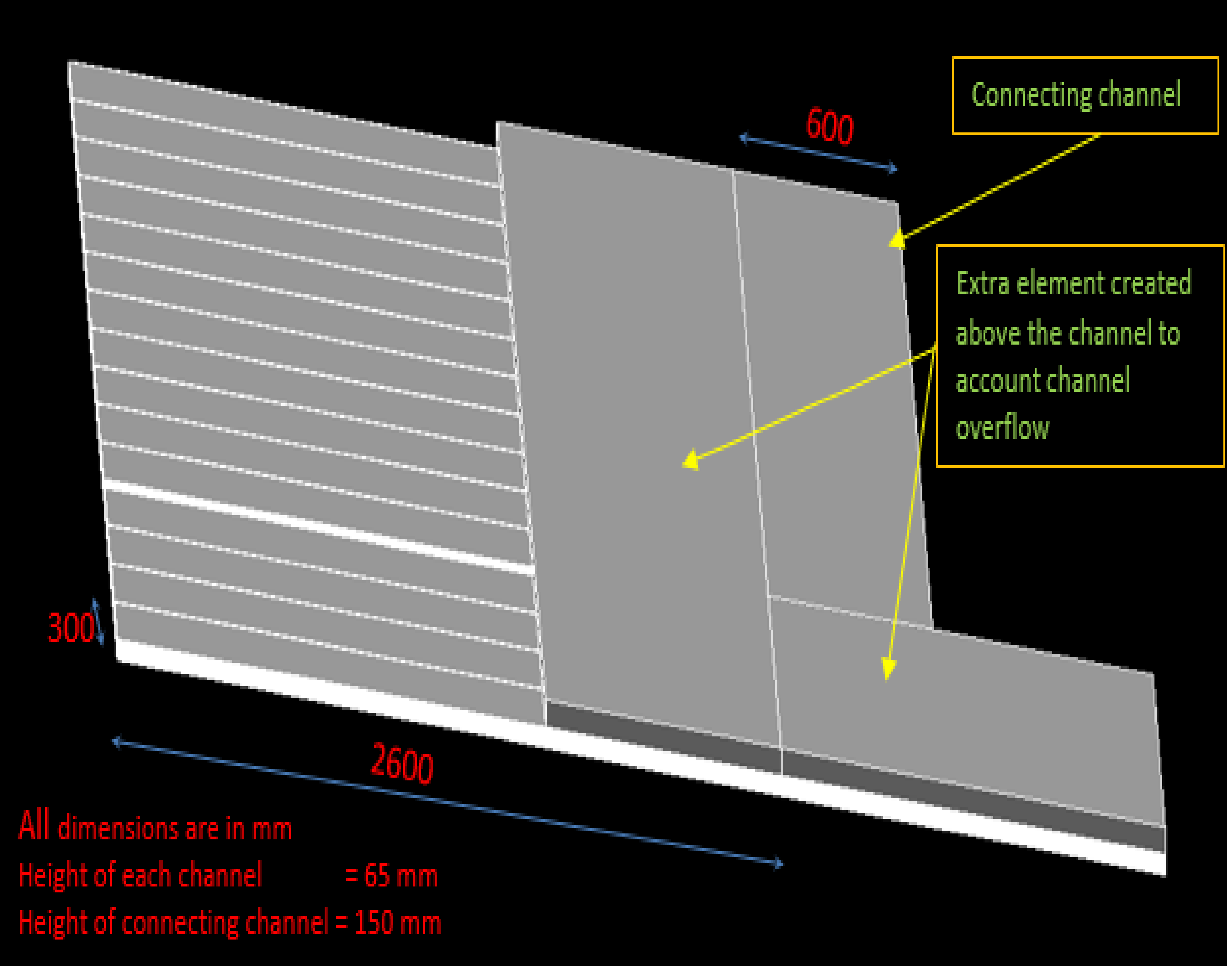


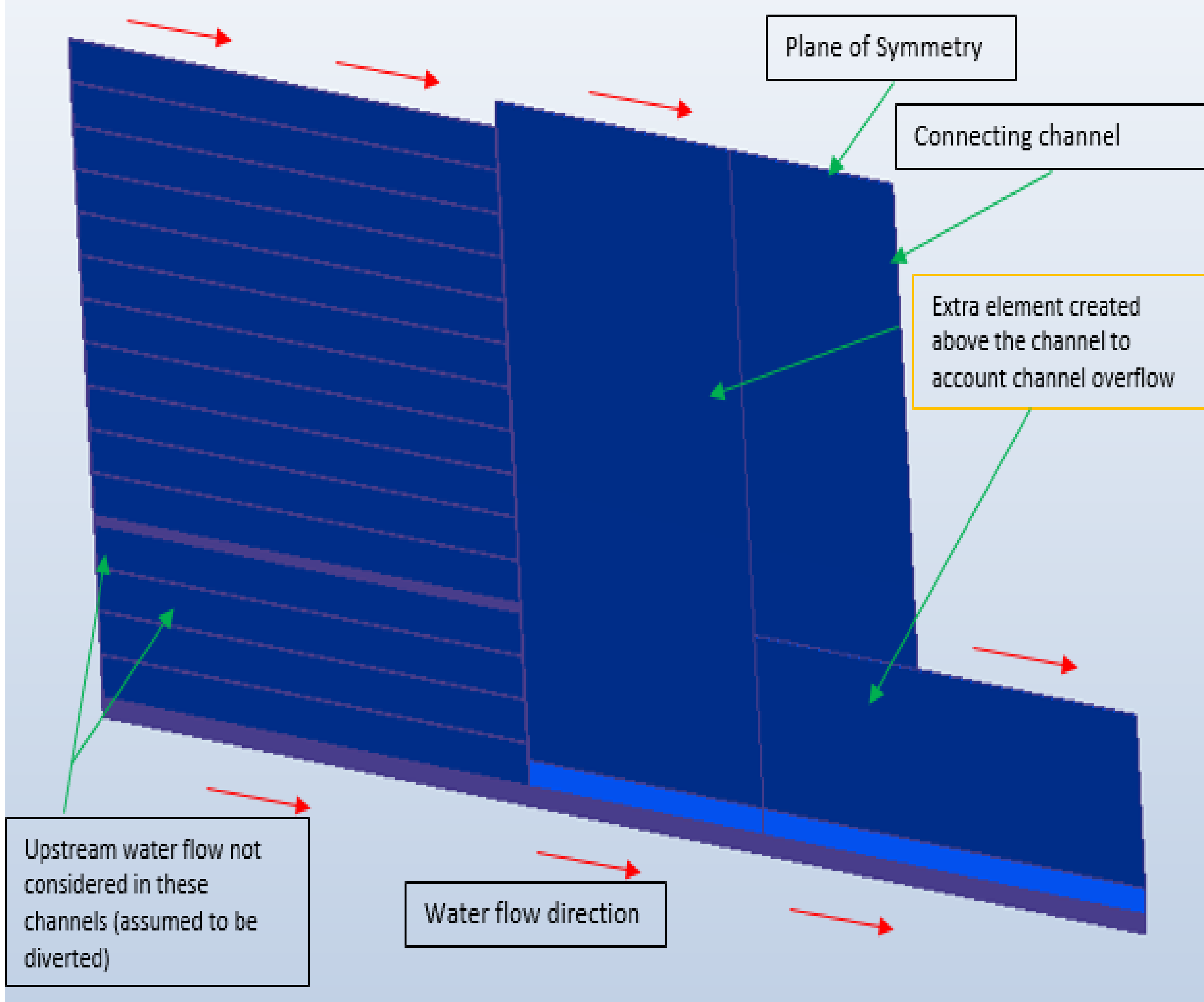


CAD model- Initial design

Connecting channel width = 120 mm

CFD model- Initial design





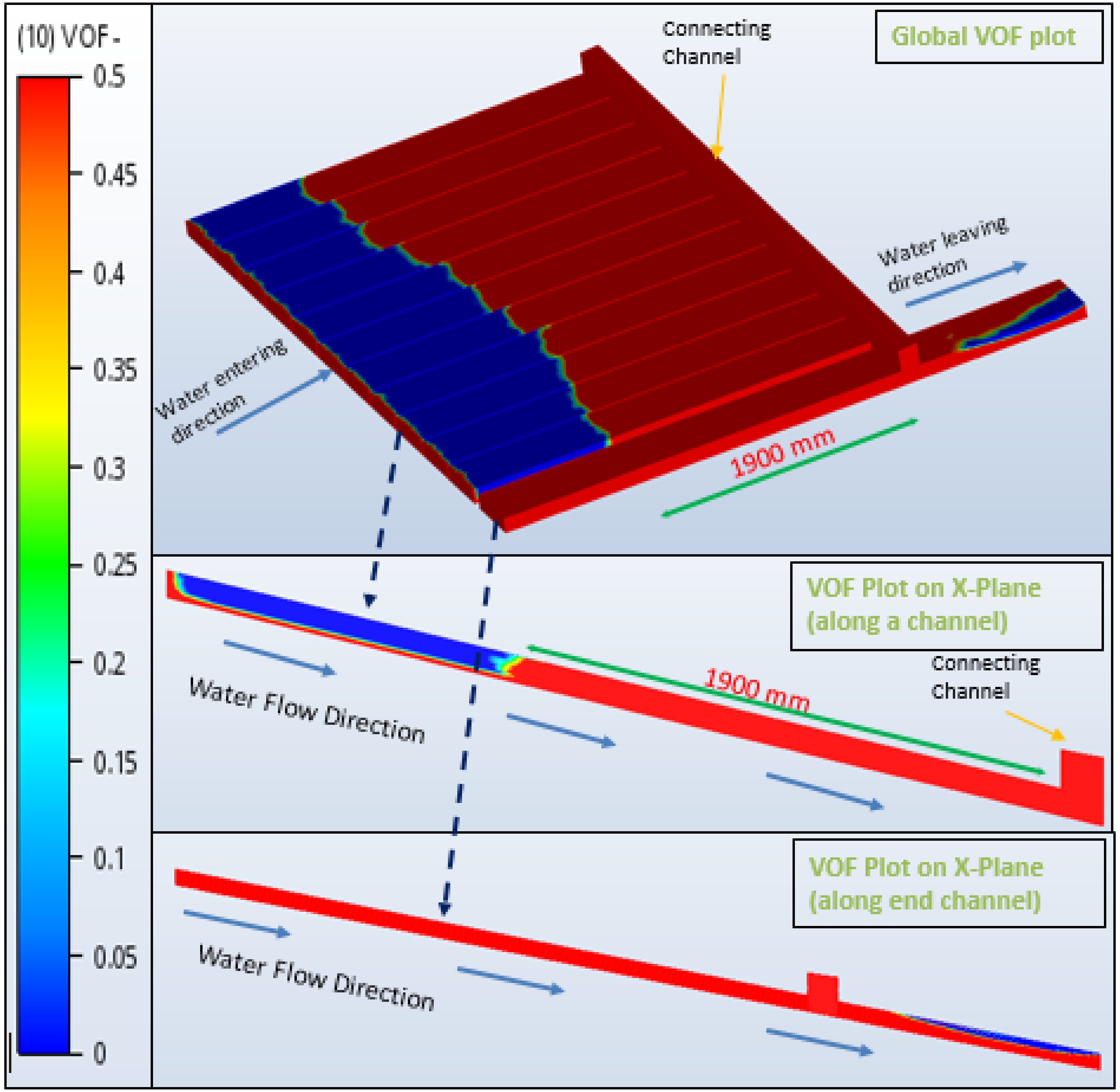
CAD model- Final design

Connecting channel width = 600 mm

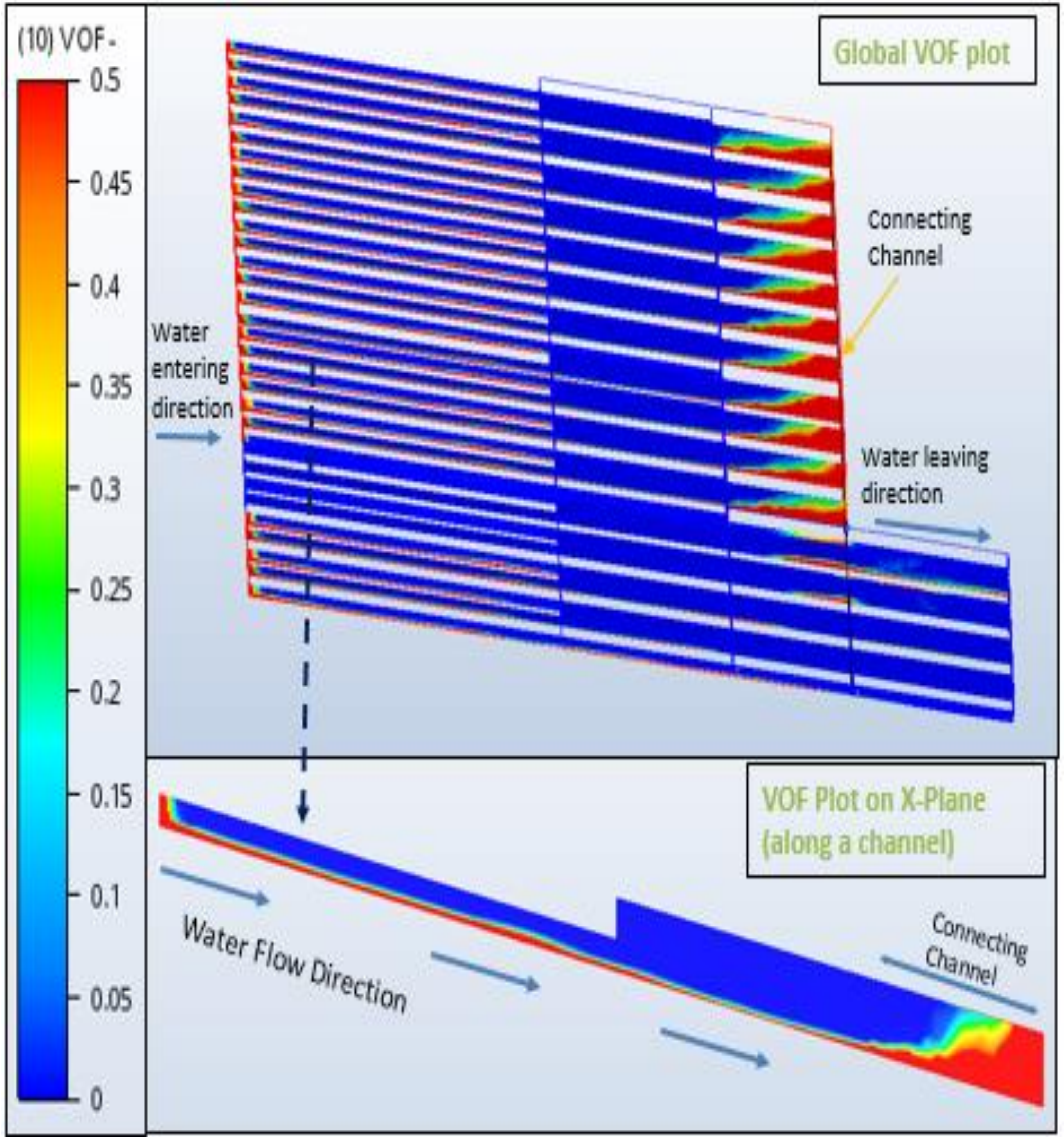
CFD model-Final design

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.


Q&A

Thank you for listening….

Keep in touch!

mmrajulu@lntecc.com

For info on who we are and what we do…

www.lntecc.com

www.larsentoubro.com

LARSEN & TOUBRO
It's all about Imagineering

L&T Construction

AUTODESK®

Make anything™