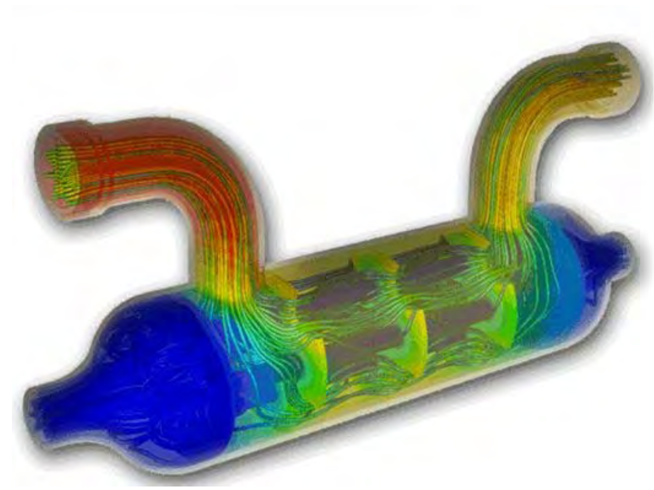
جمعية رواد الهندسة والتكنولوجيا

Theory & Applications of Computational Fluid Dynamics CFD

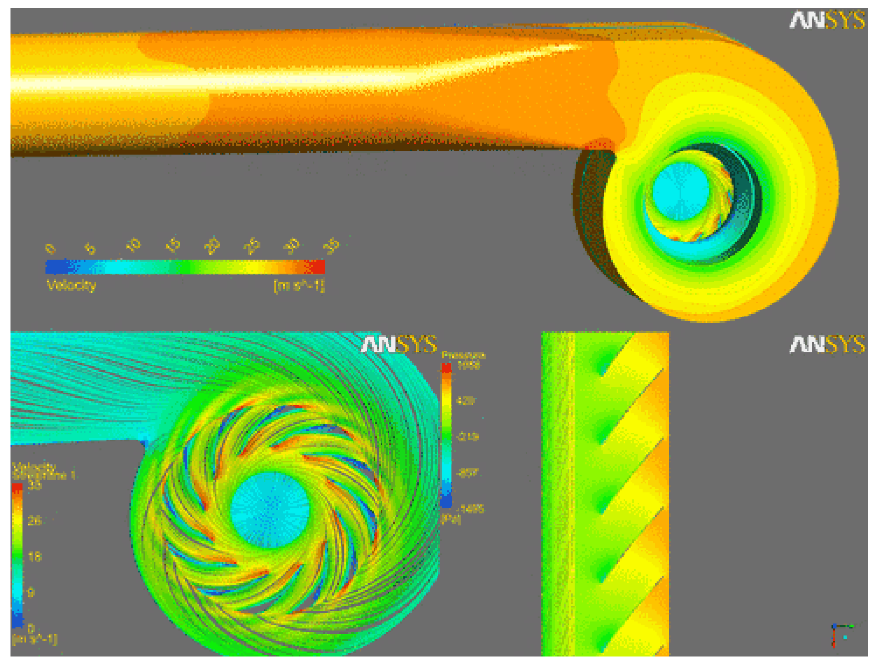


Prepared and Presented By:
Hesham Sami Abdul Munem
Mechanical Engineer –
Pressure Vessels Department - ENPPI

جمعية رواد الهندسة والتكنولوجيا

Contents:

- Presentation objective
- Fluid flow
- What is CFD
- Why use CFD
- How CFD works
- Experiments VS Simulation
- Applications
- Verification & validation of CFD Results
- Case Studies



جمعية رواد الهندسة والتكنولوجيا

Objective:

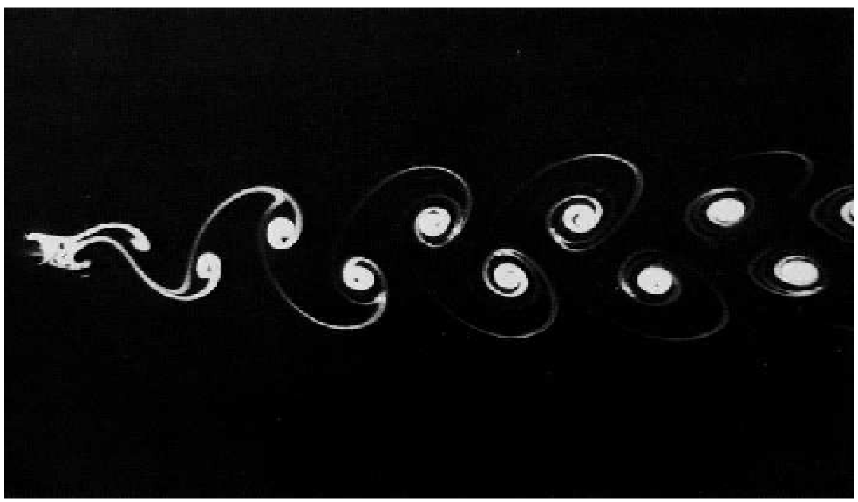
- To present a general overview for Computational Fluid Dynamics CFD techniques including some applications in different engineering design fields

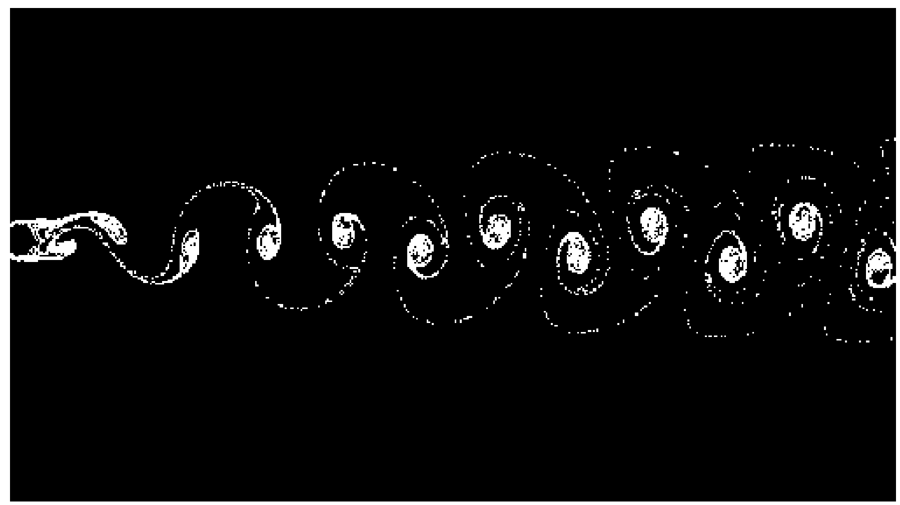
Fluid Flow

- Fluid flow is encountered in everyday's life; examples are:
  - Flow within pipes, pumps, heat exchangers, towers, chemical reactors
  - Heating, ventilating and air conditioning of buildings, cars,
  - Meteorological phenomena (wind, rain, hurricanes, floods)
  - Environmental hazards (air pollution, contaminants transport)
  - Human body (blood flow, breathing, drinking)
  - And many other examples

What is CFD?

- CFD is a part of fluid mechanics that applies *numerical methods* and algorithms in order to solve and analyze problems that relates to flows of fluid.
- CFD enables scientists and engineers to perform ‘numerical experiments’ (i.e. computer simulations) in a “virtual flow laboratory”





- Experiment
- CFD Simulation

How CFD Works?

- **<u>CFD general procedure is:</u>**

<u>I. Preprocessing</u>

1. Geometry is generated (physical bounds) of the problem is defined.
2. Volume occupied by fluid is divided into discrete cells (MESH).
3. Physical model is defined – for example, the equations of motions + enthalpy + radiation + species conservation
4. Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem
5. Initial conditions are also defined (only For transient problems)

How CFD Works?

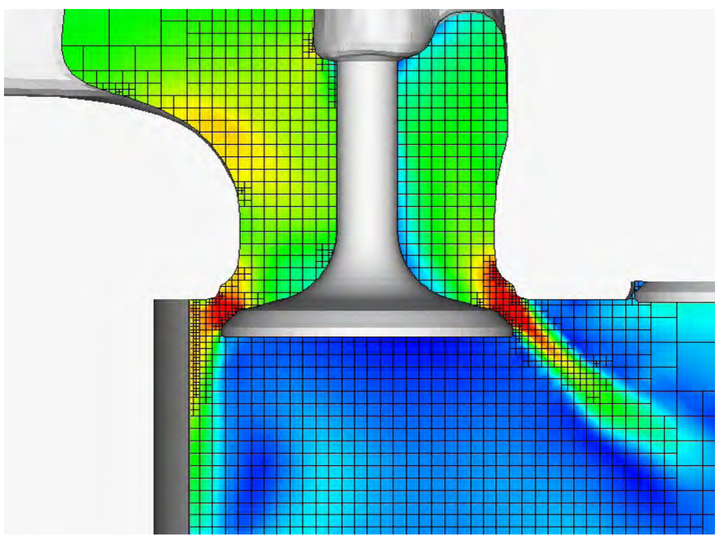
- **<u>CFD general procedure is:</u>**

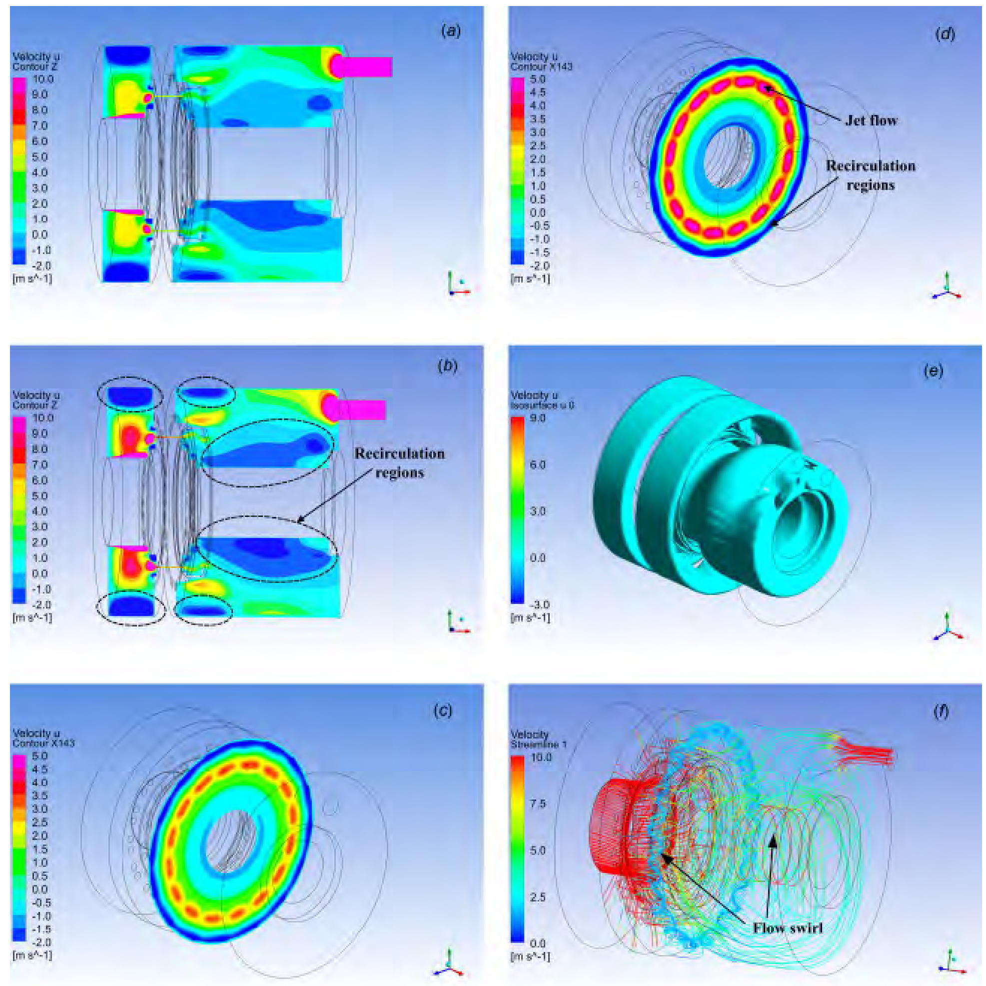
<u>II. Simulation:</u>

– Equations are solved <u>iteratively</u> over the domain

<u>III. Postprocessing</u>

– Analysis and visualization of the resulting solution are performed





Experiments vs Simulations

EXPERIMENTS	SIMULATIONS
<u>Description</u> of flow using **<u>measurements</u>**	**<u>Prediction</u>** of flow using **<u>CFD software</u>**
• for one quantity at a time	• for all desired quantities
• at a limited number of points and time instance	• with high resolution in space and time
• for a laboratory scale model	• for the actual flow domain
• for a limited range of problems and operating conditions	• for virtually any problem and realistic operating conditions

Experiments vs Simulations

EXPERIMENTS	SIMULATIONS
• Expensive • slow • sequential • Equipment and personnel are difficult to transport	• cheap(er) • fast(er) • parallel • CFD software is portable, easy to use and modify

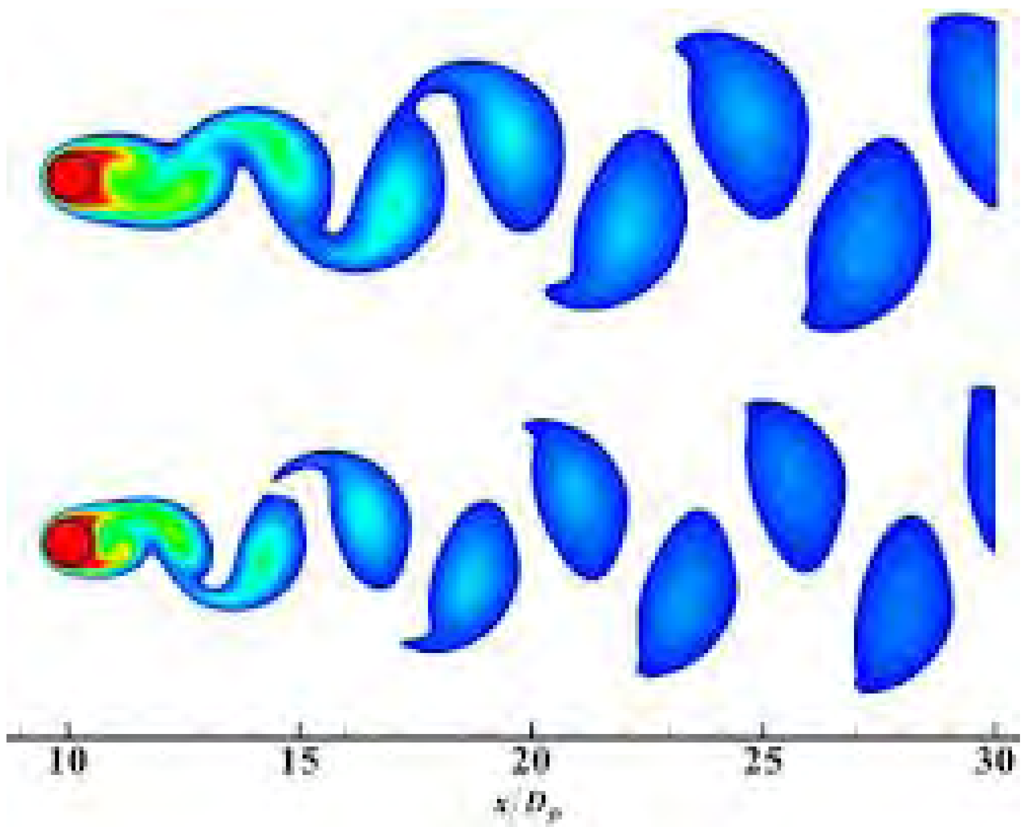
- Fluid flows can be classified into:

VISCOUS	**INVISCID**
COMPRESSIBLE	**INCOMPRESSIBLE**
STEADY	**UNSTEADY**
LAMINAR	**TURBULENT**
SIGNLE-PHASE	**MULTI-PHASE**

- CFD simulation has a greater reliability
  - ❑ For laminar/slow flows than for turbulent/fast ones
  - ❑ For single-phase flows than for multi-phase ones
  - ❑ For chemically inert systems than for reactive flows

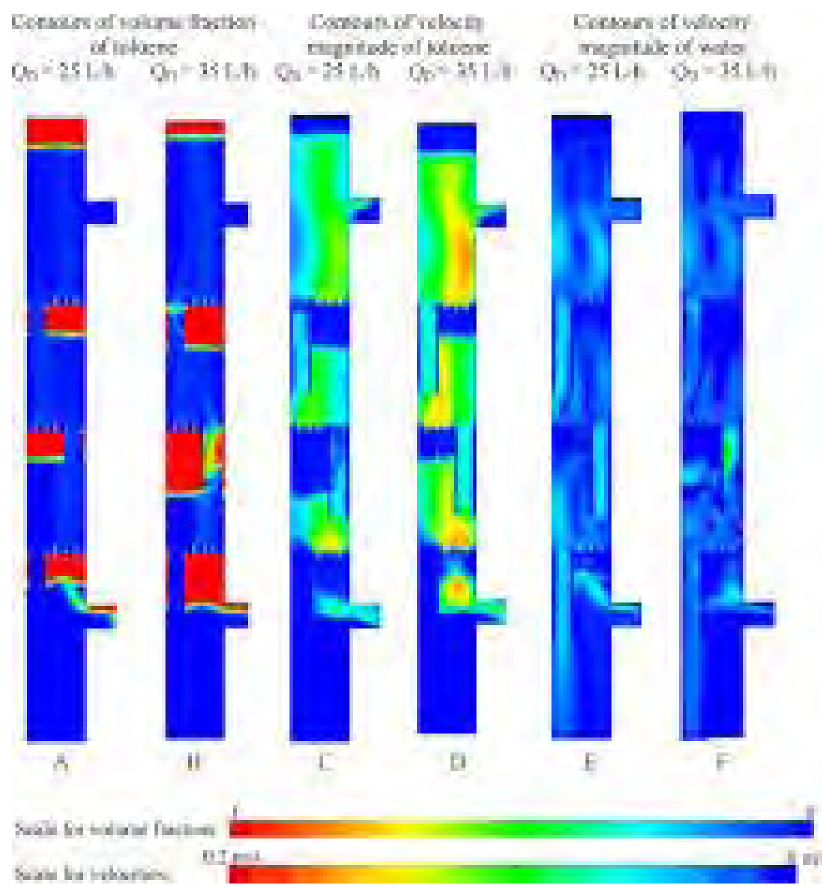
Typical examples of CFD Industrial Applications

- Design of tall towers against vortex shedding



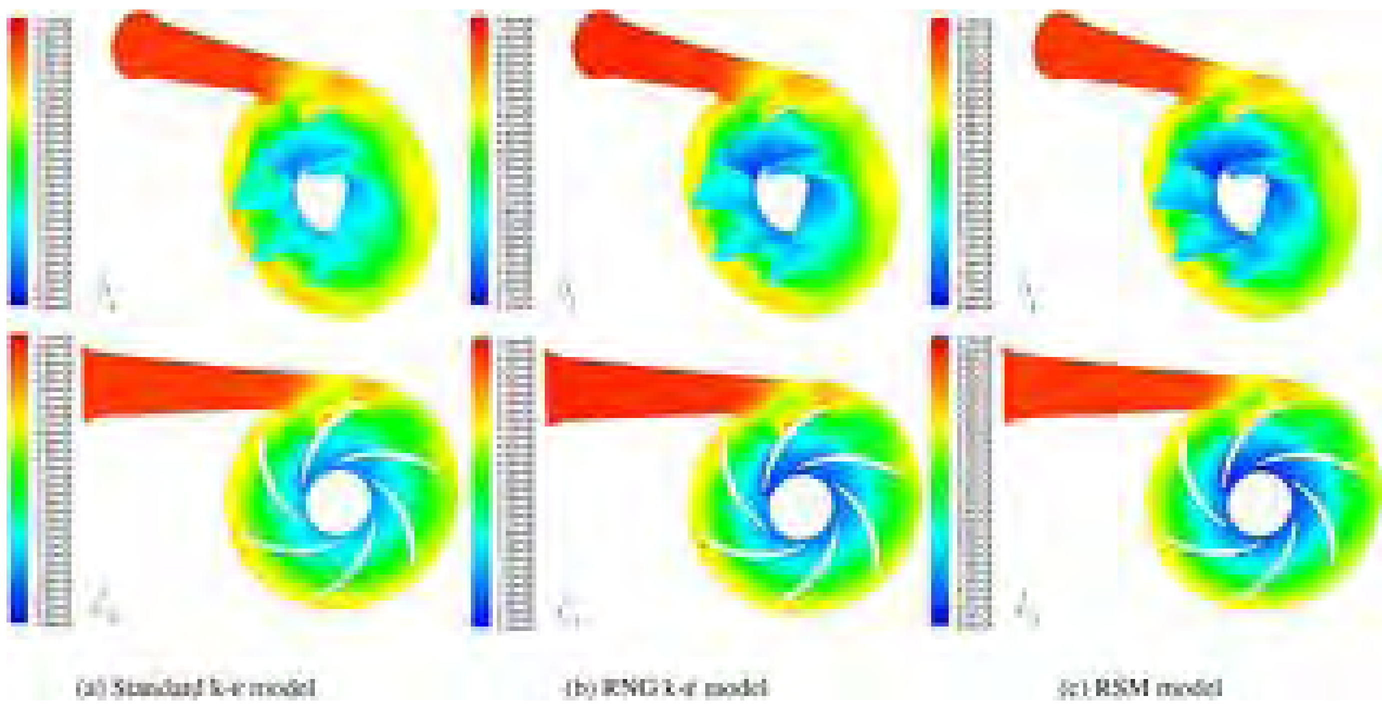
Examples of CFD Applications

- Design of vessels & towers internals



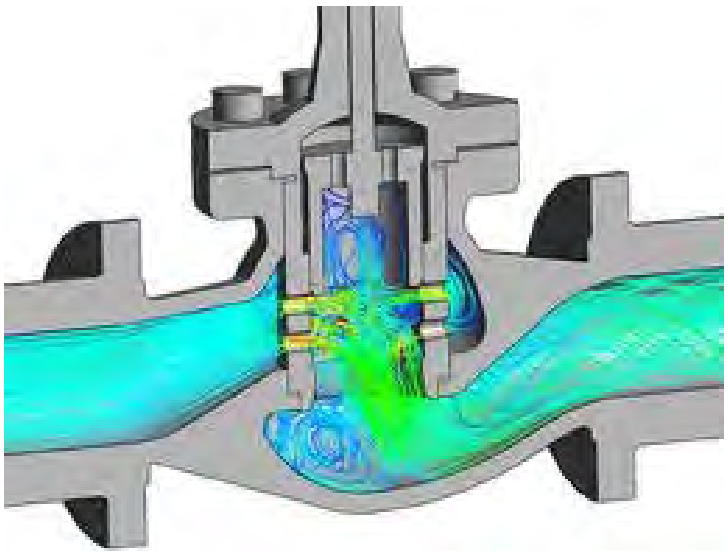
Examples of CFD Applications

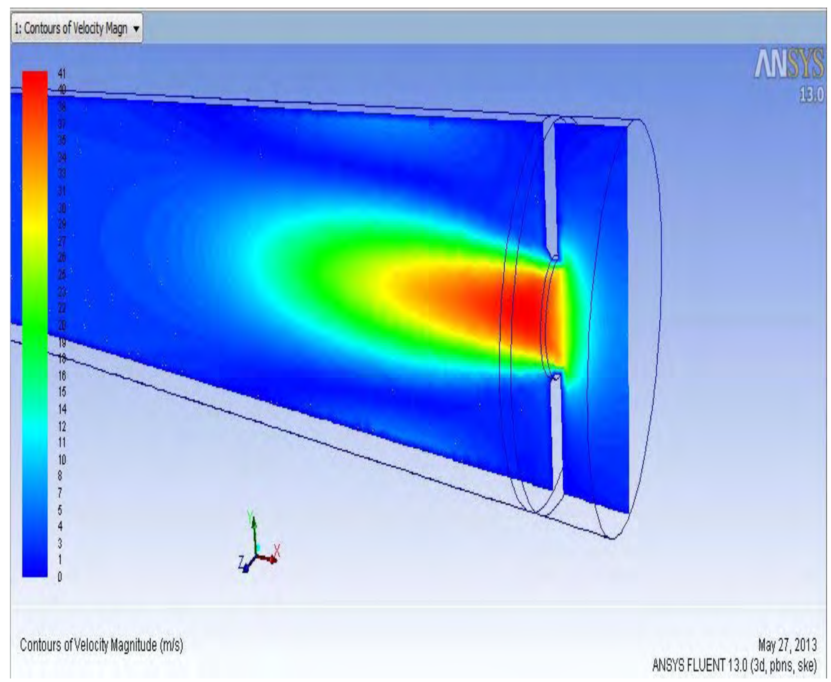
- Design of pumps



Examples of CFD Applications

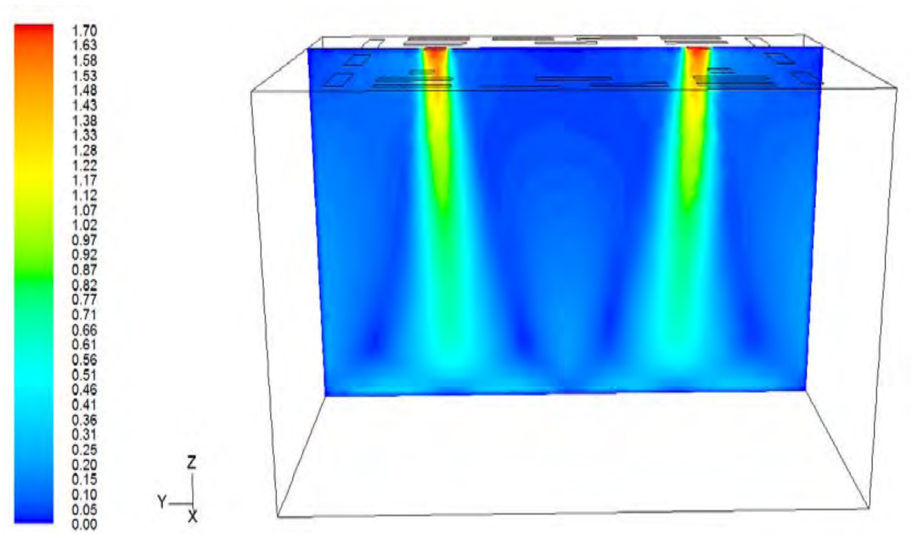
- Design of pipes, piping components, e.g. valves, orifice plates





Examples of CFD Applications

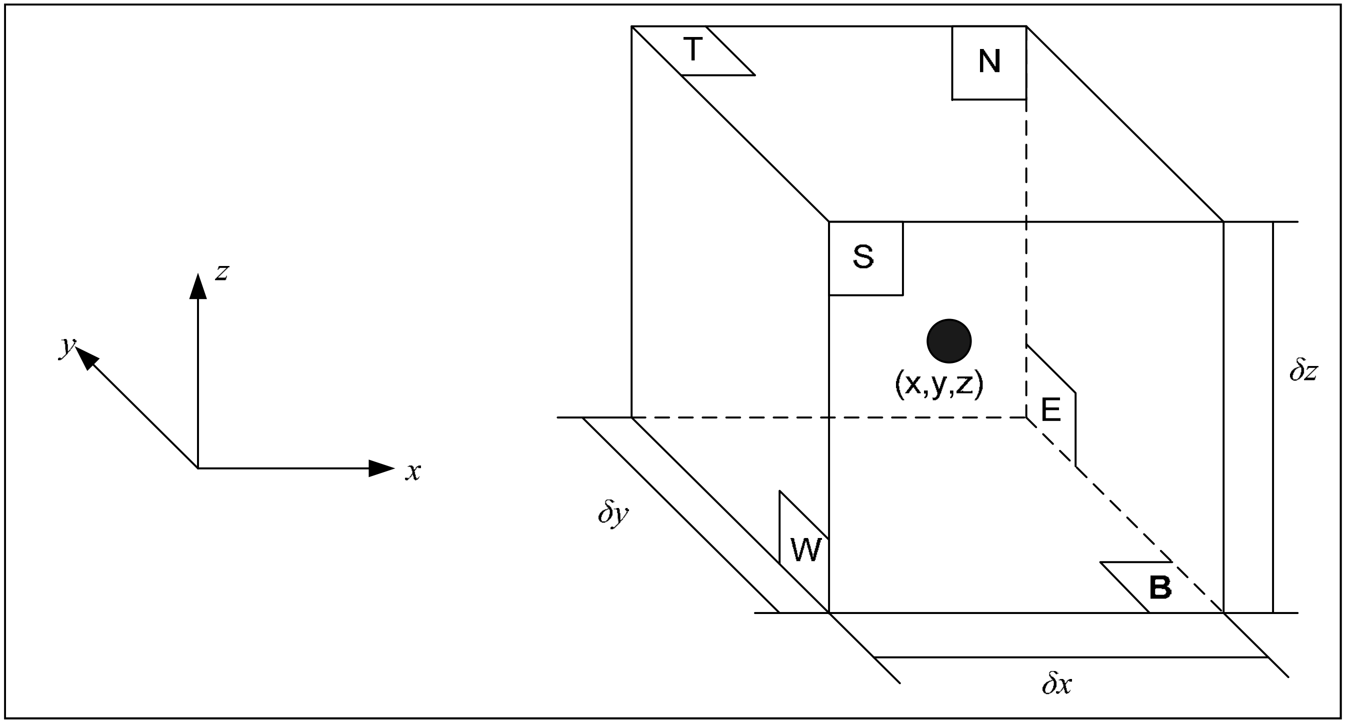
- Design of HVAC systems



Governing Equations

Governing Equations

Generally, governing equations are equations that describe the fluid motion inside the domain in full three-dimensional flow.



Fluid element for conservation laws

Governing Equations (Cont.)

Mass Conservation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{\upsilon} \right) = S_m$$

Momentum Conservation

$$\frac{\partial}{\partial t}\left(\rho \vec{\upsilon} \right) + \nabla \cdot \left(\rho \vec{\upsilon} \vec{\upsilon} \right) = -\nabla p + \nabla \cdot \left(\bar{\bar{\tau}} \right) + \rho \vec{g} + \vec{F}$$

$$\bar{\bar{\tau}} = \mu \left[\left(\nabla \vec{v} + \nabla \vec{v}^{T} \right) - \frac{2}{3} \nabla \cdot \vec{v} I \right]$$

Governing Equations (Cont.)

Energy Conservation

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{\upsilon}(\rho E + p)) = \nabla \cdot \left(k_{eff} \nabla T - \sum_j h_j \vec{J}_j + (\tau_{eff} \cdot \vec{\upsilon}) \right) + S_h$$

$$E = h - \frac{p}{\rho} + \frac{v^2}{2}$$

Species Transport Equation

$$\frac{\partial}{\partial t}\left(\rho Y_i\right) + \nabla \cdot \left(\rho \vec{v} Y_i\right) = -\nabla \cdot \vec{J}_i + R_i + S_i$$

Governing Equations (Cont.)

Turbulence Modeling

– ***k*-ε model**

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

- **Turbulent kinetic Energy (*k*)**

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k$$

- **Dissipation Rate Epsilon (ε)**

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

Verification of CFD Codes

“ARE WE SOLVING THE EQUATIONS RIGHT”?)

- **Examine the computer programming by visually checking the source code,**
- **Examine iterative convergence by monitoring the residuals, relative changes of integral quantities**
- **Examine consistency (check if relevant conservation principles are satisfied)**

Validation of CFD Models

Validation amounts to checking if the model itself is adequate for practical purposes

- **(loosely speaking, the question is:**

"ARE WE SOLVING THE RIGHT EQUATIONS"?)

- **Verify the code to make sure that the numerical solutions are correct.**
- **Compare the results with available experimental data (making a provision for measurement errors) to check if the reality is represented accurately enough.**
- **Perform sensitivity analysis and a parametric study to assess the inherent uncertainty due to the insufficient understanding of physical processes.**
- **Try using different models, geometry, and initial/boundary conditions.**

Available Commercial CFD Software

- **ANSYS FLUENT:** http://www.fluent.com
- **ANSYS CFX:** http://www.ansys.com
- **STAR-CD:** http://www.cd-adapco.com
- **FEMLAB:** http://www.comsol.com

CASE STUDIES

CASE STUDIES

1- The Use Of CFD to avoid cavitation in orifice plates

2- The Use Of CFD to predict flow regimes and thermal comfort in air conditioned space

Case Study 1

The Use Of CFD to avoid cavitation in orifice plates

The Use Of CFD to avoid cavitation in orifice plates

- Fluid cavitation due to sharp pressure drop was thought as the prime reason.
- CFD analysis has been made to investigate flow behaviour and effect of change of orifice configuration on enhancement of flow and eliminating cavitation.

Orifice Cavitation

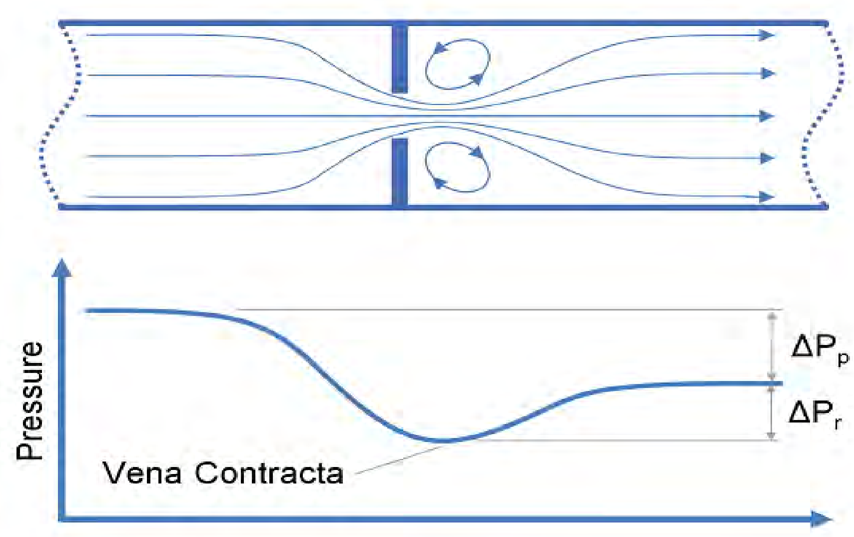
In liquid systems, cavitation occurs due to the swift formation and collapse of vapour bubbles, which causes the releases substantial energy and therefore leads to considerable noise and perhaps vibration due to interface with pipe structure.

Orifice Cavitation

In liquid systems, cavitation occurs due to the swift formation and collapse of vapour bubbles, which causes the releases substantial energy and therefore leads to considerable noise and perhaps vibration due to interface with pipe structure.

Orifice Cavitation

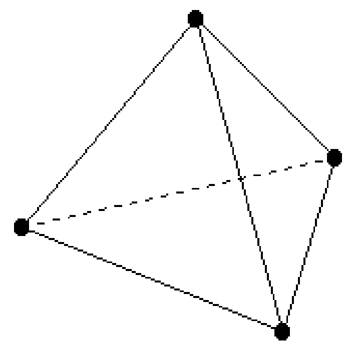
For a simple restriction orifice with one hole in the centre. As fluid passes through the hole, velocity increases; maximum velocity is reached soon after orifice hole resulting in what is called (Vena Contracta).



Meshed geometry

Geometry Construction and Grid Generation

- The geometry construction and grid generation during this study is carried out by utilizing GAMBIT® 2.3.16 program to perform uniform three-dimensional fine tetrahedral finite volumes mesh.



Tetrahedral cell

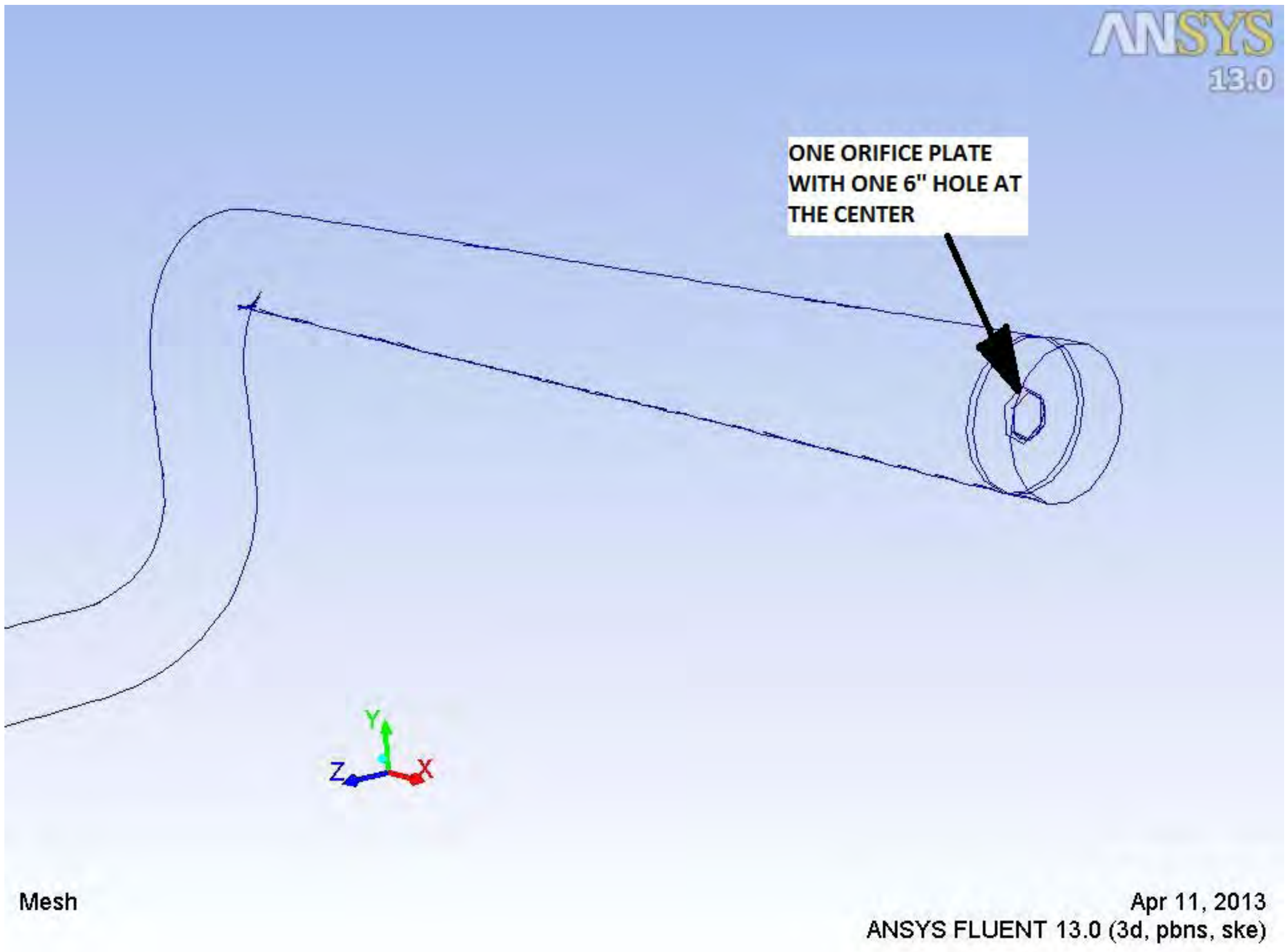
Then exported this meshing into FLUENT ® 13 program to solve the governing equations (after specifying the boundary, internal and initial conditions for the constructed geometry) then simulate the flow patterns inside the case studies.

THE USE OF CFD TO AVOID CAVITATION IN ORIFICE PLATES

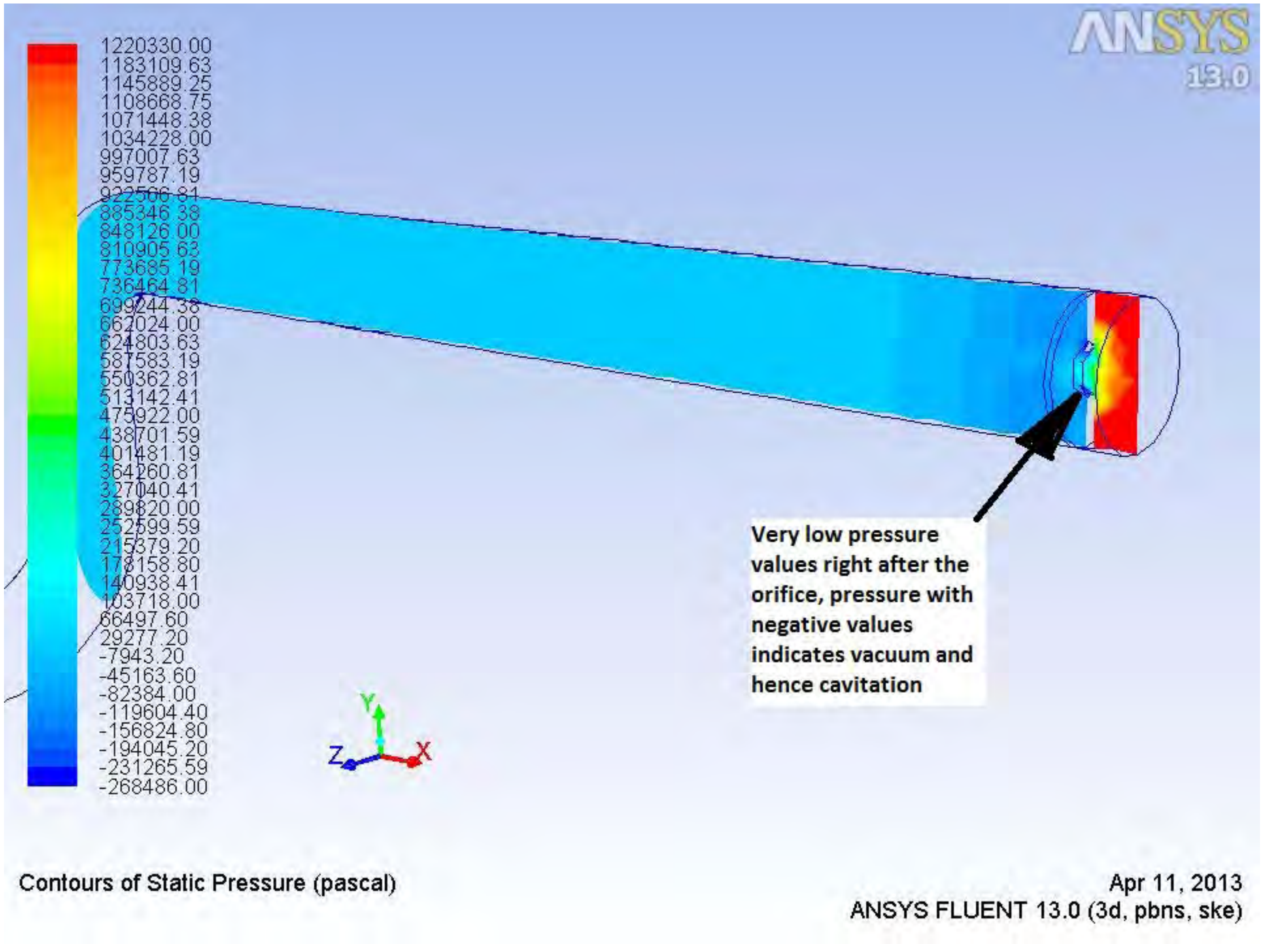
The present case study investigated three alternatives for orifice configurations:

- **Basic configuration: Single-hole single stage**
- **First design enhancement: Multi-perforated single stage**
- **Second design enhancement: Multi-perforated multi-stage orifice**

Original design (One single hole orifice plate)

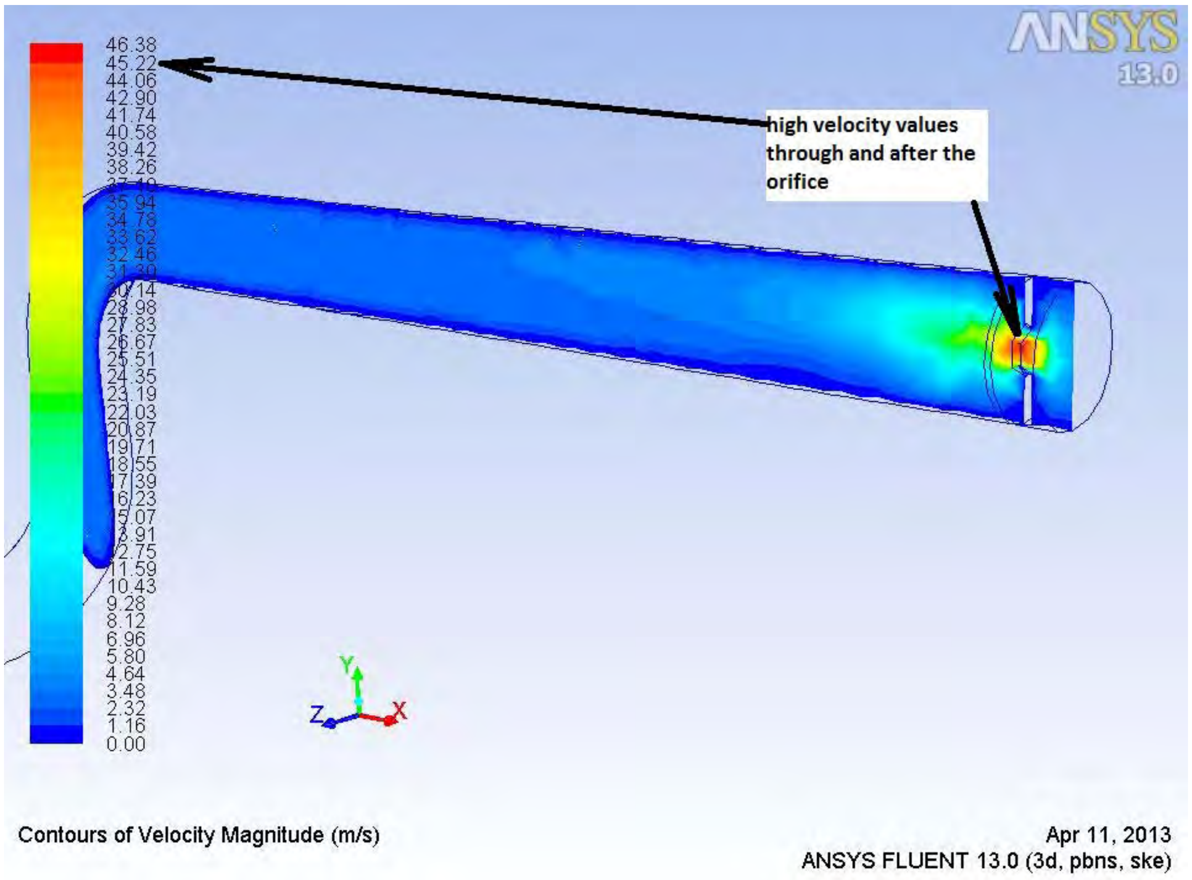


Original design (One single hole orifice plate)



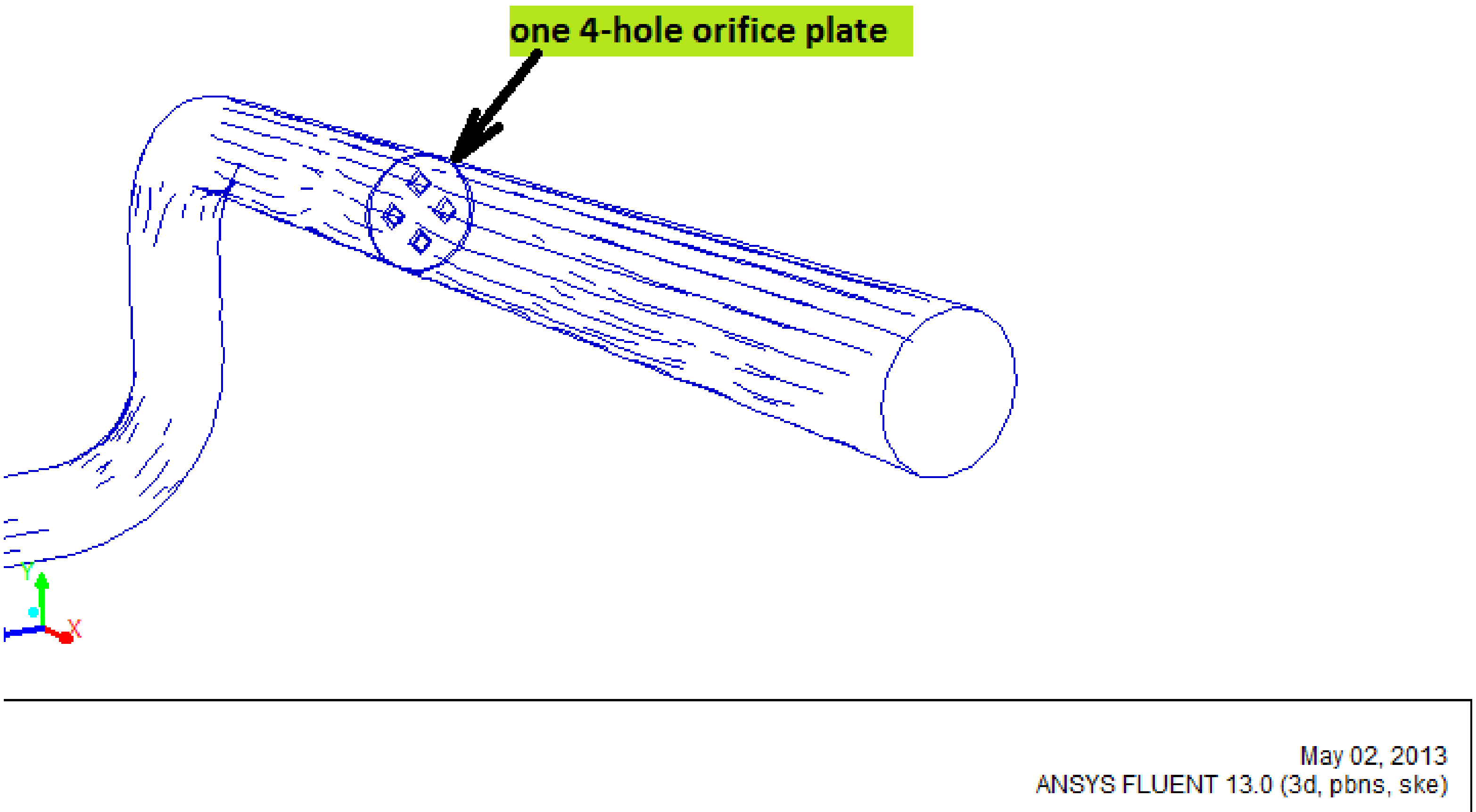
Pressure dist. For one single hole orifice plate

Original design (One single hole orifice plate)



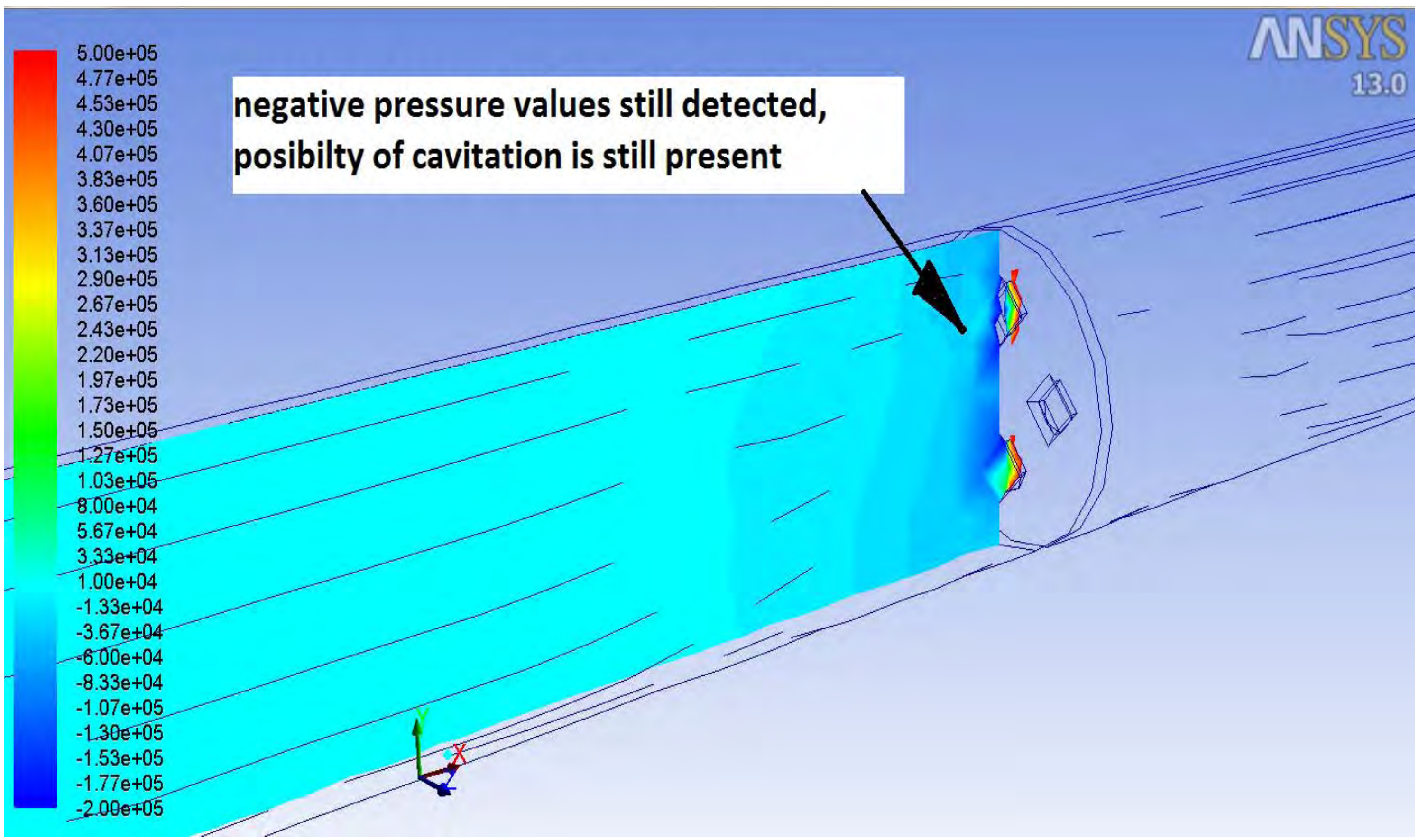
Velocity dist. For one single hole orifice plate

First design enhancement: Multi-perforated single stage



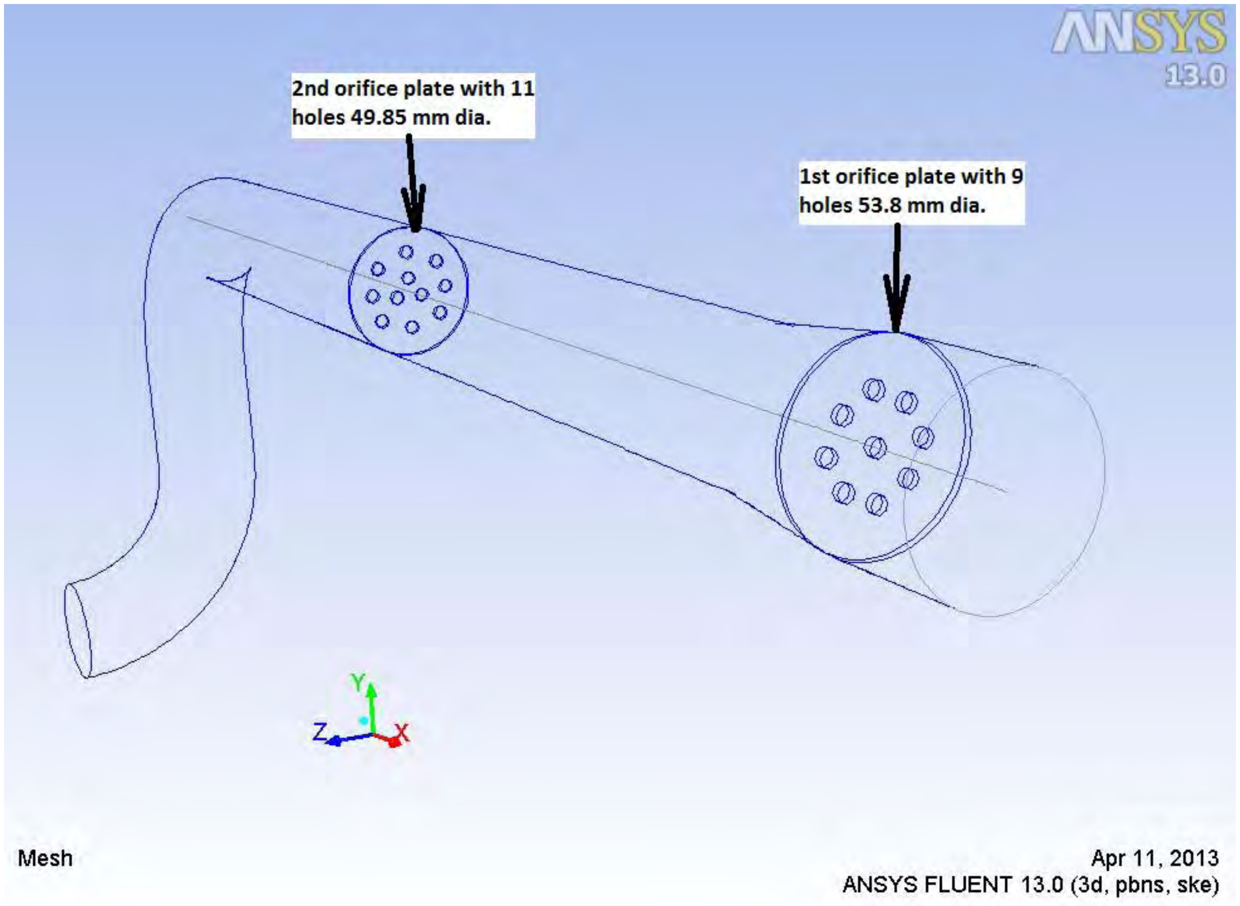
One 4 - hole orifice plate

First design enhancement: Multi-perforated single stage



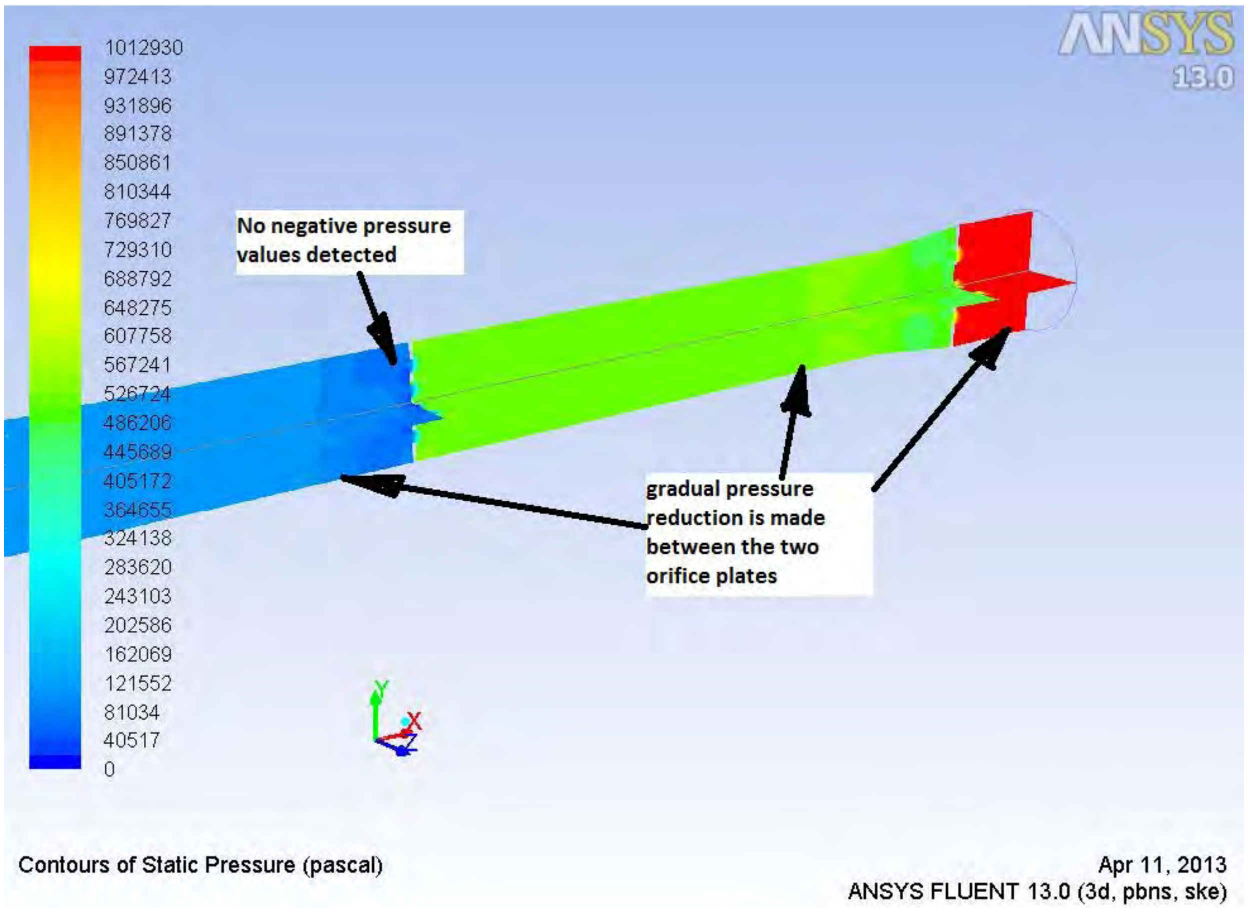
Pressure distribution for one 4 - hole orifice plate

Second design enhancement: Multi-perforated multi-stage



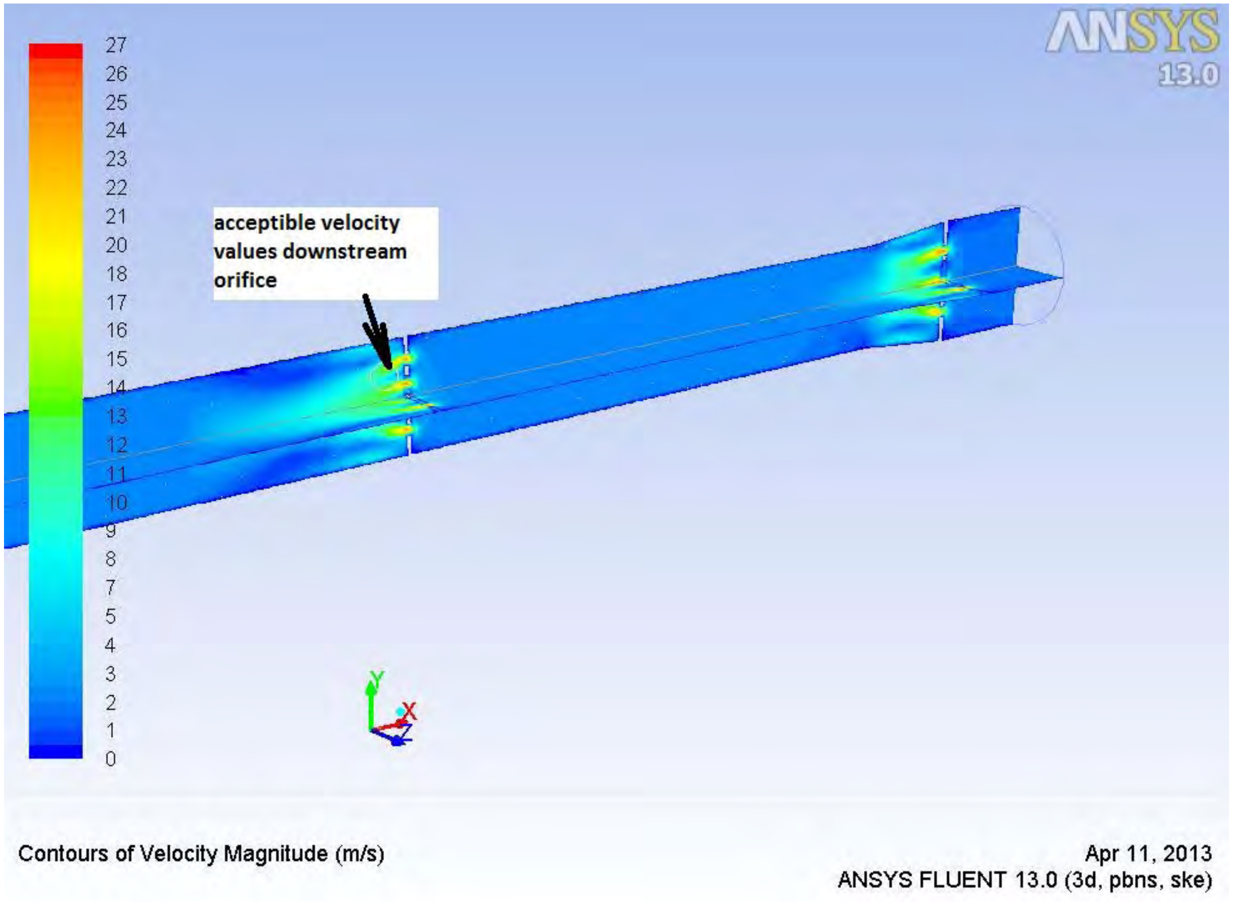
Two multi-hole orifice plates

Second design enhancement: Multi-perforated multi-stage



Pressure dist. for two multi-hole orifice plates

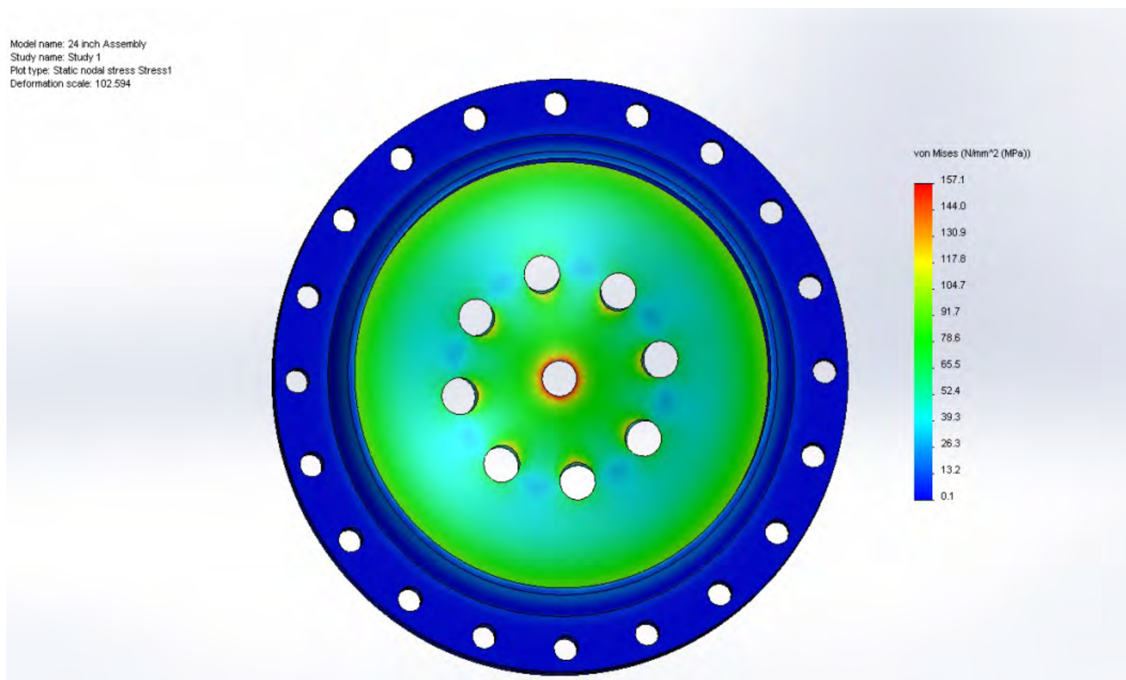
Second design enhancement: Multi-perforated multi-stage



Velocity dist. for two multi-hole orifice plates

Second design enhancement: Multi-perforated multi-stage

Furthermore, stress analysis, using finite element analysis FEA, has been made to ensure strength of multi-perforated orifices and their ability to hold induced load due to fluid pressure. FEA analysis was made using SolidWorks® Release 2013. Example to stress values contour is shown figure 6



Case Study 2

The Use Of CFD to predict flow regimes and thermal comfort in air conditioned space

MOTIVATION

- **To provide an idea to the most suitable decision can be taken by interested experts who intend to make design for air conditioning systems of squash arenas as well as other sporting arenas**

- **The decision is related to the selection of the location and distribution of air supply/return ports within area to be air conditioned**

Scope:

- Investigation of air flow patterns, thermal behaviours and carbon dioxide dispersion in an air-conditioned squash court;

Objective:

- The main targets during this work are:

- Satisfying the players/spectators’ thermal comfort conditions and,

- Improving the indoor air quality

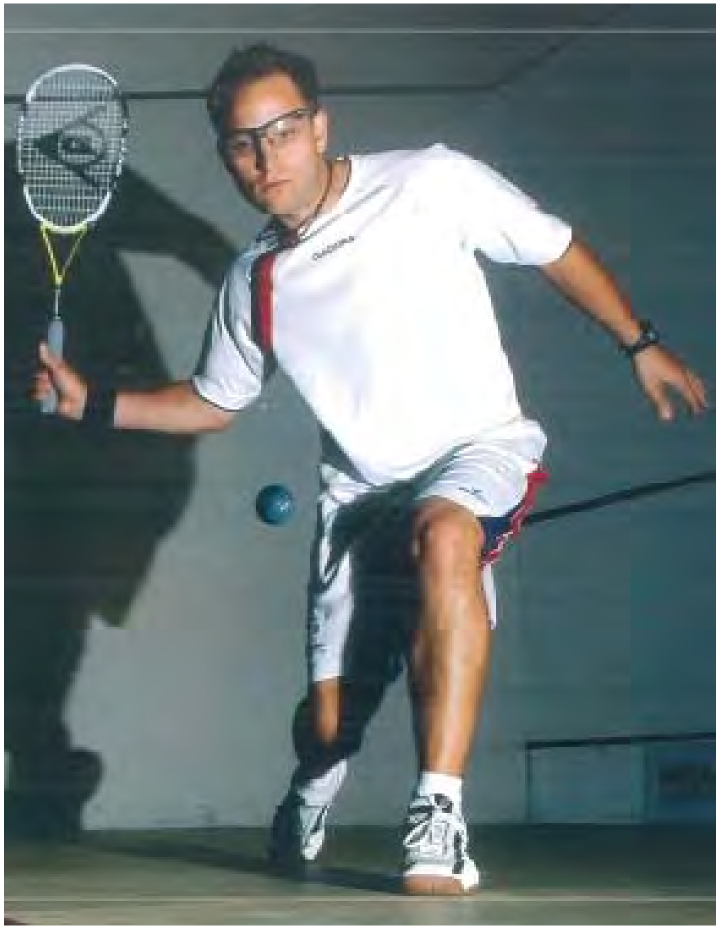
Comfort Air Conditioning

- The process of treating air to control simultaneously its TEMPERATURE, HUMIDITY, CLEANLINESS, and DISTRIBUTION to meet the comfort requirements of the occupants of the conditioned space

Factors Affecting Human Comfort:

PHYSICAL FACTORS

- Air temperature
- Relative humidity
- Mean radiant temperature
- Air velocity
- Air purity



PERSONAL FACTORS

- Activity level
- Clothing value

August 30th 2011

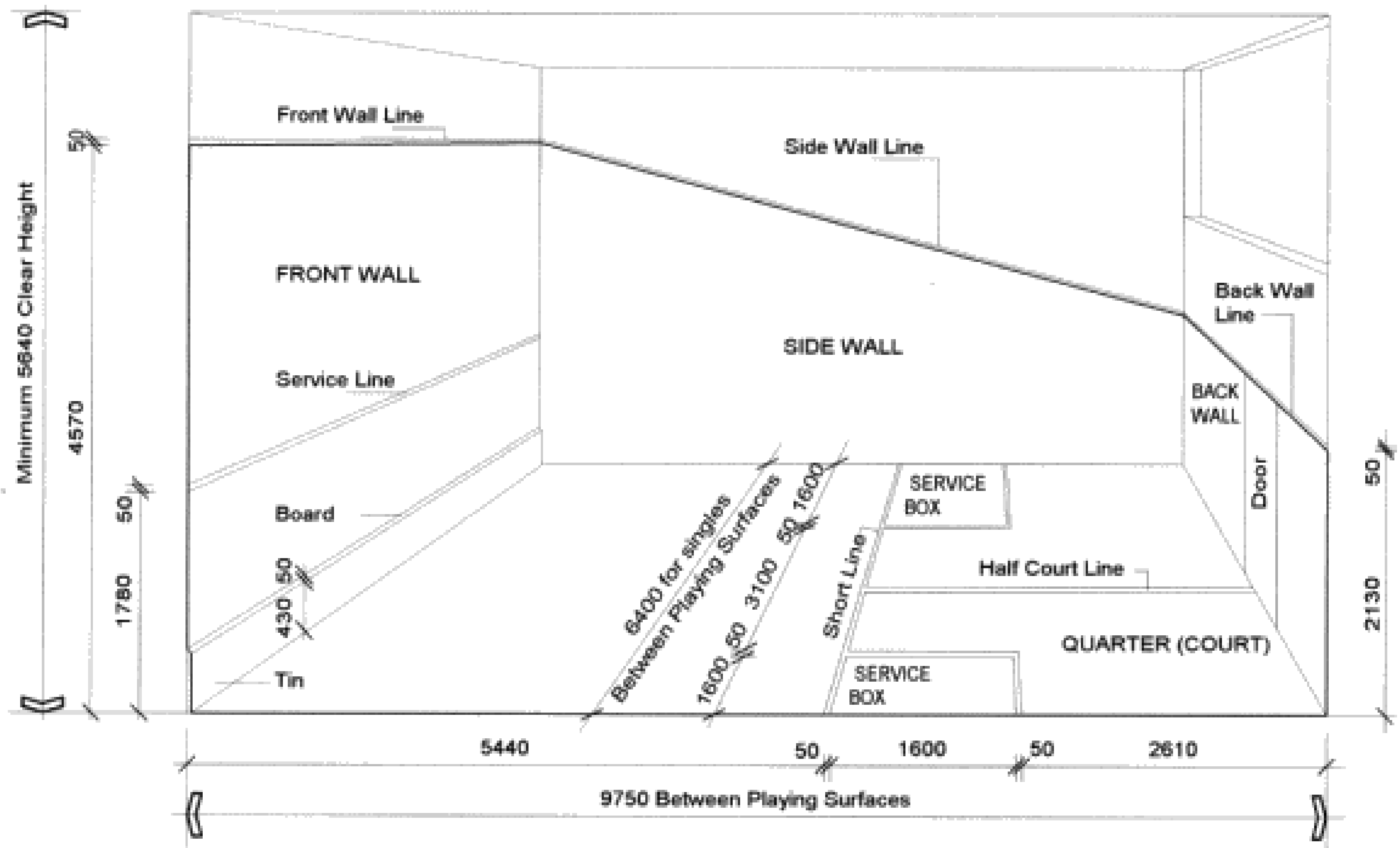
Air Properties Recommended values (WSF Recommendation)

Air Property	Recommended Range
Temperature °C [°F]	15 to 20 [59 to 68]
Relative Humidity %	Below 55
Air Velocity [m/s]	0.13 to 0.25

Model Description

Court Specifications

Standard Court Dimensions

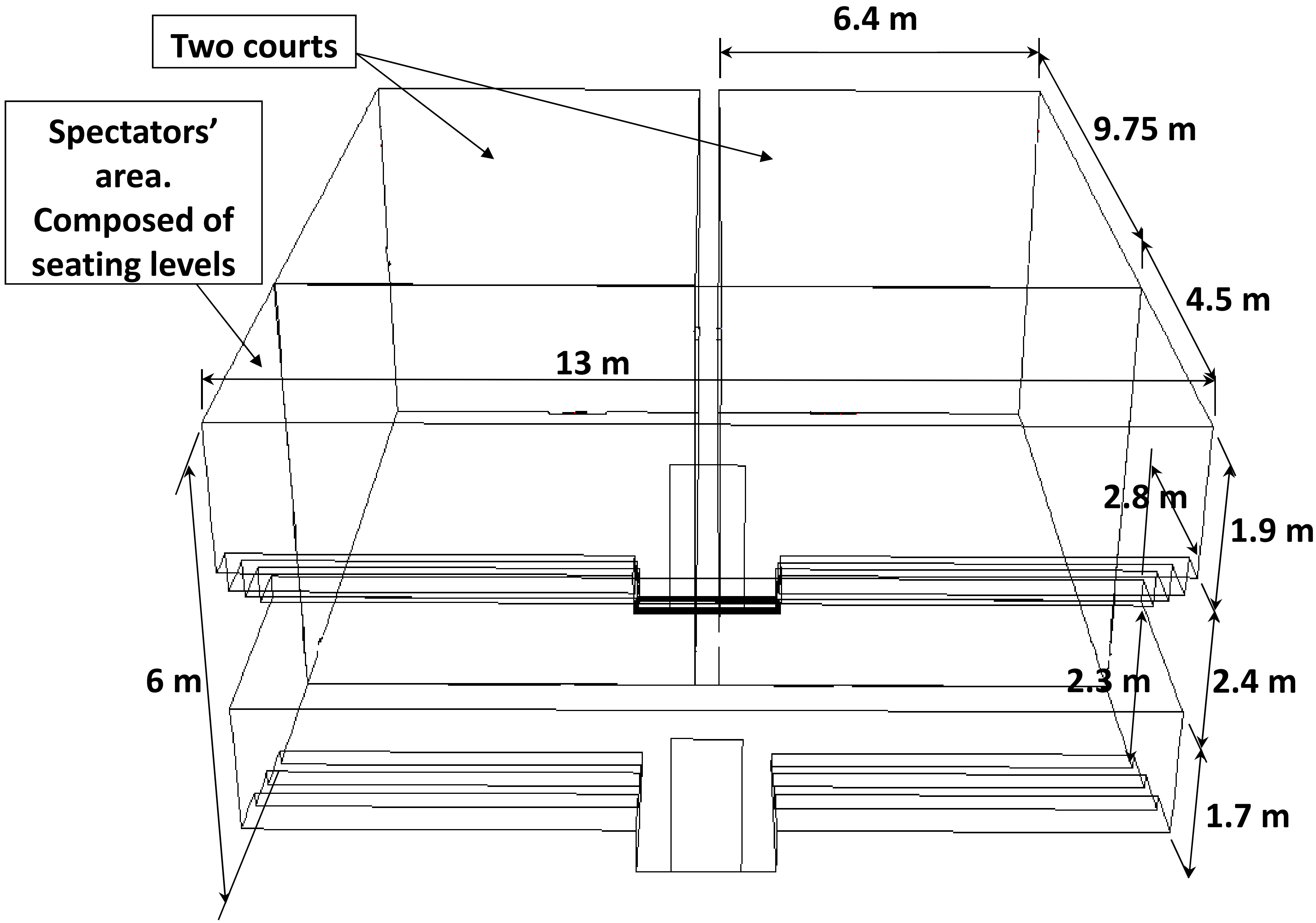


Squash arena under this study is composed of two zones

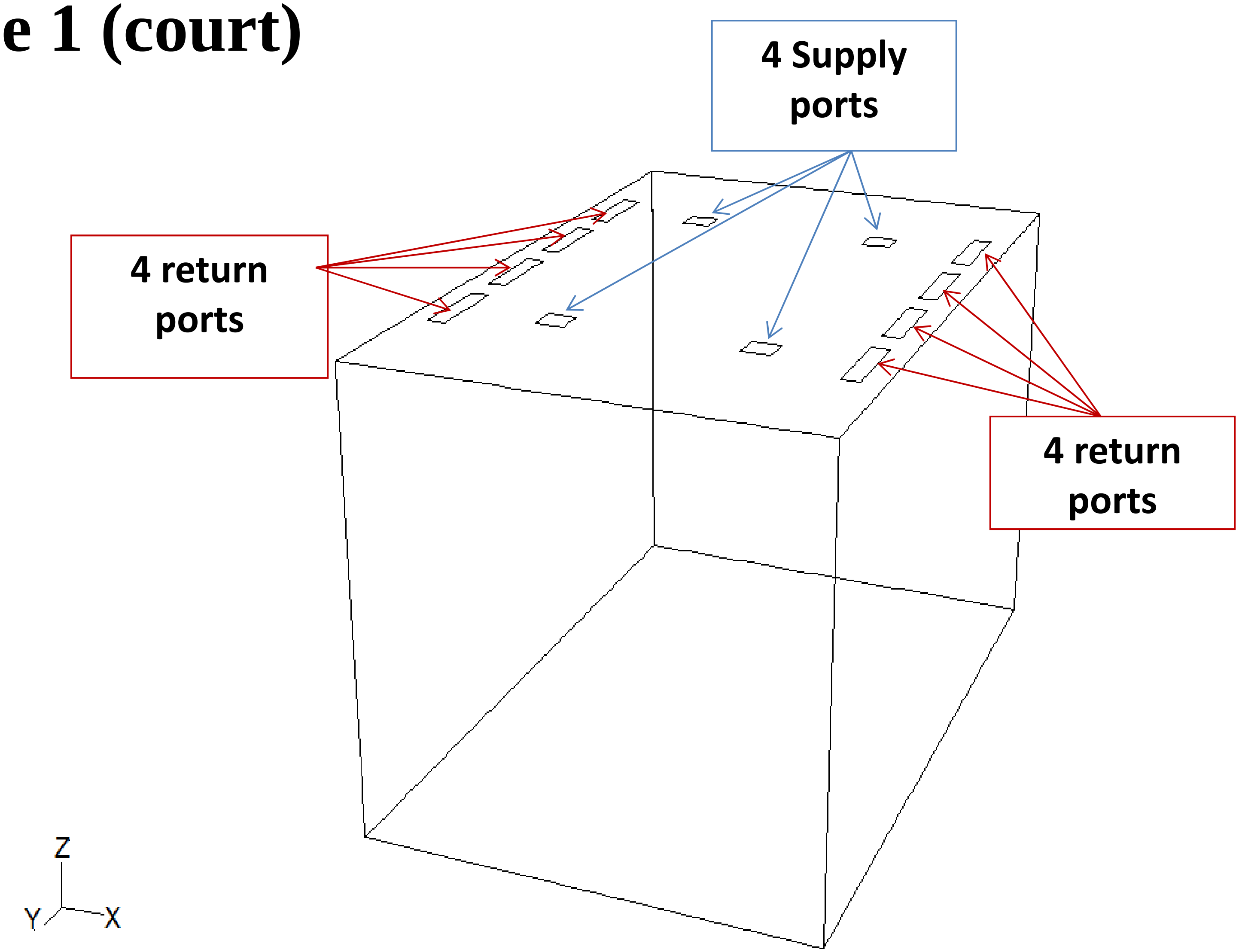
TWO COURTS

ONE COMMON SPECTATORS' AREA

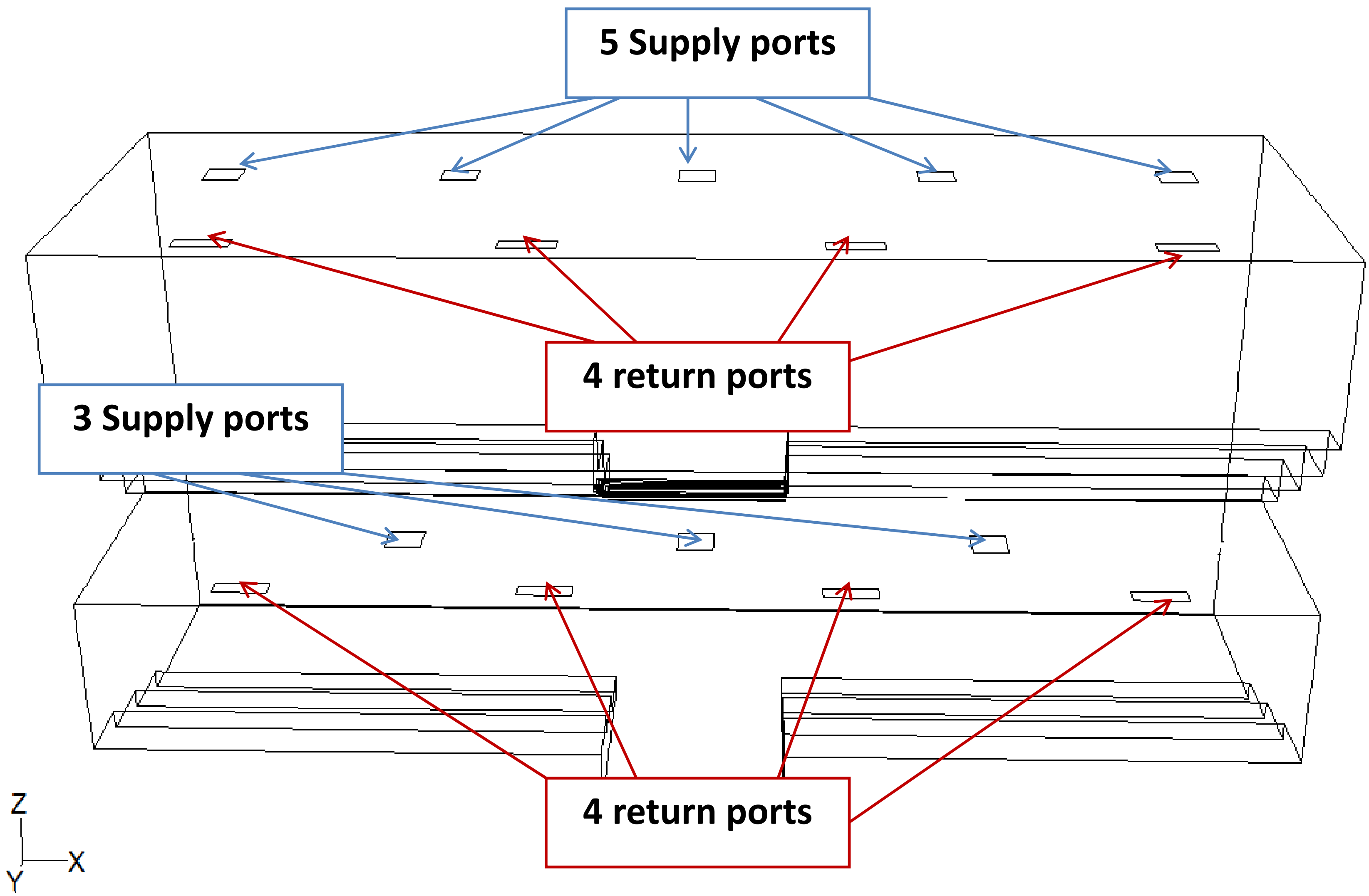
Case Studies Specifications



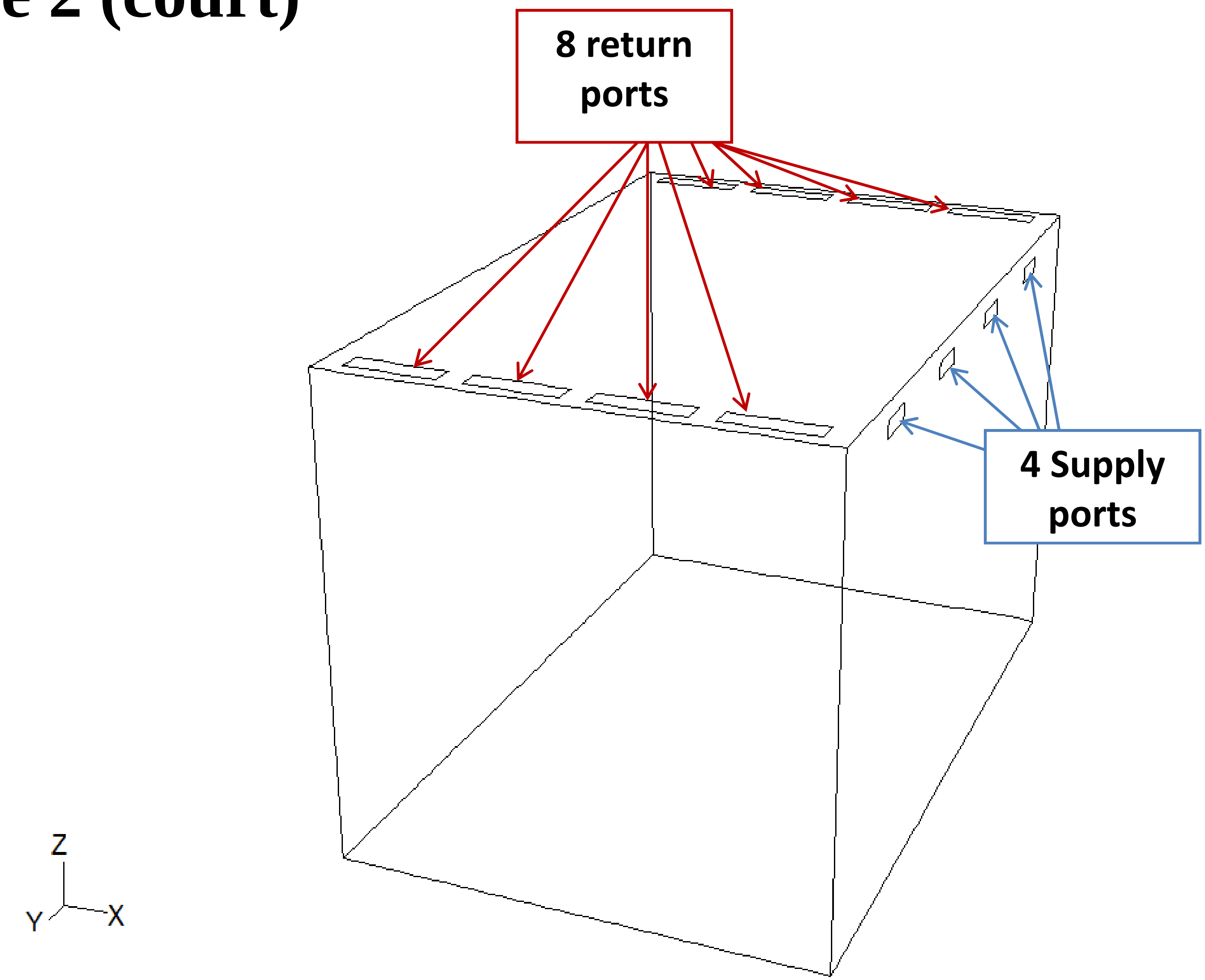
Case 1 (court)



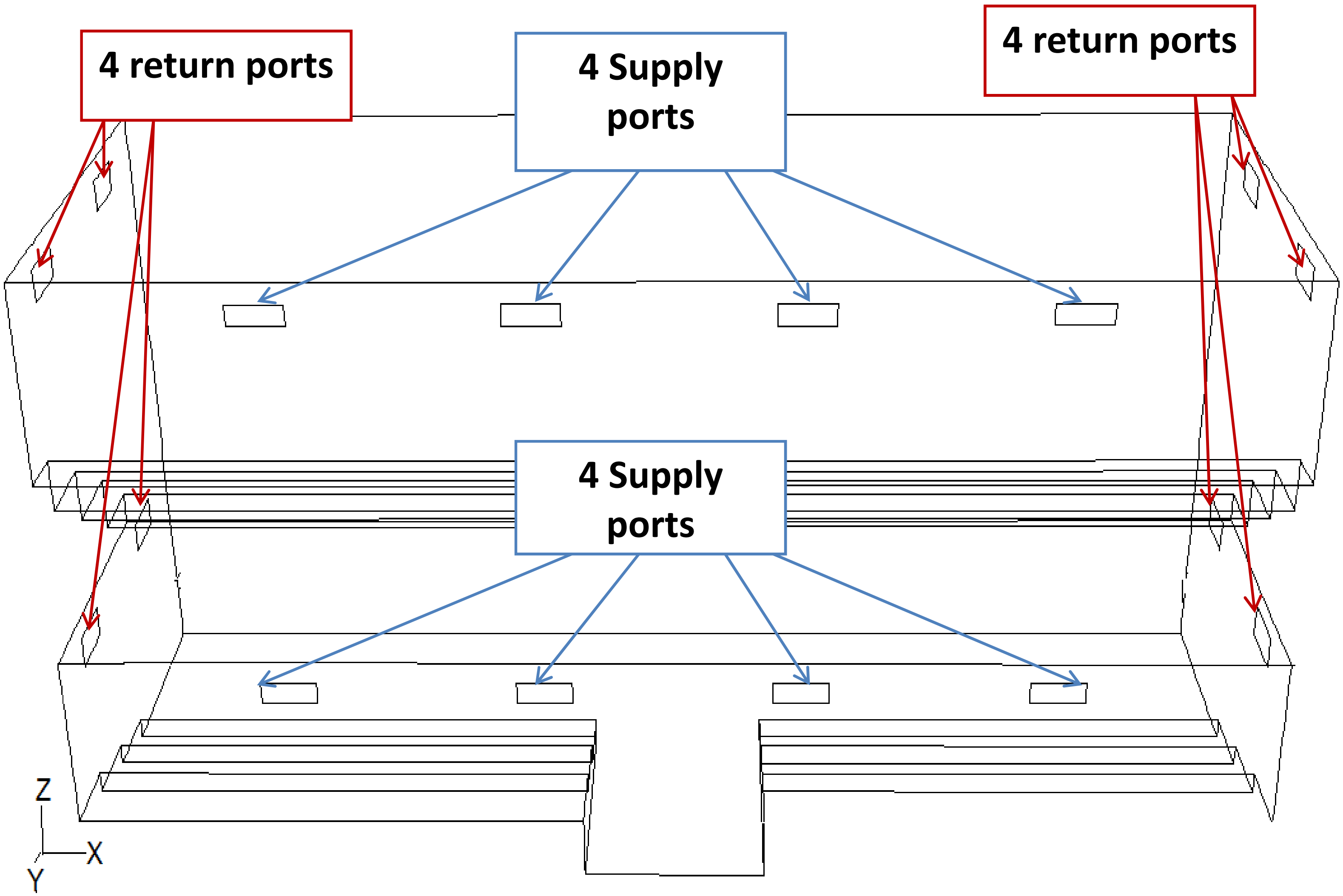
Case 1 (spectators' area)



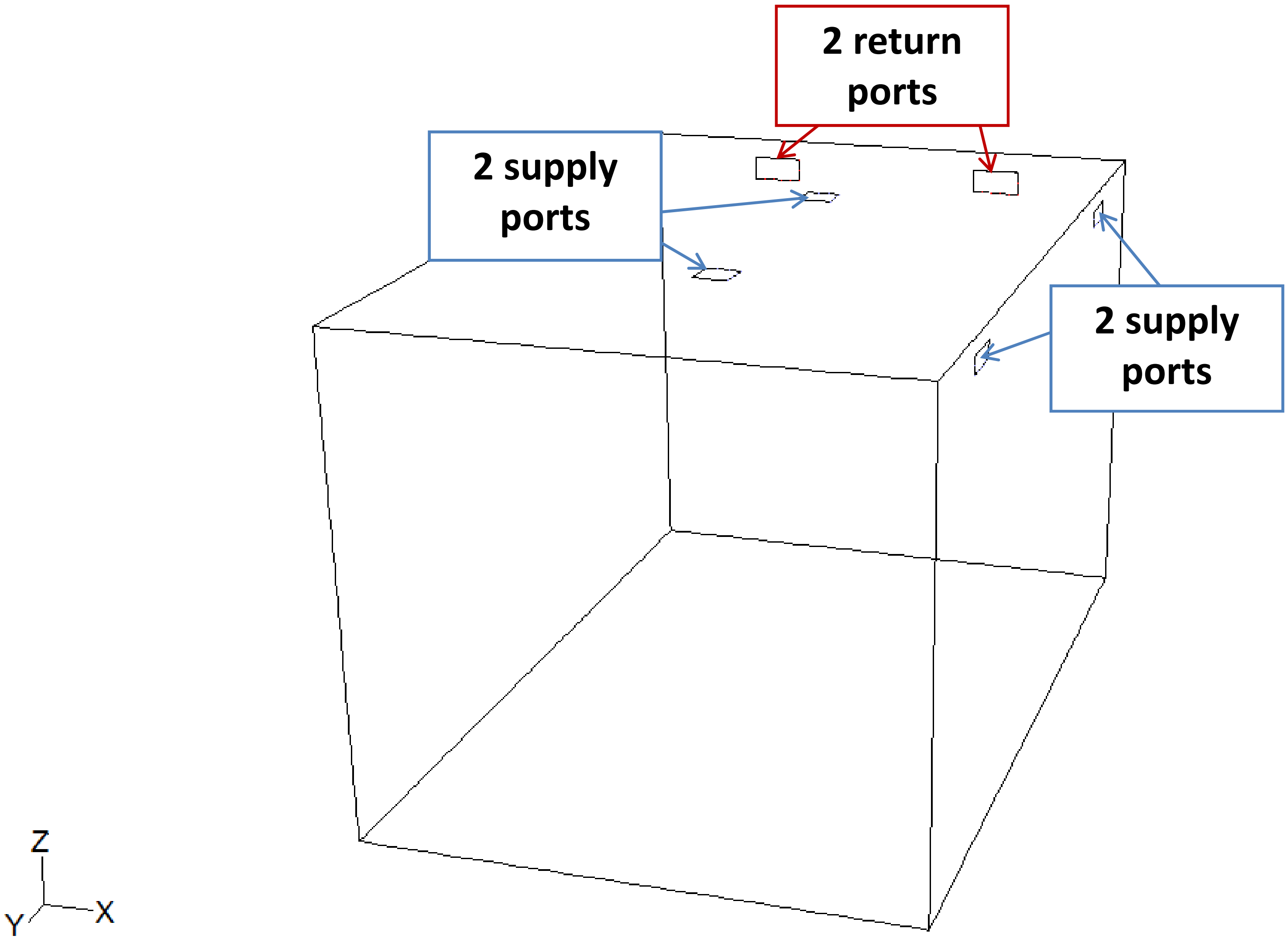
Case 2 (court)



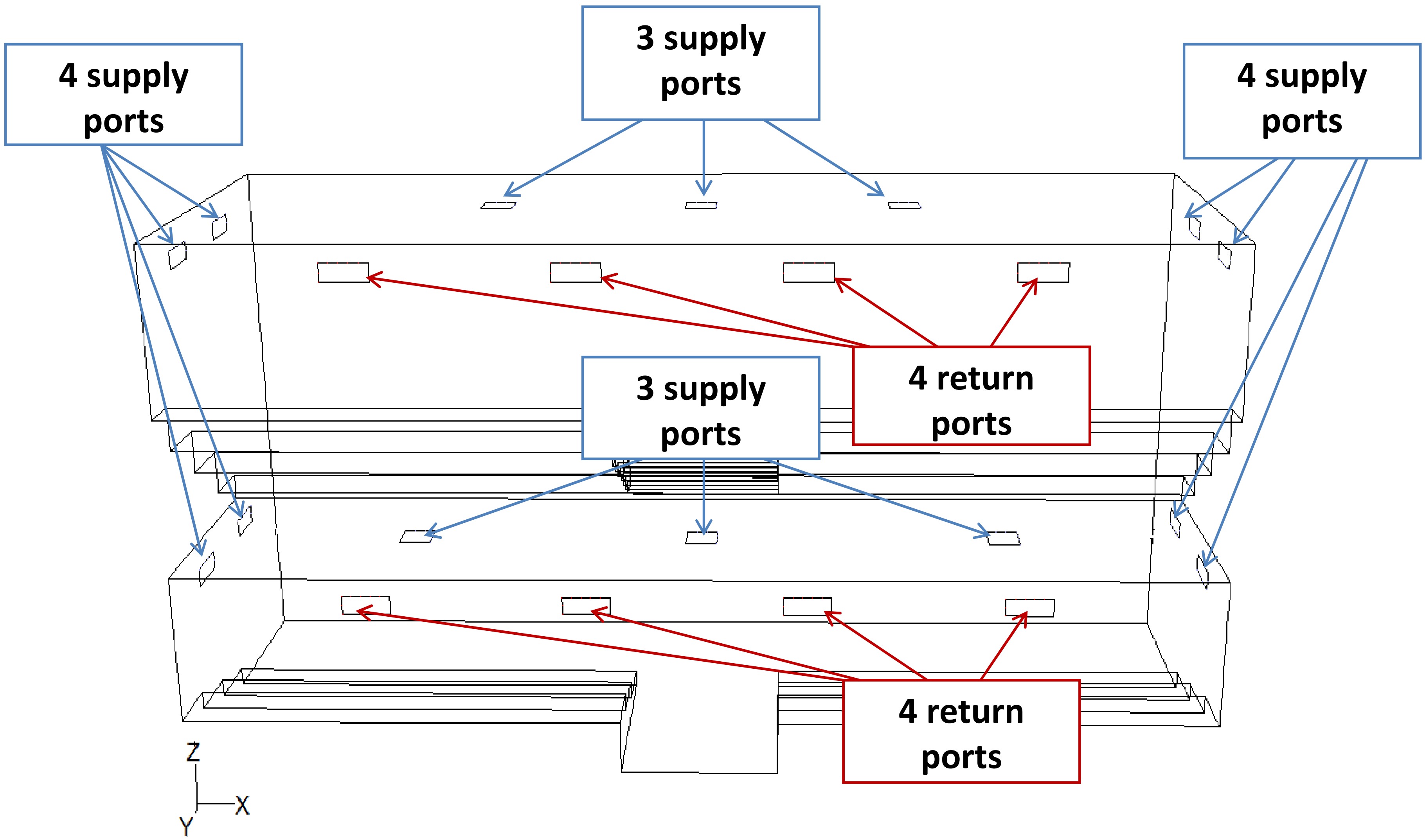
Case 2 (spectators' area)



Case 3 (court)



Case 3 (spectators' area)



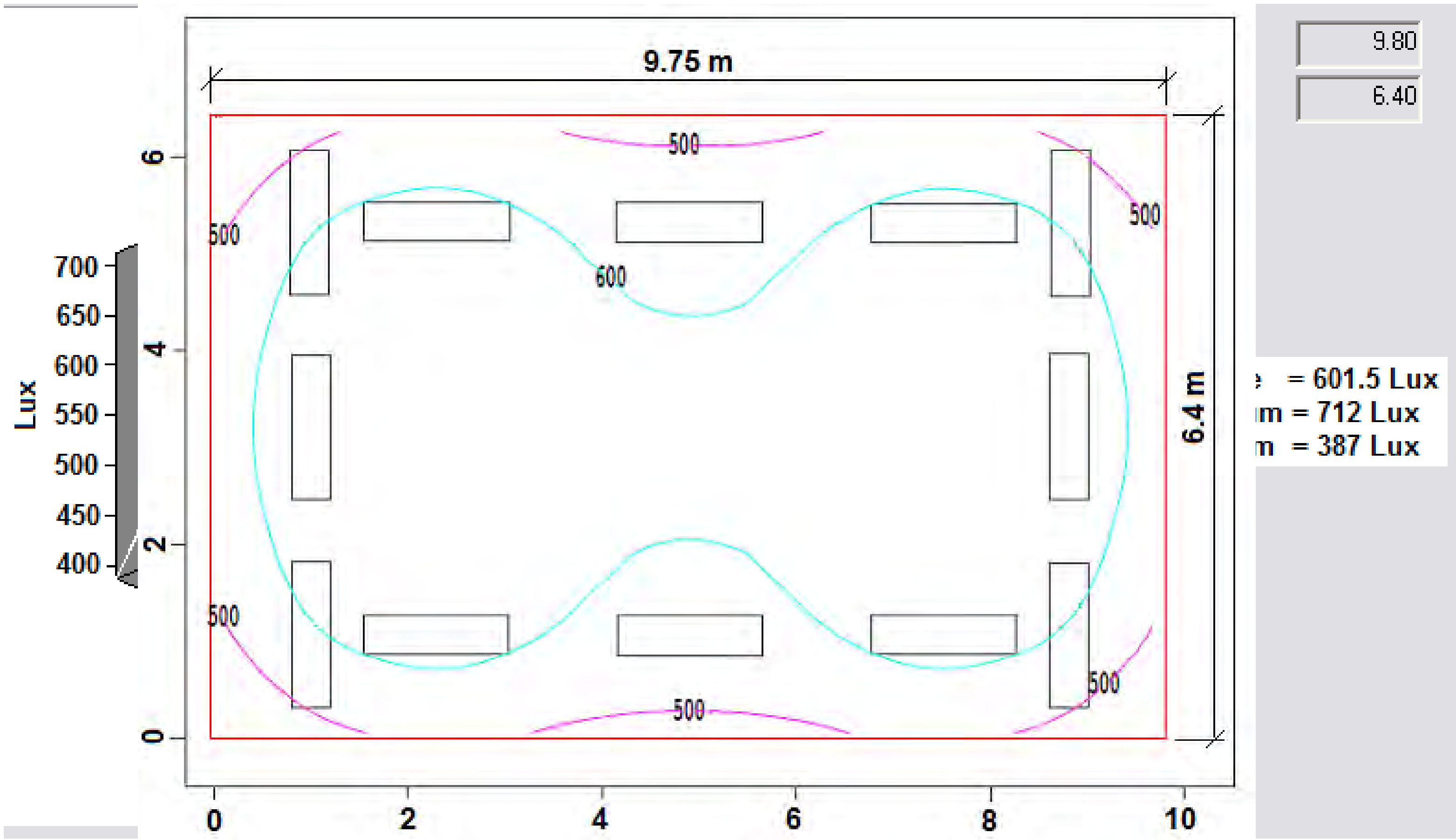
Boundary Conditions (case 1)

Inlet Air Conditions				Supply Description			Return Description		
Supply Ports	V_S [m/s]	T_S [K]	RH_S [%]	No. of Ports	Size [in]	Air Flow Rate [cfm]	Extract Ports	No. of Ports	Size [in]
S1 to S4	1.7	289	65	4	16 x16	550	R1 to R8	8	48 x 10
S9 to S13	2.6	289	60	5	16 x16	850	R5 to R12	8	24 x12
S14 to S16	1.2	289	60	3	16 x16	400			

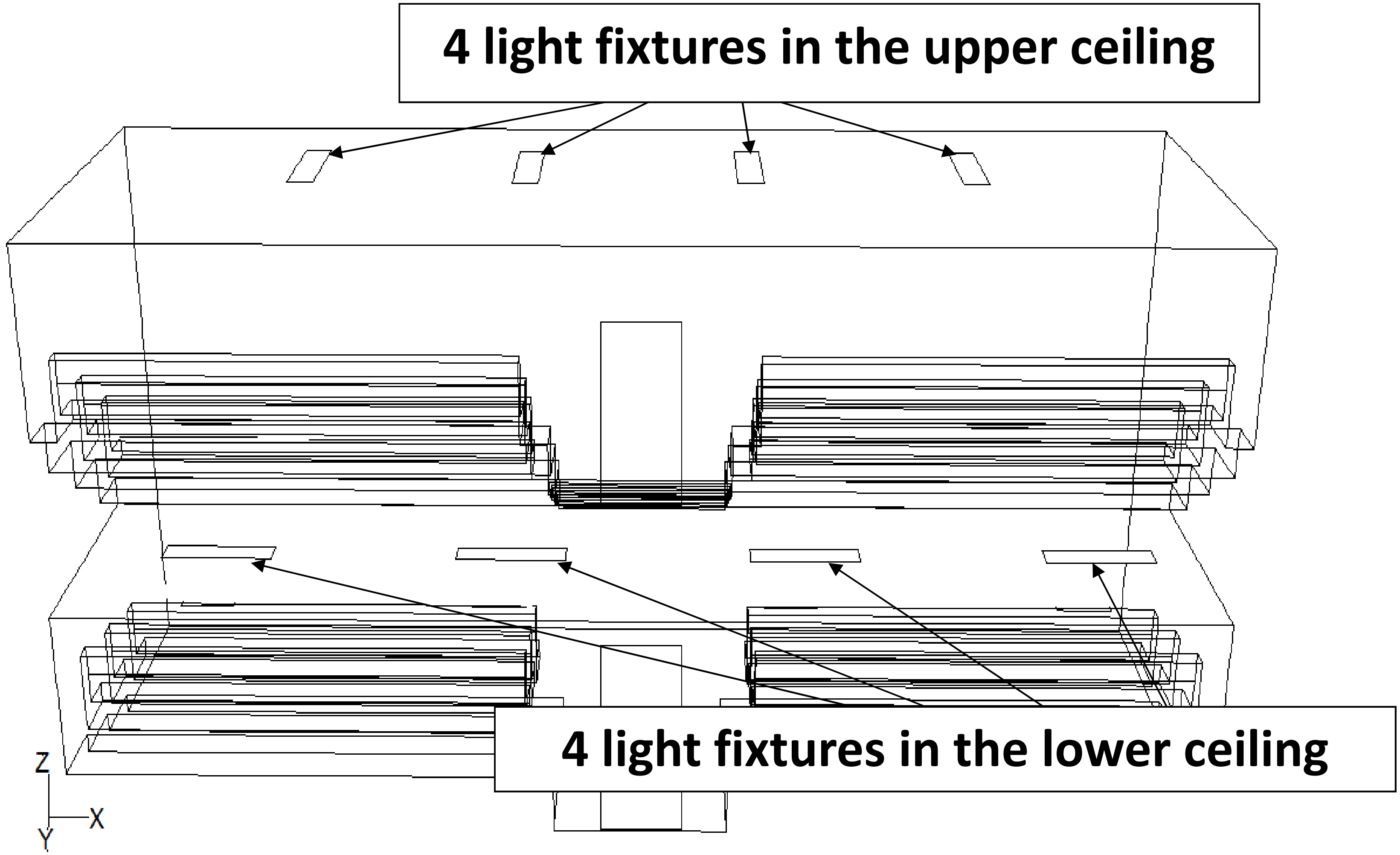
Lighting (courts)

Each court is lit by 12 boxes [1.20 m in length x 0.04 m in width] fixed in court ceiling, with 36 Watt fluorescent lights each.

Commercial software CHALMIT® Version 3.04 is used to determine the most suitable distribution of light fixtures inside courts



Lighting (spectators' area)



Lighting

Light Load per lighting box is as follows:

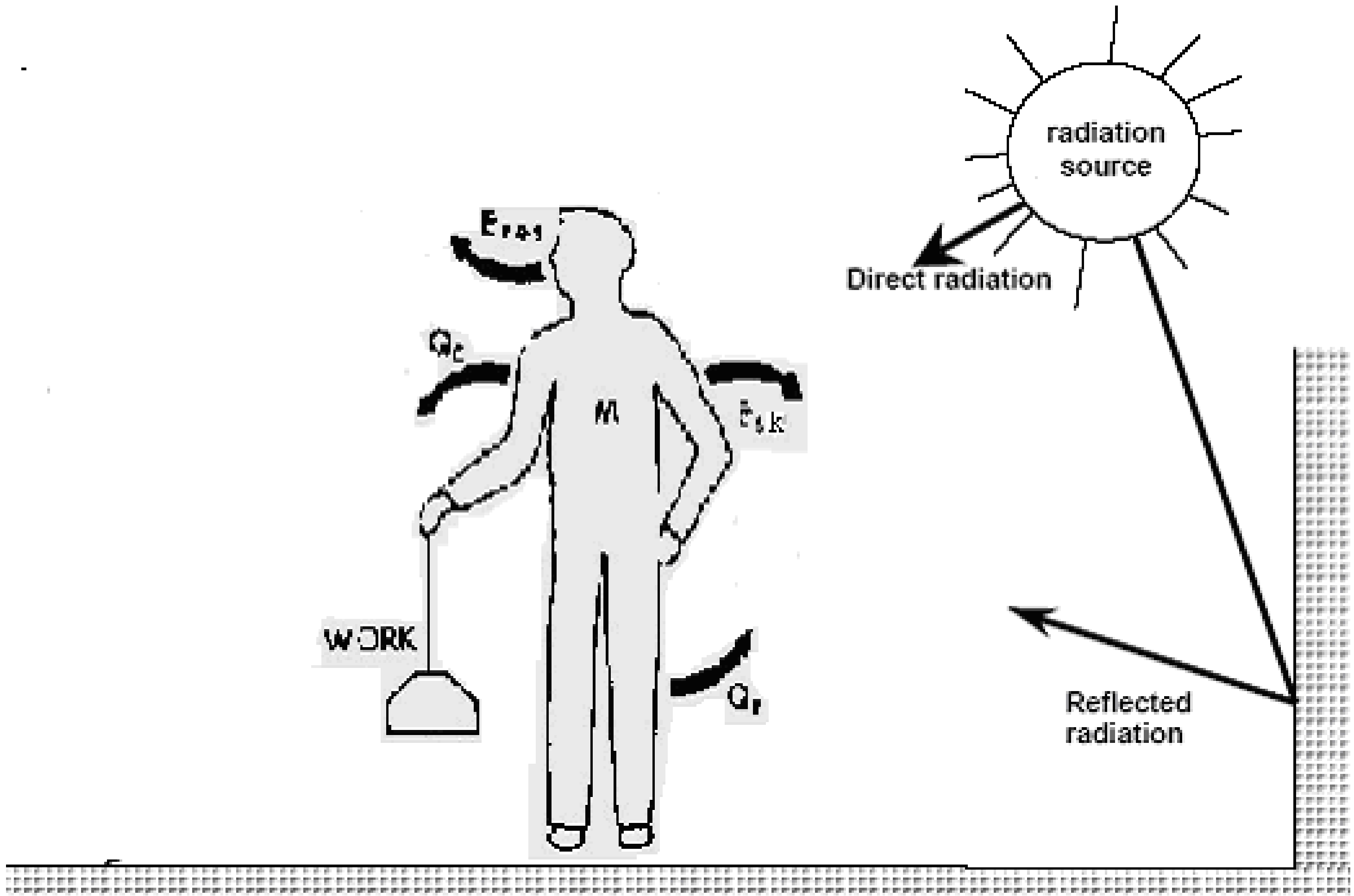
$$Q = 36\ [W/tube] \times 4\ [tubes/box] \times 1.2 \times 1$$
$$= 173\ W$$

To estimate light load Per box per unit area, the following equation is used:

$$Q/\ Box\ Area = 173/\ (1.2 \times 0.3) = 480\ W/m^2$$

EFFECT OF PEOPLE

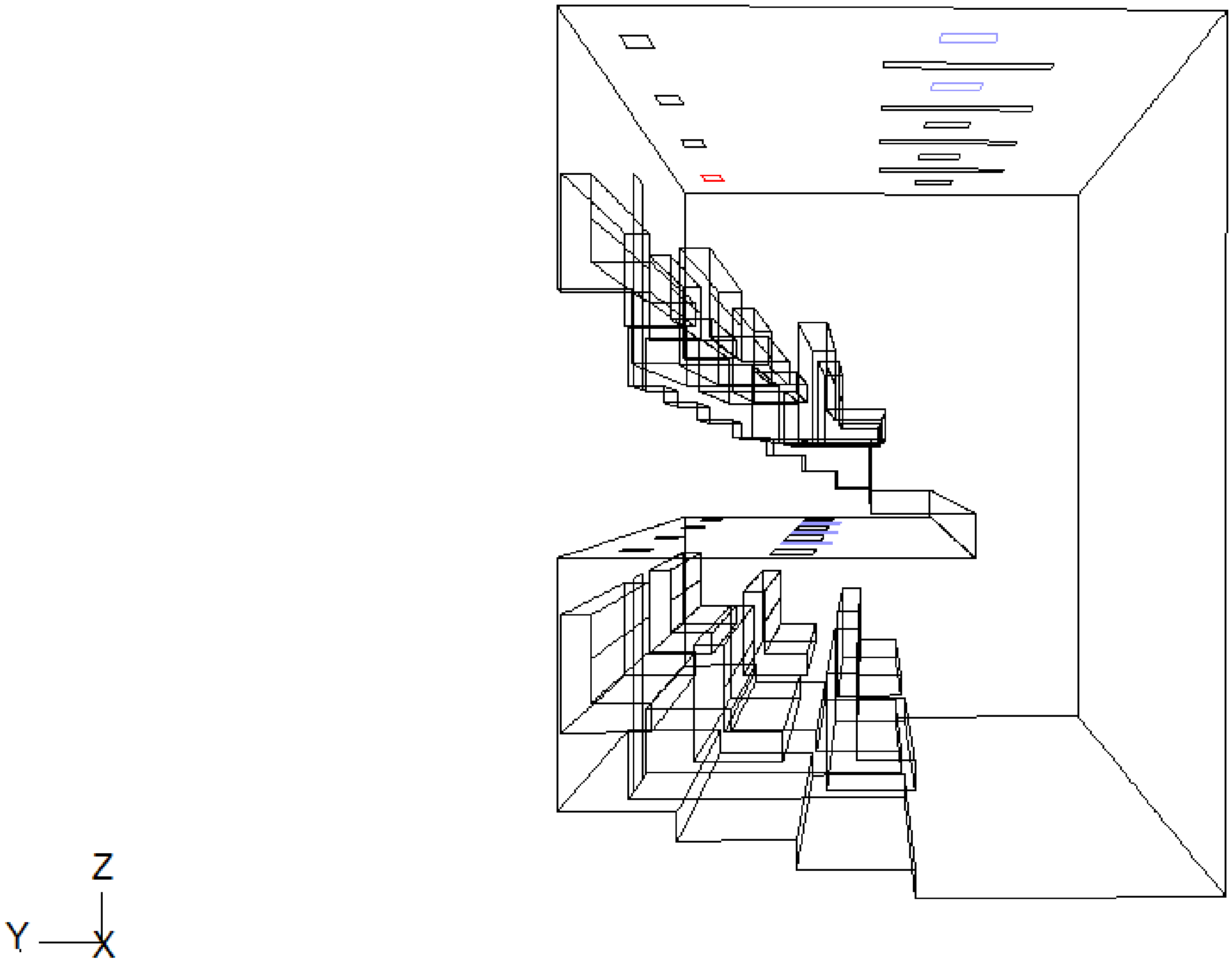
EFFECT OF PEOPLE



$$M = W + C + R + E_{SK} + E_{RES}$$

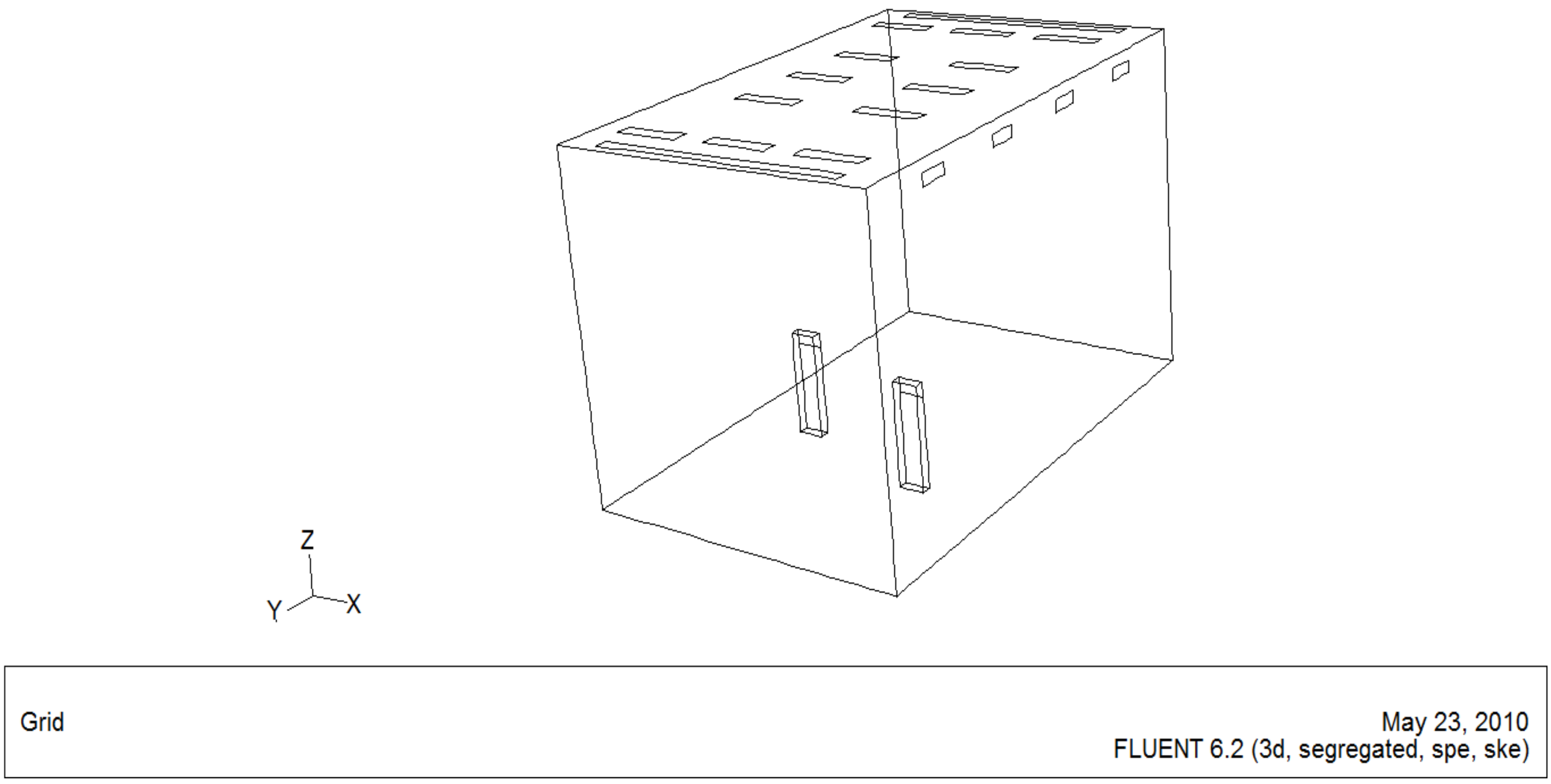
Effect Of People (Cont.)

SPECTATORS' MODEL



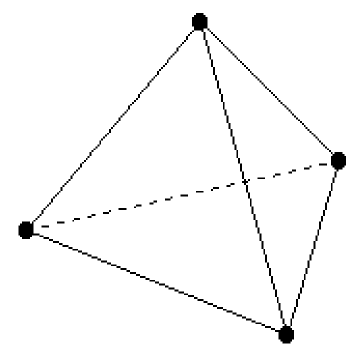
Effect Of People (Cont.)

PLAYERS' MODEL



Geometry Construction and Grid Generation

- The geometry construction and grid generation during this study is carried out by utilizing GAMBIT® 2.2 program to perform uniform three-dimensional fine tetrahedral finite volumes mesh.

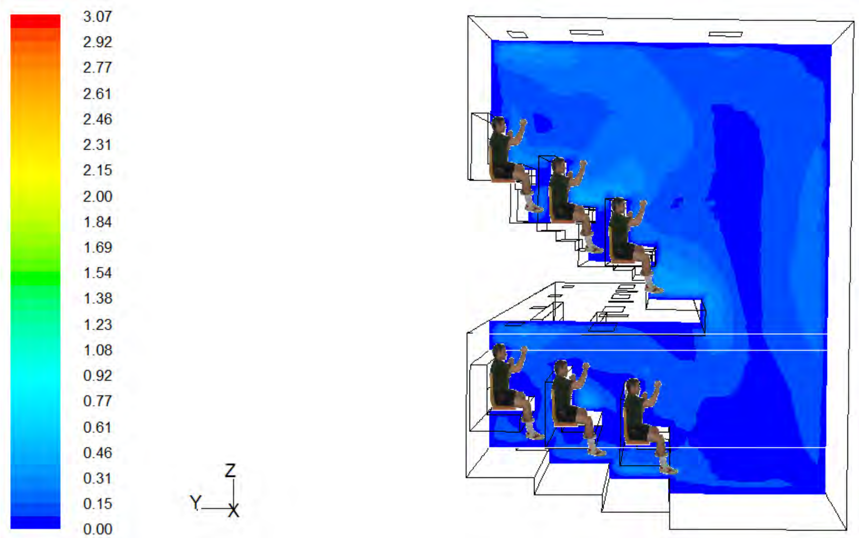


Tetrahedral cell

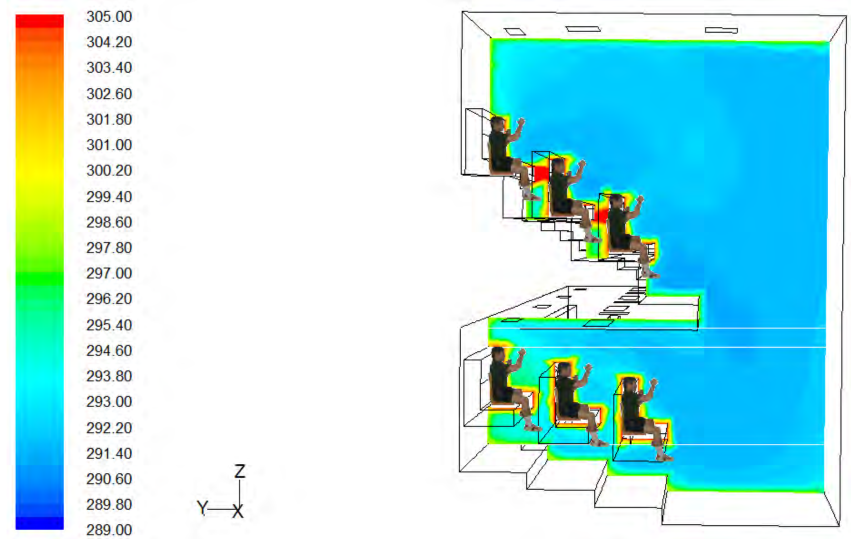
- Then exported this meshing into FLUENT ® 6.2 program to solve the governing equations (after specifying the boundary, internal and initial conditions for the constructed geometry) then simulate the flow patterns inside the case studies.

Results and Discussions

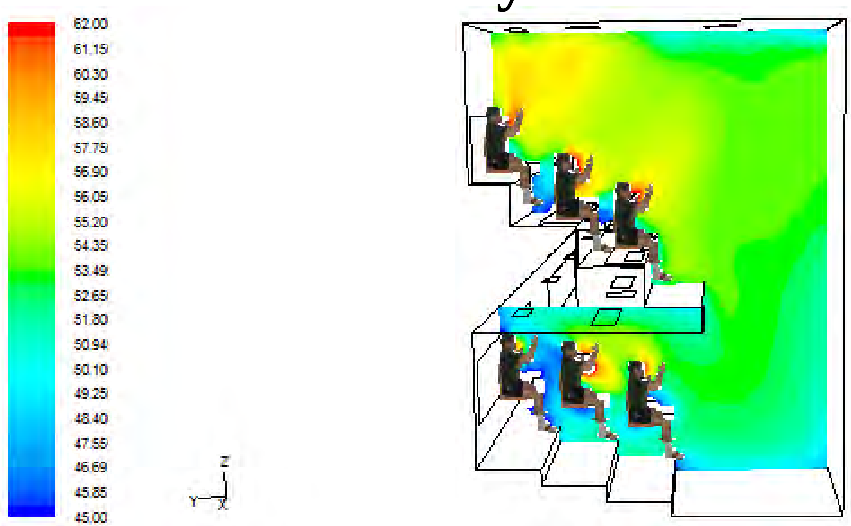
Ceiling Air Supply Design Case (Spectators' area)



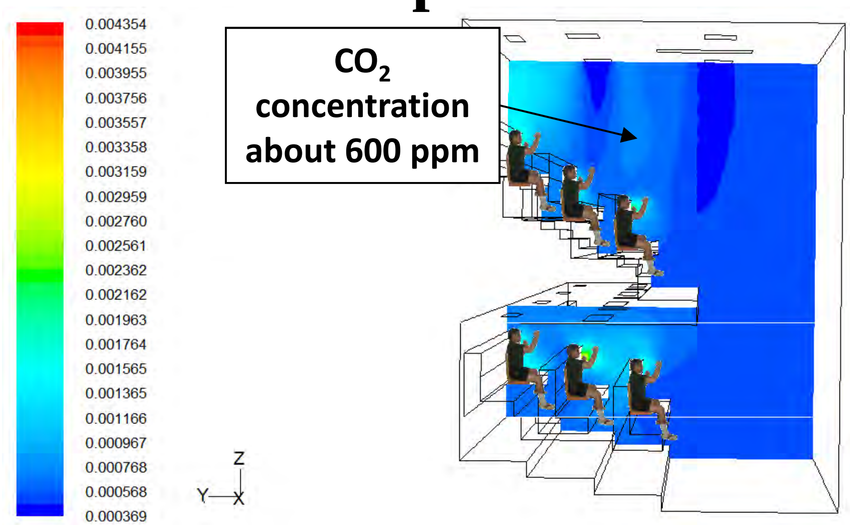
velocity



temperature

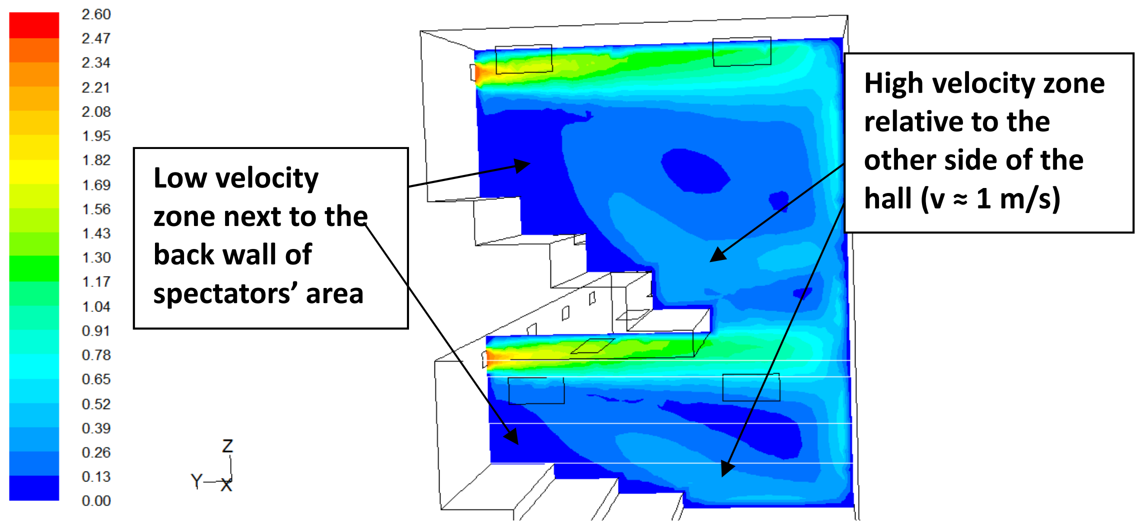


relative humidity

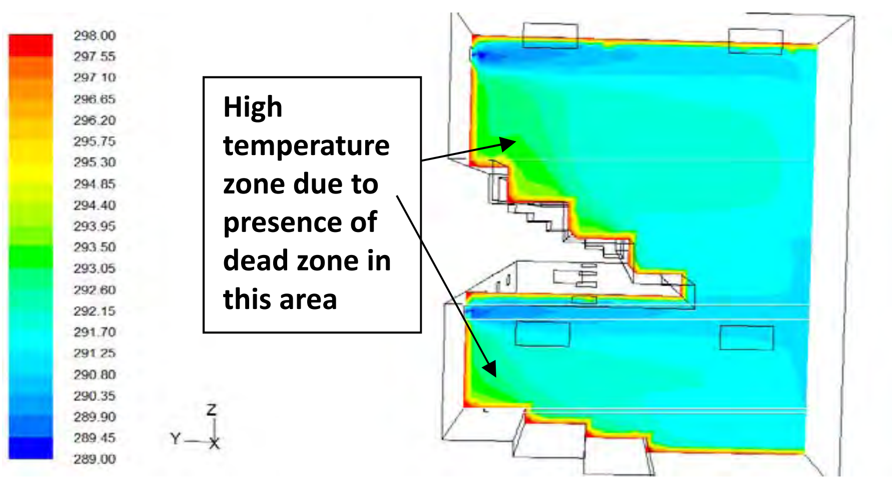


CO_2 concentrations

Case 2 (spectators' area)

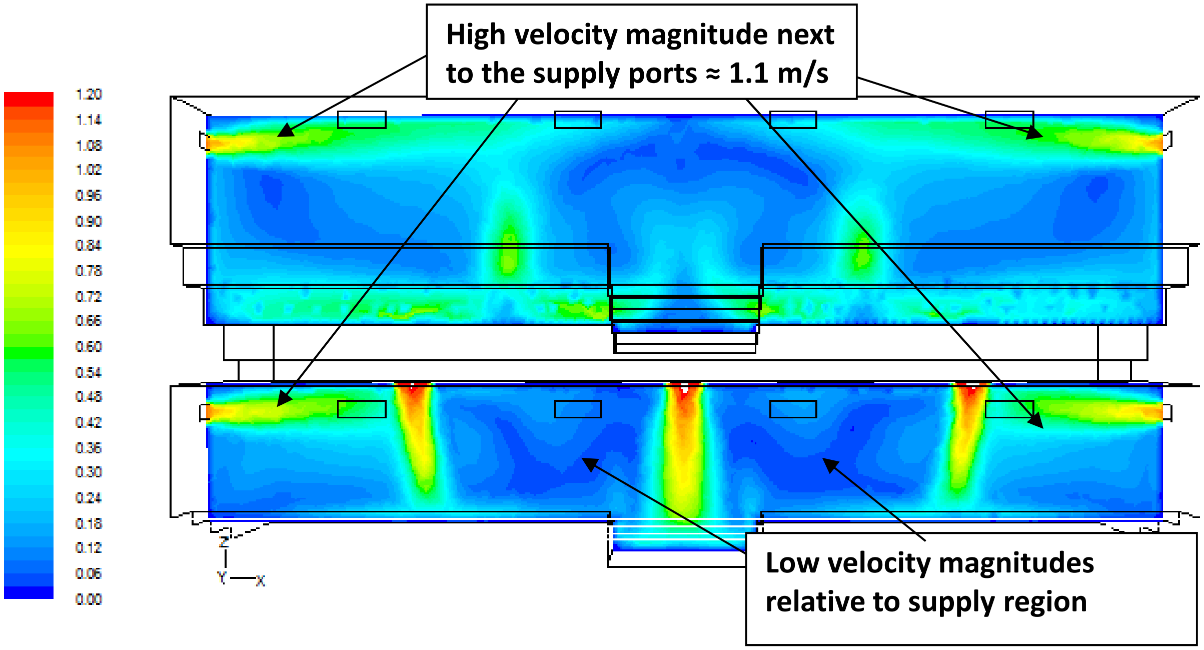


velocity



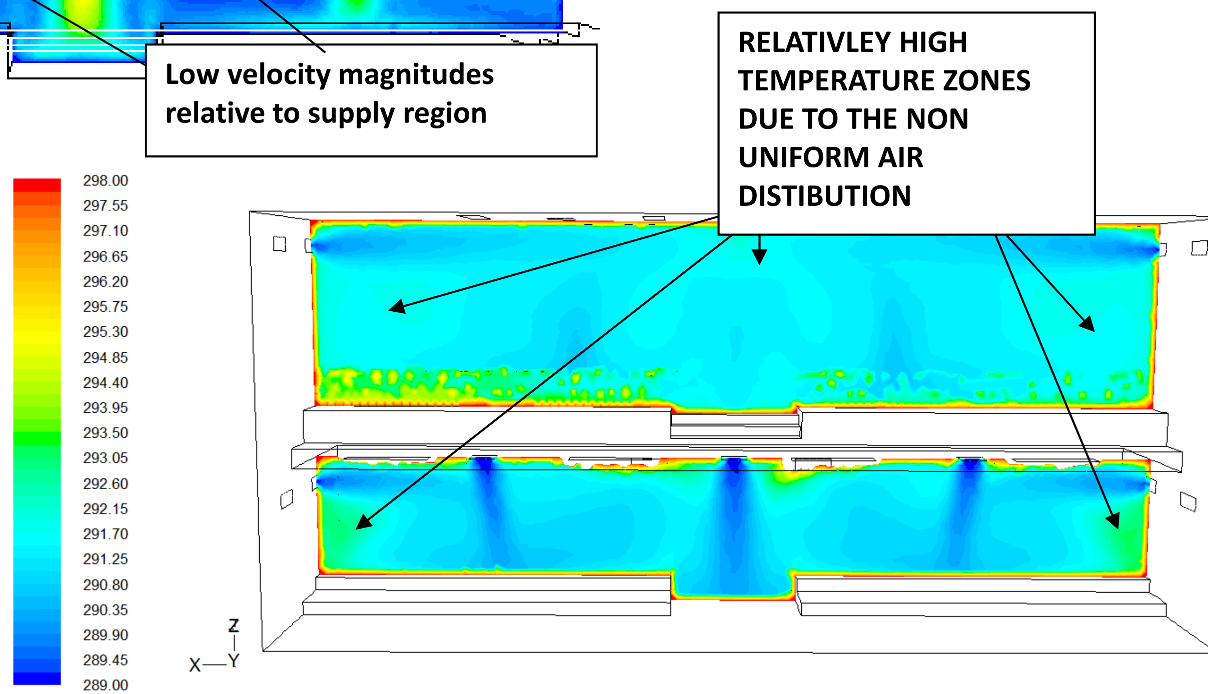
temperature

Case 3 (spectators' area)

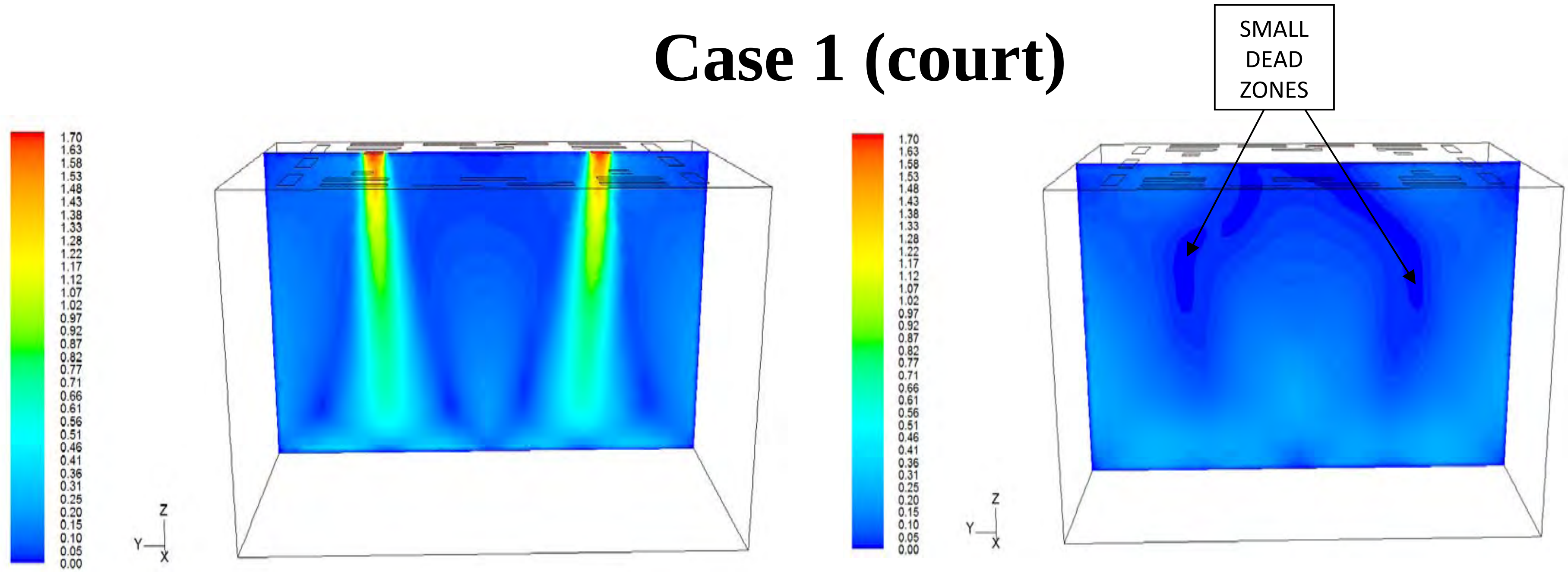


velocity

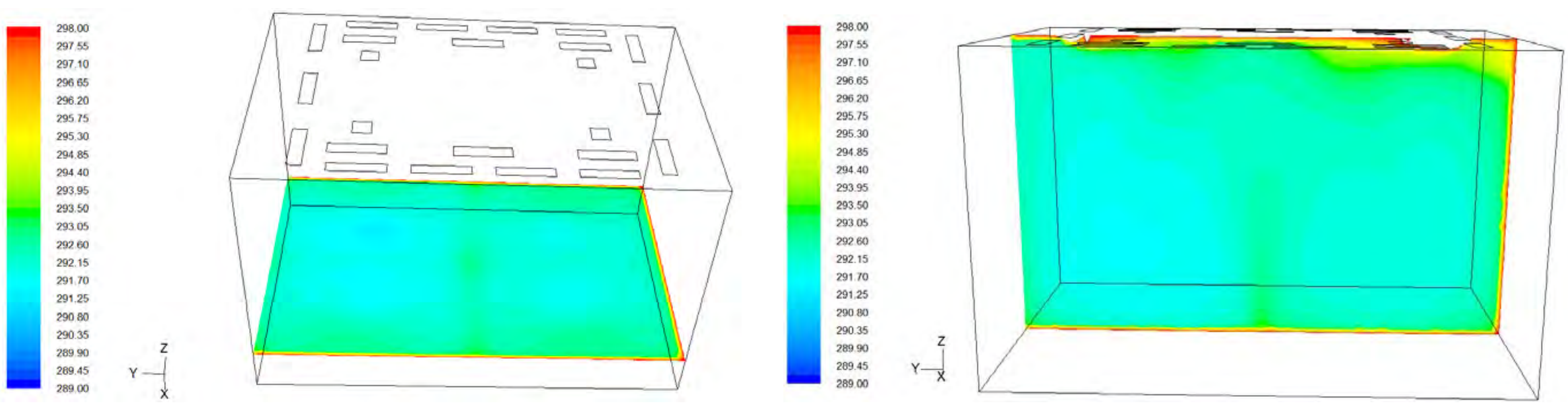
temperature



Case 1 (court)

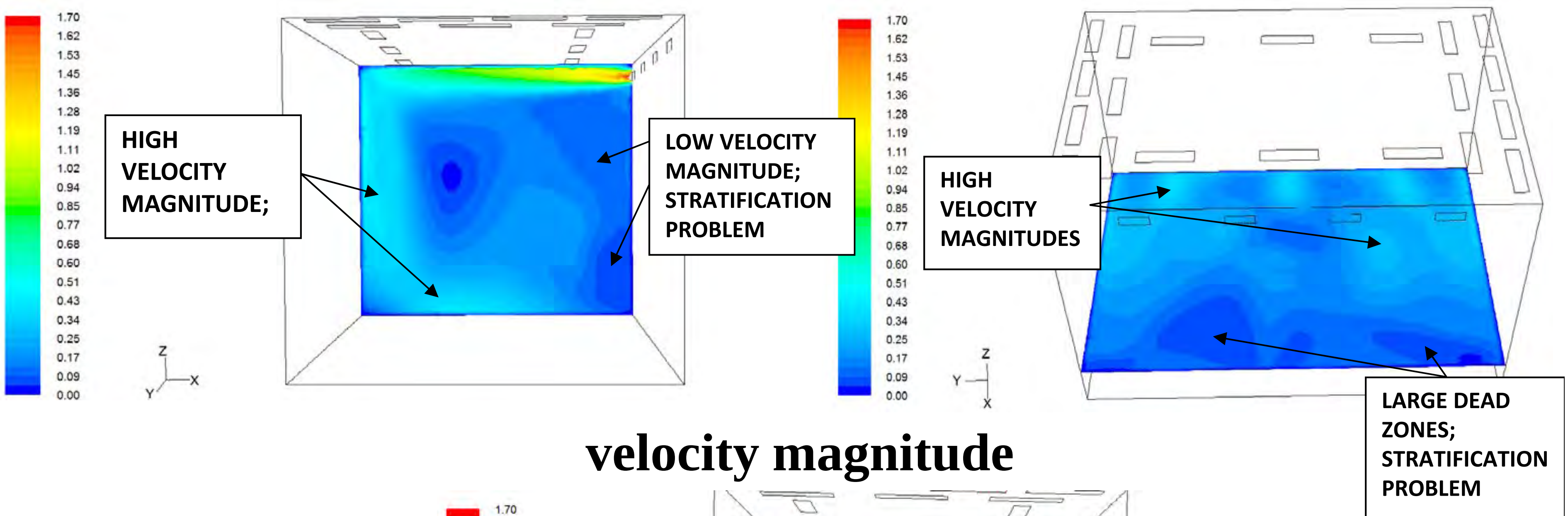


velocity

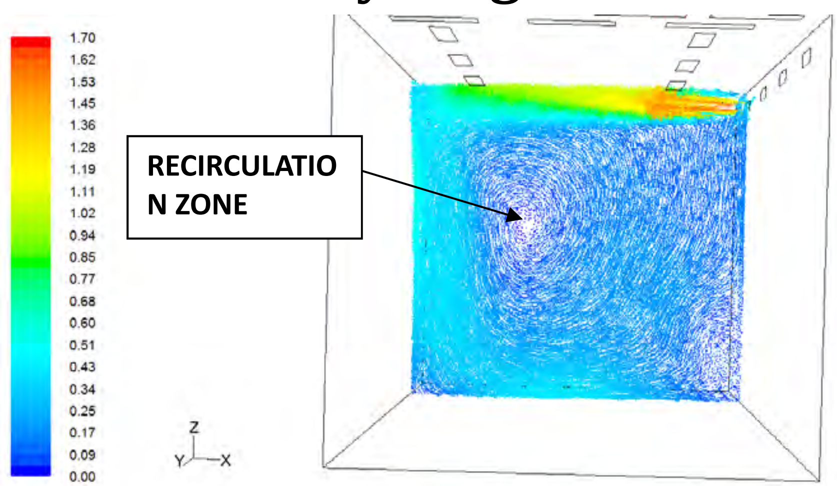


temperature

Case 2 (court)

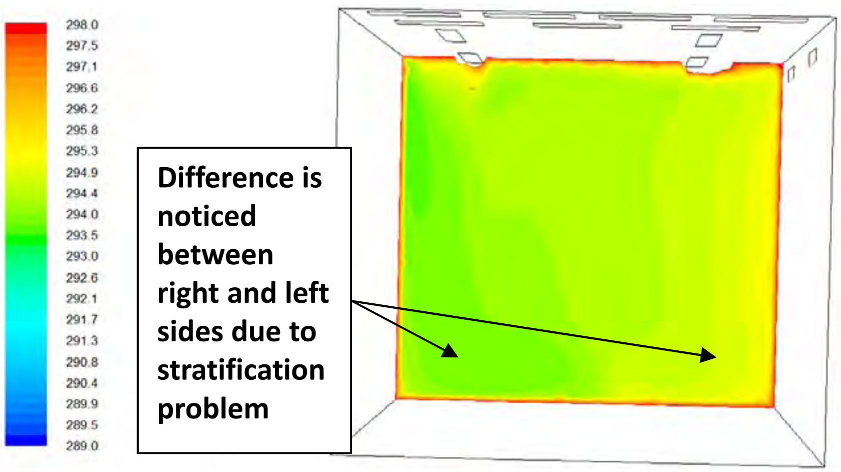


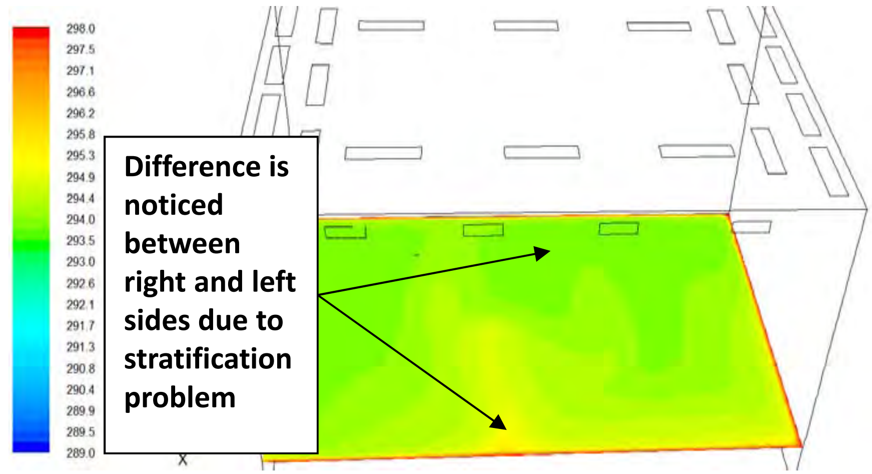
velocity magnitude



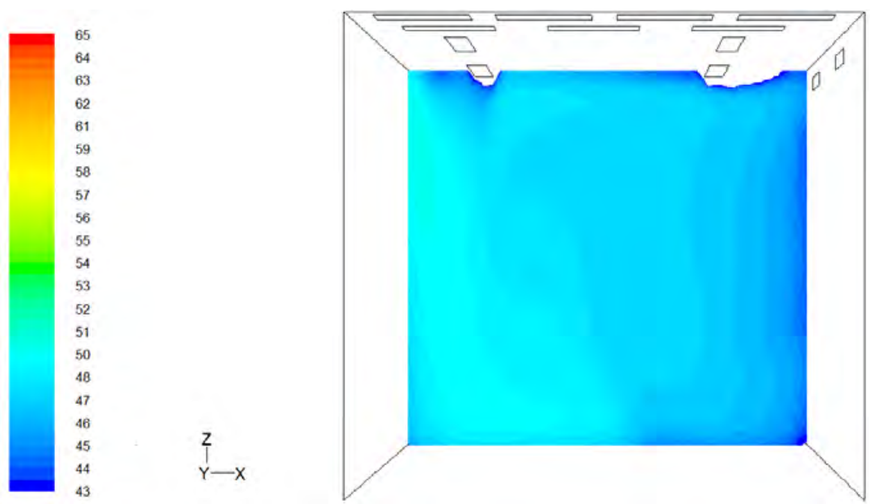
velocity vectors

Case 2 (court)



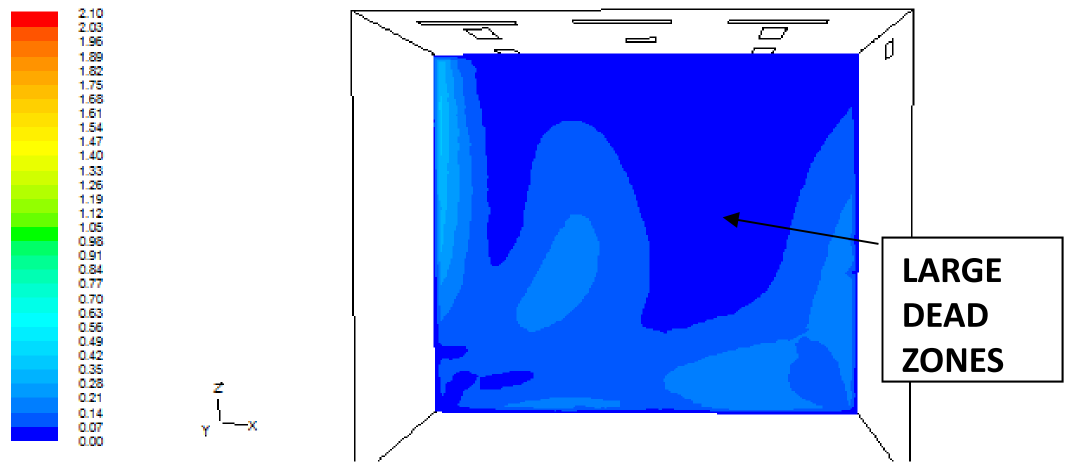


temperature

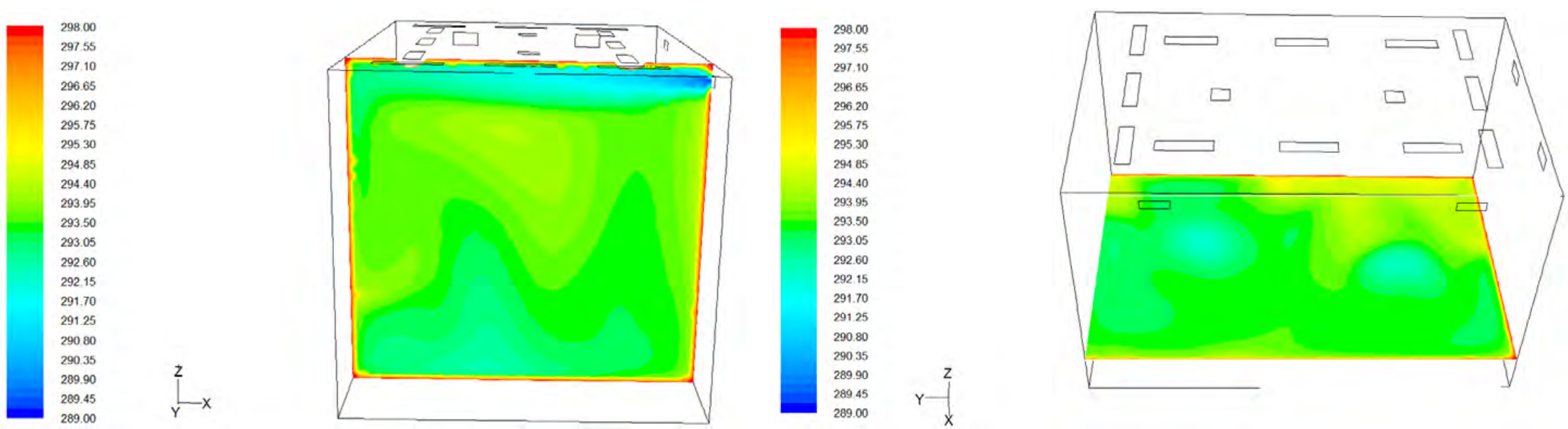


relative humidity

Case 3 (court)



velocity



temperature

CONCLUSION

CASE 1 WINS

Case Study 2 Conclusions

- **Side walls air supply cases are rejected due to uncomfortable conditions**

- **Ceiling air supply (Case 1) showed homogeneous air velocity distribution; It also showed homogeneous distribution and acceptable values of temperature, relative humidity and CO_2 concentration.**