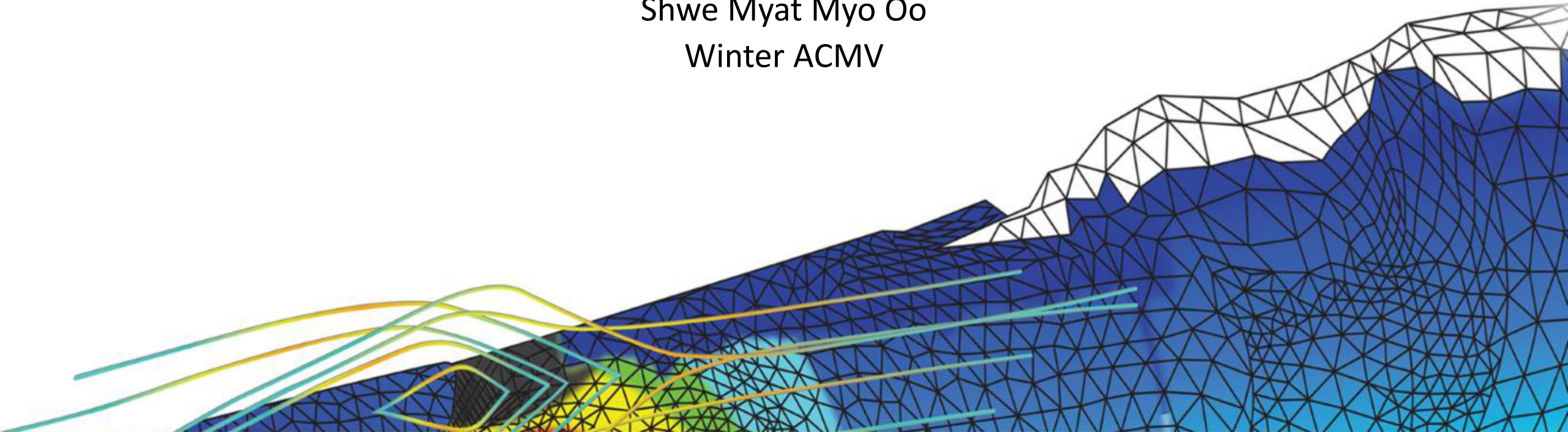# Introduction to CFD (Computational Fluid Dynamics): Simulation of Ventilation for Gamonepwint Market

Shwe Myat Myo Oo

Winter ACMV

Introduction

SPEAKER BACKGROUND

2012-2016

Bachelor of Science in Mechanical Engineering (B.S.M.E)

Certificate in Entrepreneurship

2017-2019

Lead Engineer, Technical Development Division

2019-2021

Master of Science in Energy Systems (MS-ES)

The American Society of Mechanical Engineers
Founded 1880

This Certificate is Awarded to

Shwe Myat Myo Oo

For Successful Completion of
Introduction to Computational Fluid Dynamics

Date Completed: **3/11/2018**
2.30 Continuing Education Units

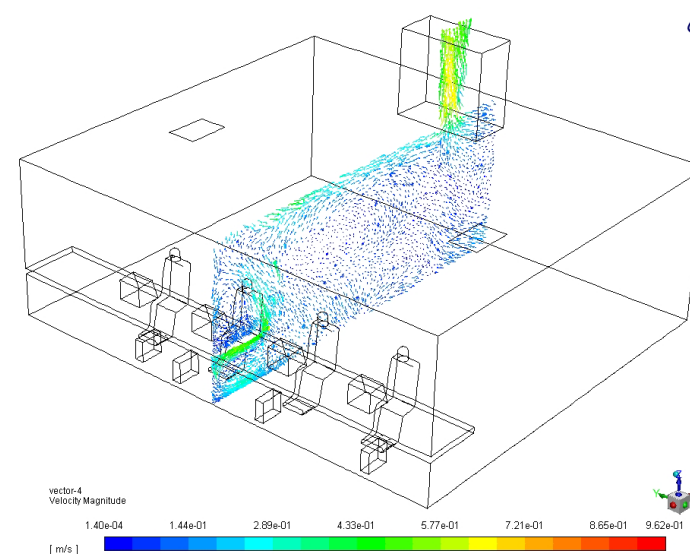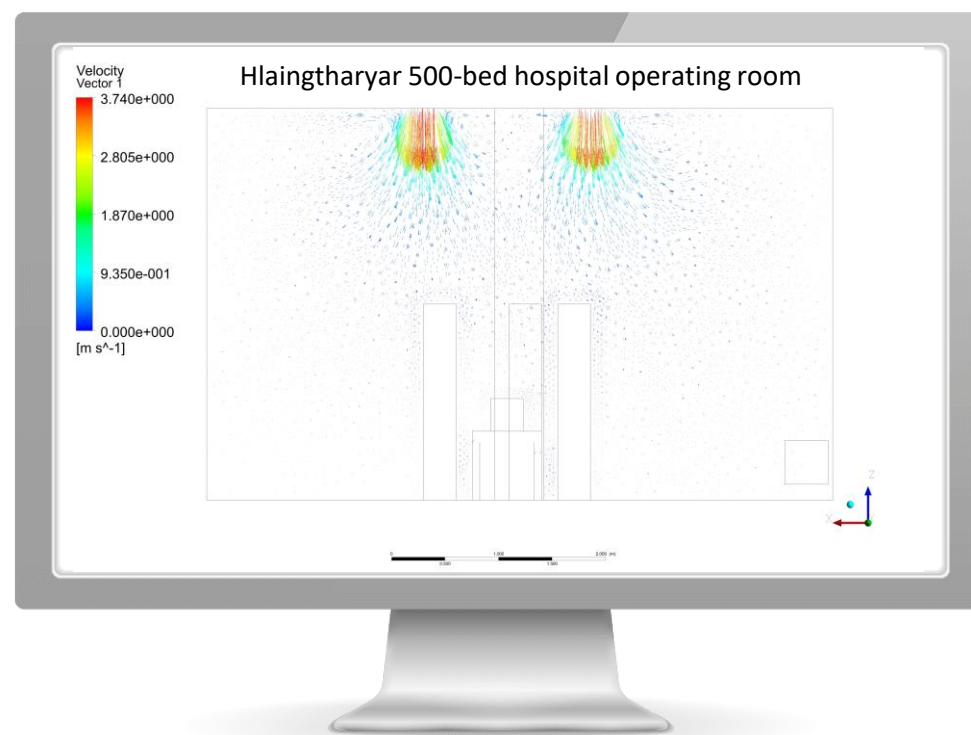Phil W. Hamilton
Interim Executive Director

Arin Ceglia
Director, Learning & Development

ASME Training & Development is accredited by the International Association for Continuing Education and Training (IACET) and is authorized to issue the IACET CEU.
As an IACET Accredited Provider, ASME Training & Development offers CEUS for its programs that qualify under the ANSI/IACET Standard.

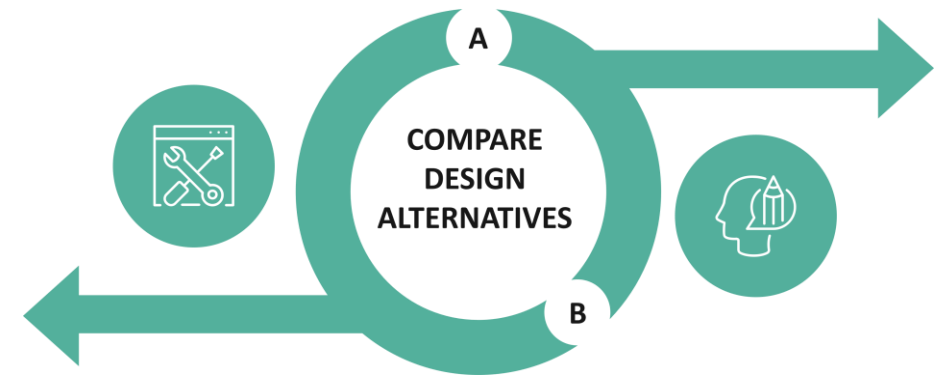ASME Two Park Avenue, New York, New York 10016

Hlaingtharyar 500-bed hospital operating room
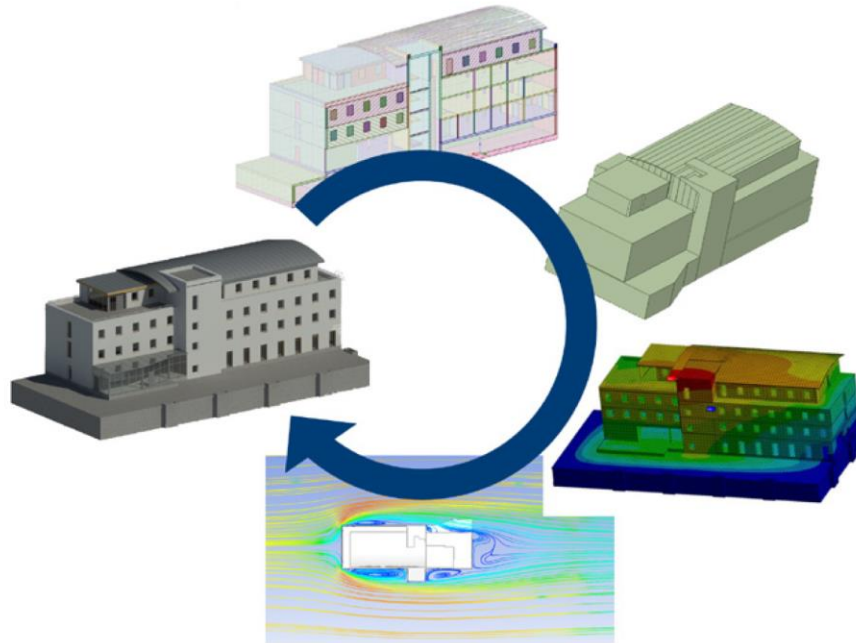
Velocity Vector 1
3.740e+000
2.805e+000
1.870e+000
9.350e-001
0.000e+000
[m s^-1]

vector-4
Velocity Magnitude

1.40e-04   1.44e-01   2.89e-01   4.33e-01   5.77e-01   7.21e-01   8.65e-01   9.62e-01
[m/s]

WHY CFD?

COST SAVINGS

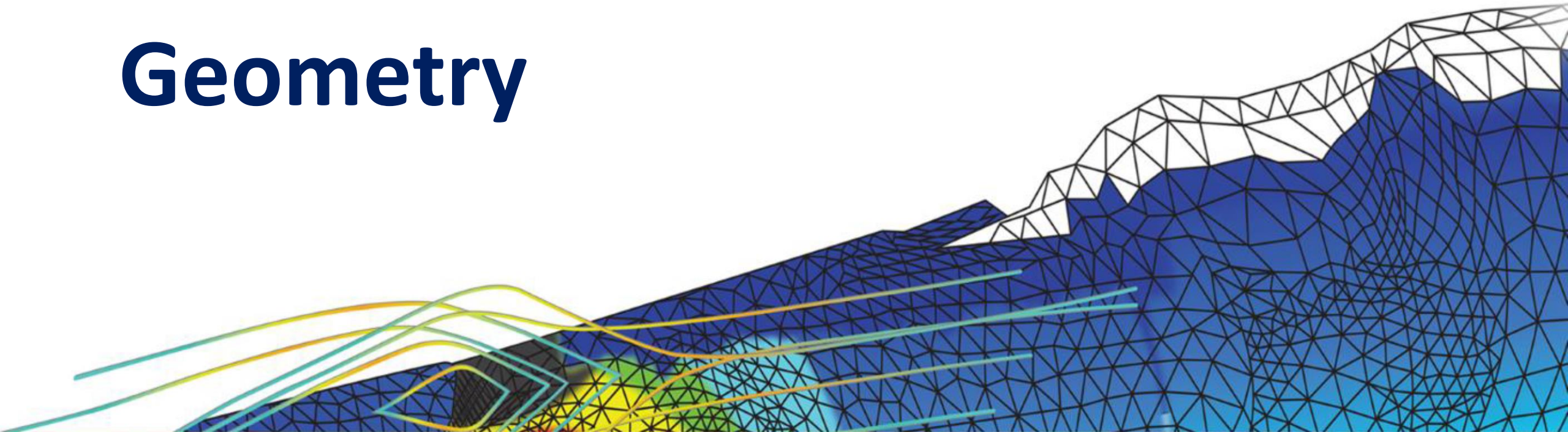**BROAD APPLICABILITY**

DESIGN COMPARISON

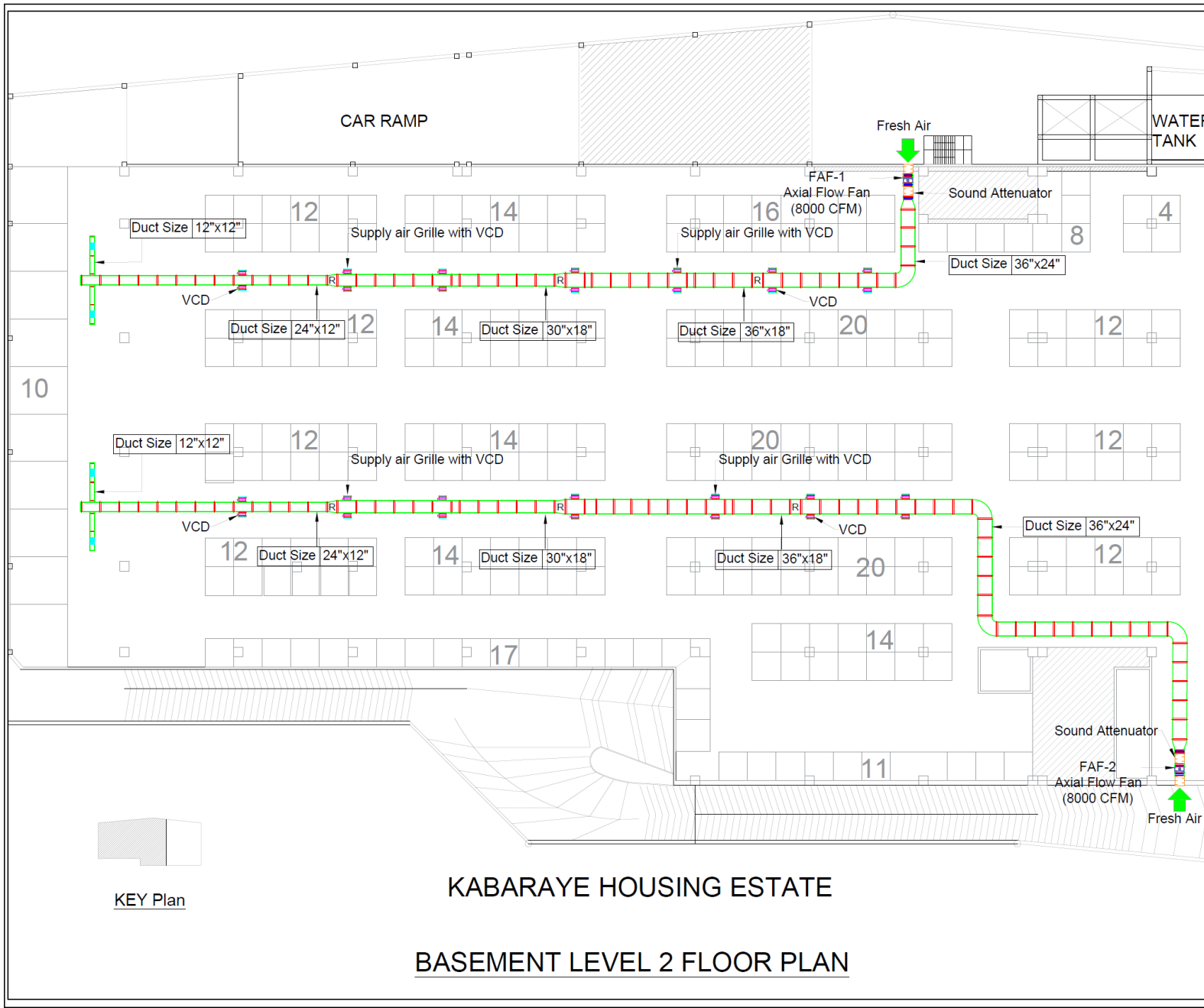COMPARE DESIGN ALTERNATIVES

A

B

CFD For Buildings

- External aerodynamics (wind load)
- Thermal comfort, ventilation analysis
- Airflow optimization
- Special applications (data center cooling, hospital operating rooms, etc)
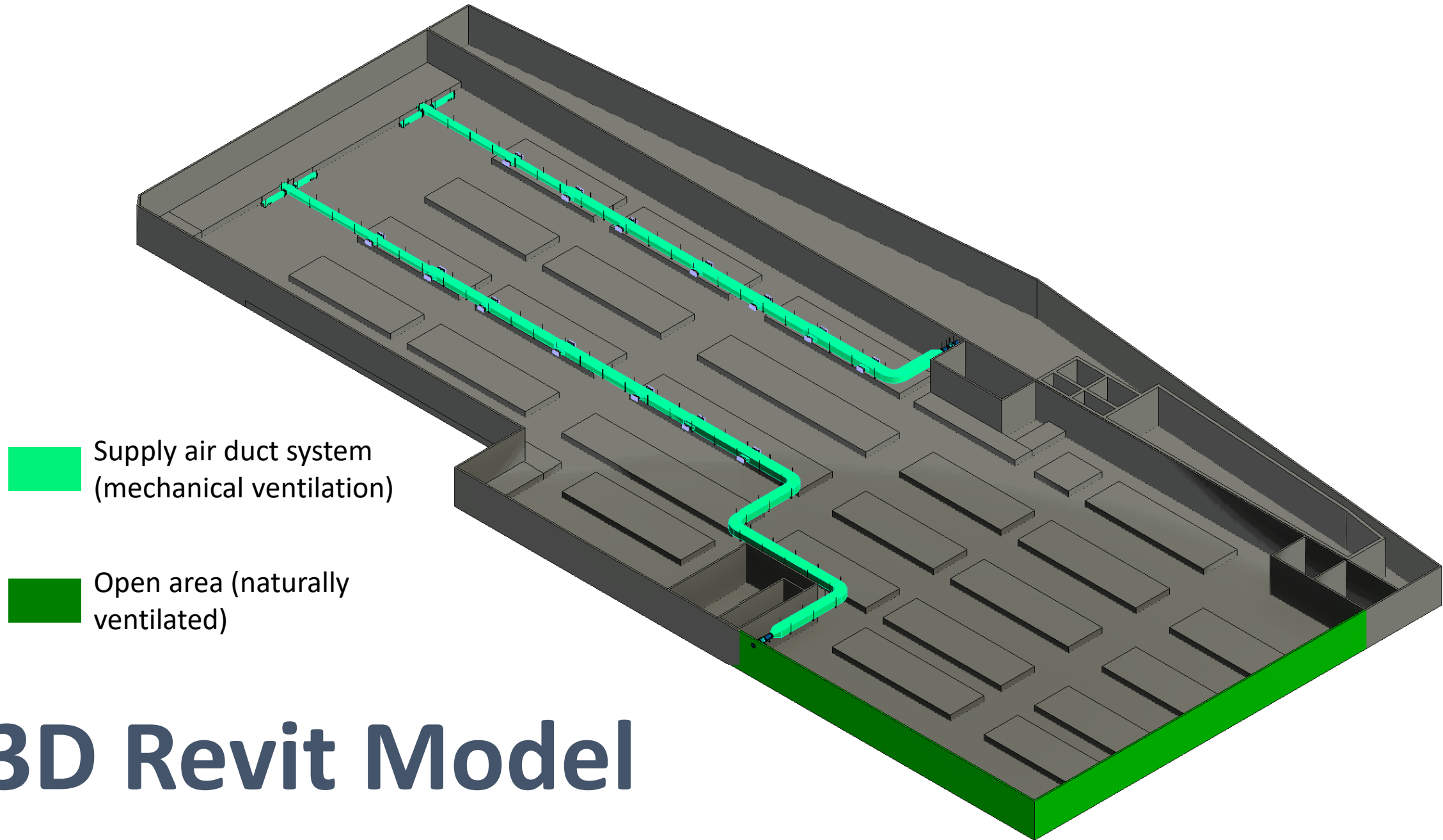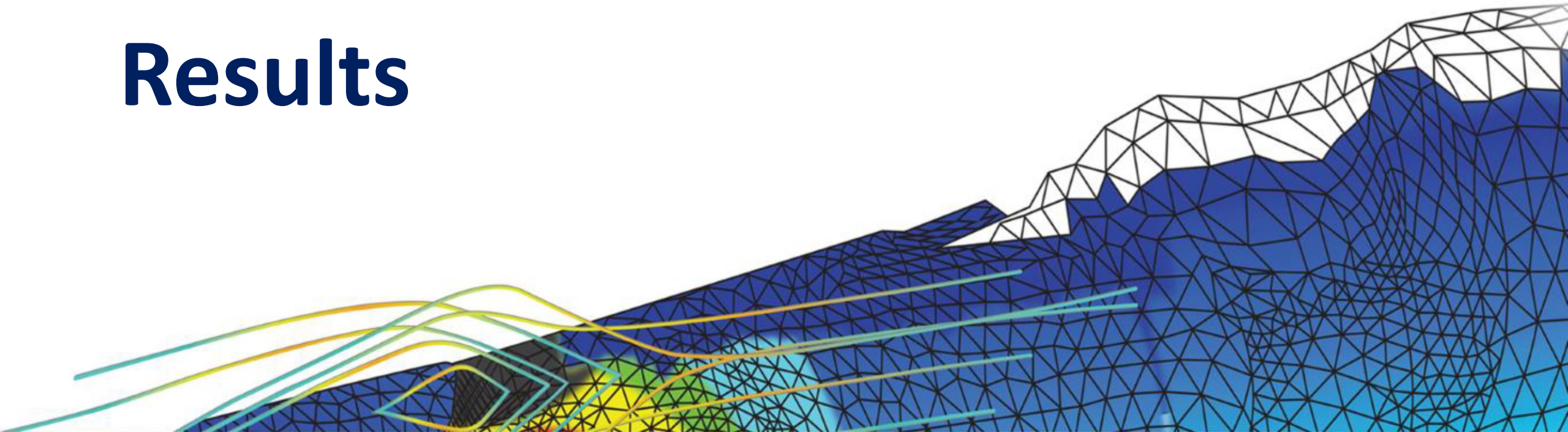
Geometry

CAR RAMP

WATER TANK

Fresh Air

FAF-1
Axial Flow Fan
(8000 CFM)

Sound Attenuator

12   14   16   4

Duct Size 12"x12"   8

Supply air Grille with VCD   Supply air Grille with VCD

Duct Size 36"x24"

VCD

Duct Size 24"x12"   12   14   Duct Size 30"x18"   Duct Size 36"x18"   20   12

VCD

10

Duct Size 12"x12"   12   14   20   12

Supply air Grille with VCD   Supply air Grille with VCD

VCD   Duct Size 36"x24"

VCD   12   Duct Size 24"x12"   14   Duct Size 30"x18"   Duct Size 36"x18"   20   12

17   14

Sound Attenuator

11   FAF-2
Axial Flow Fan
(8000 CFM)

Fresh Air

KEY Plan

KABARAYE HOUSING ESTATE

BASEMENT LEVEL 2 FLOOR PLAN

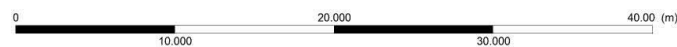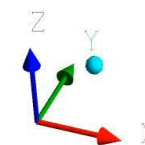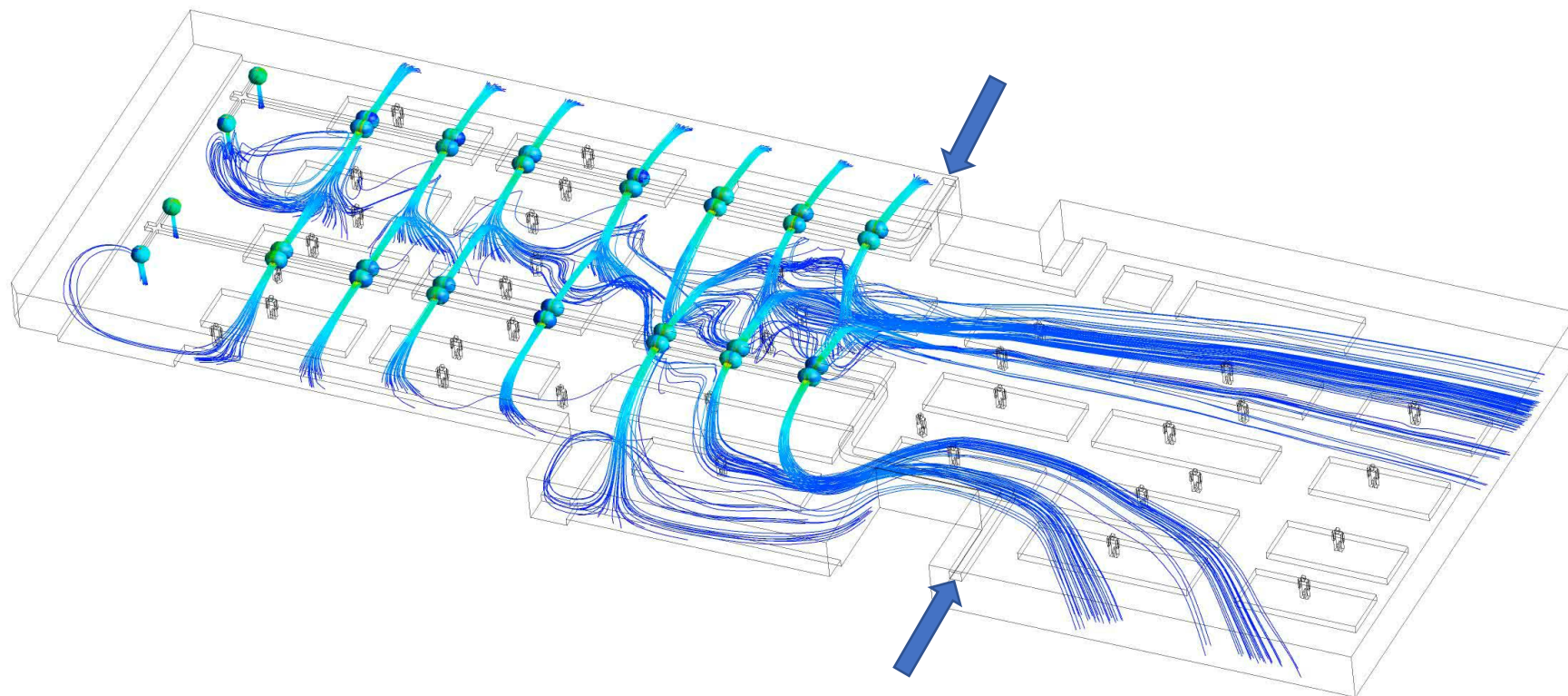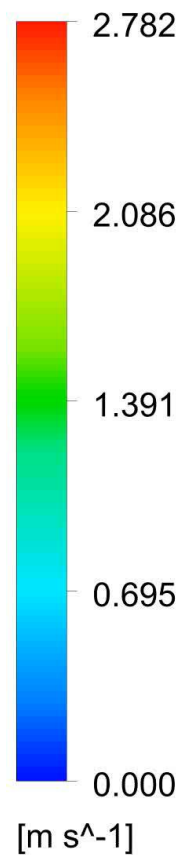| APPLICANT	
PROJECT	KABARAYE HOUSING ESTATE
	BLOCK NO.
	LOT NO.
	BAHAN TOWNSHIP
SUBJECT	Mechanical Ventilation Fan Location and Duct Layout Plan (Basement-2)
SCALE	1 : 220
DATE	July 2020
DRAWN BY;	Aung Myint Zaw
CHECKED BY;	Abel Pwa
APPROVED BY;	Shwe Yu Kyaw
DRAWING NO;	MV 01 / KBA / Y2K20
REVISE NO;	

Supply air duct system
(mechanical ventilation)

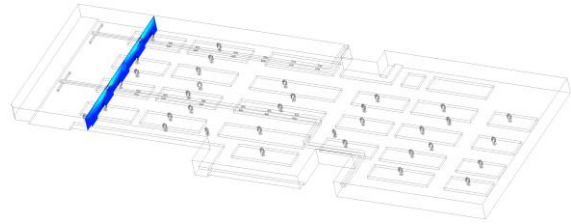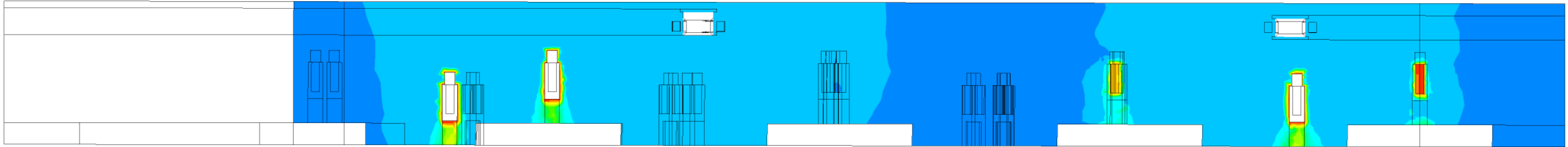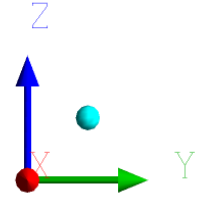Open area (naturally
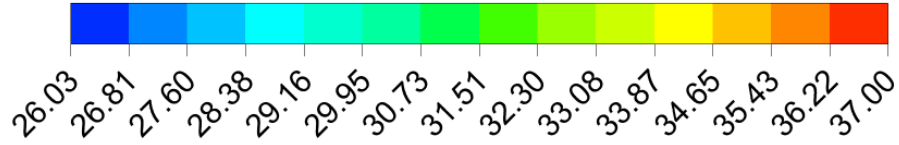ventilated)

3D Revit Model

Results

Velocity

2.782

2.086

1.391

0.695

0.000

[m s^-1]

YANGON CLIMATE (MYANMAR)

Weather data source:
https://en.climate-data.org

Temperature [C]

26.03 26.81 27.60 28.38 29.16 29.95 30.73 31.51 32.30 33.08 33.87 34.65 35.43 36.22 37.00

Velocity [m s^-1]

0.00 0.15 0.29 0.44 0.59 0.73 0.88 1.02 1.17 1.32 1.46 1.61 1.76 1.90 2.05 2.20 2.34 2.49 2.64 2.78
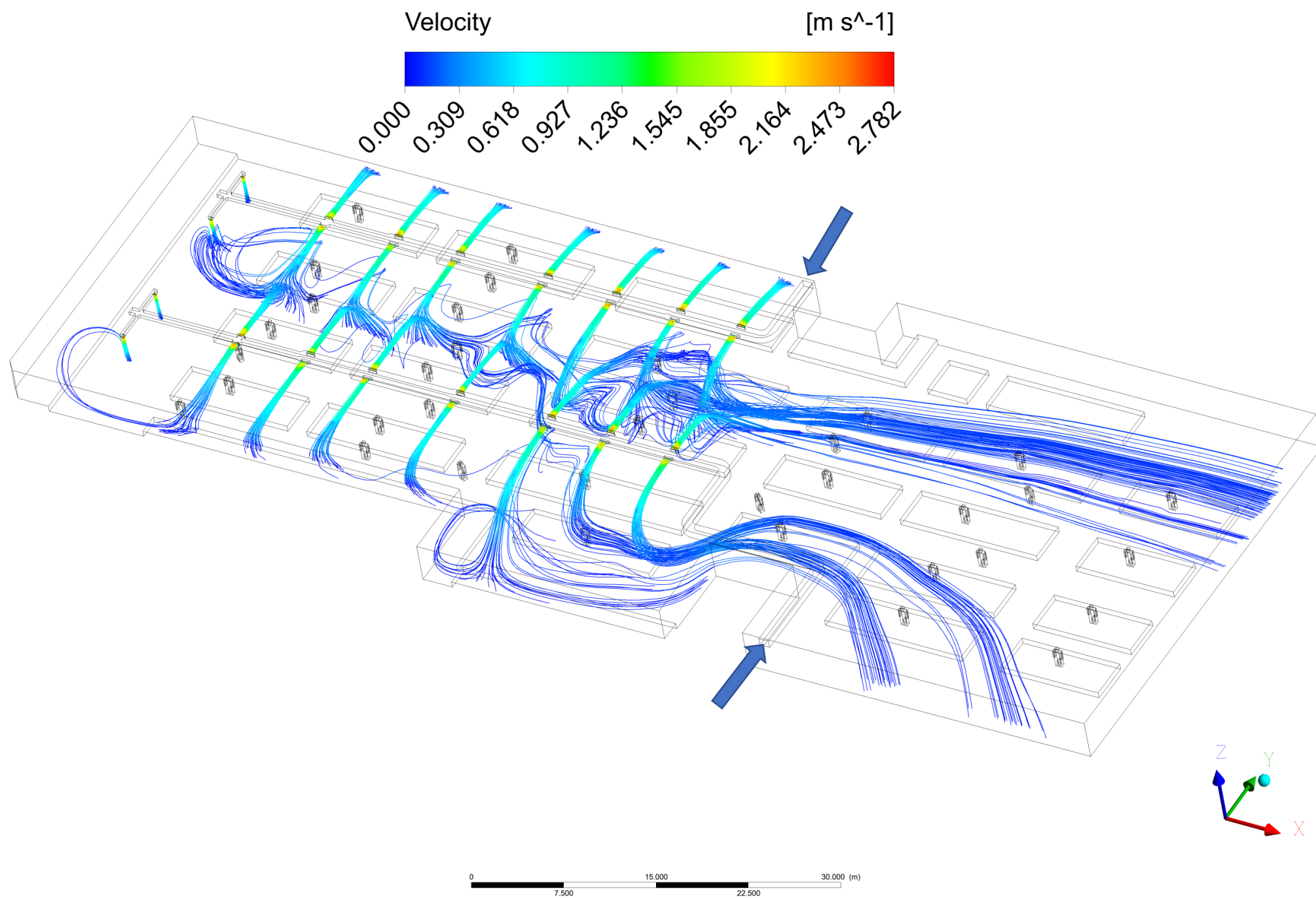
Velocity [m s^-1]

0.000 0.309 0.618 0.927 1.236 1.545 1.855 2.164 2.473 2.782

0   7.500   15.000   22.500   30.000 (m)
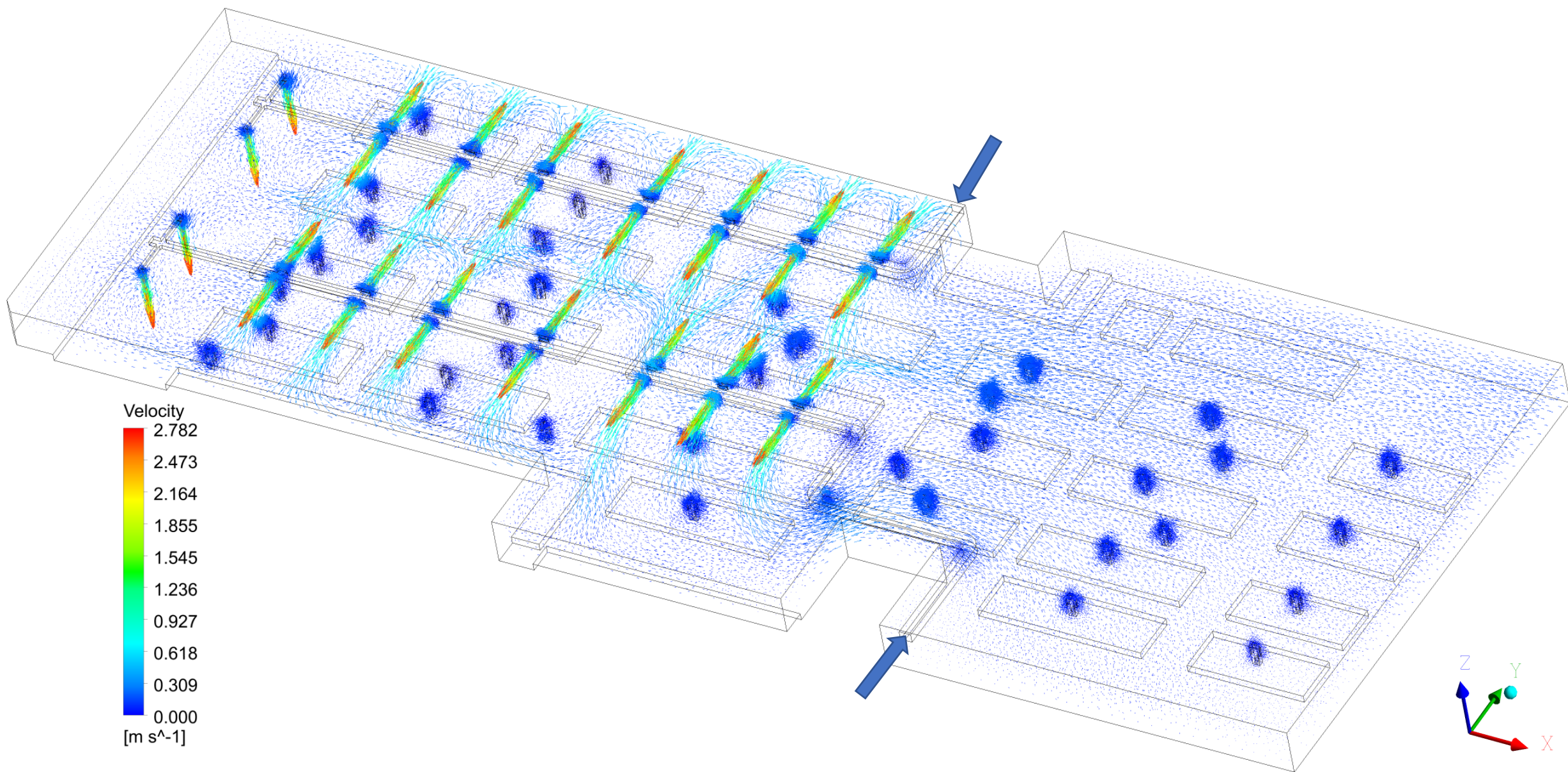
Velocity

2.782
2.473
2.164
1.855
1.545
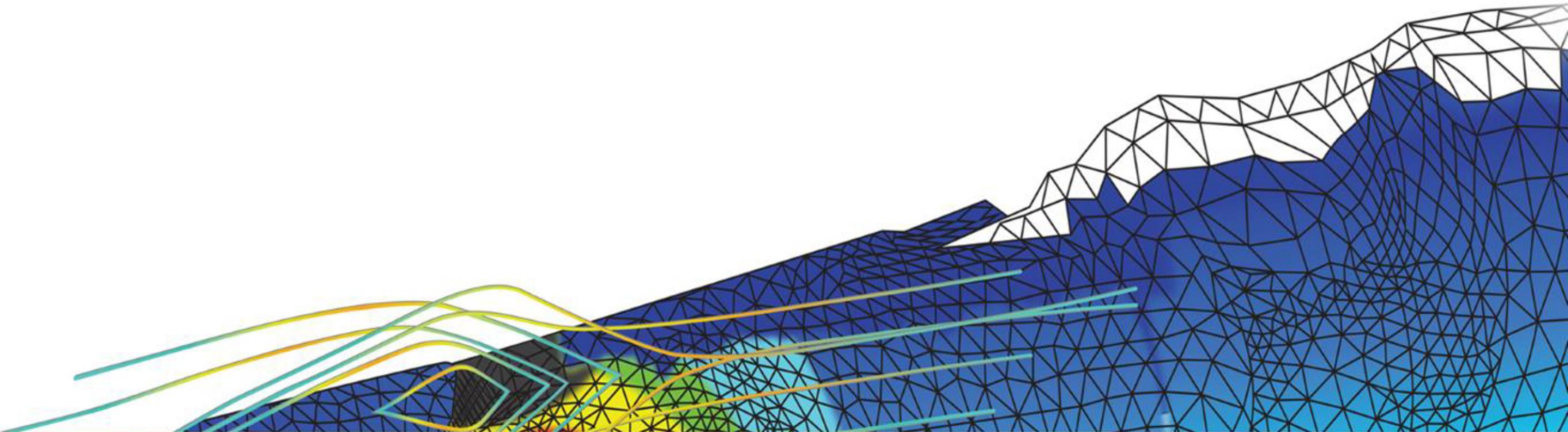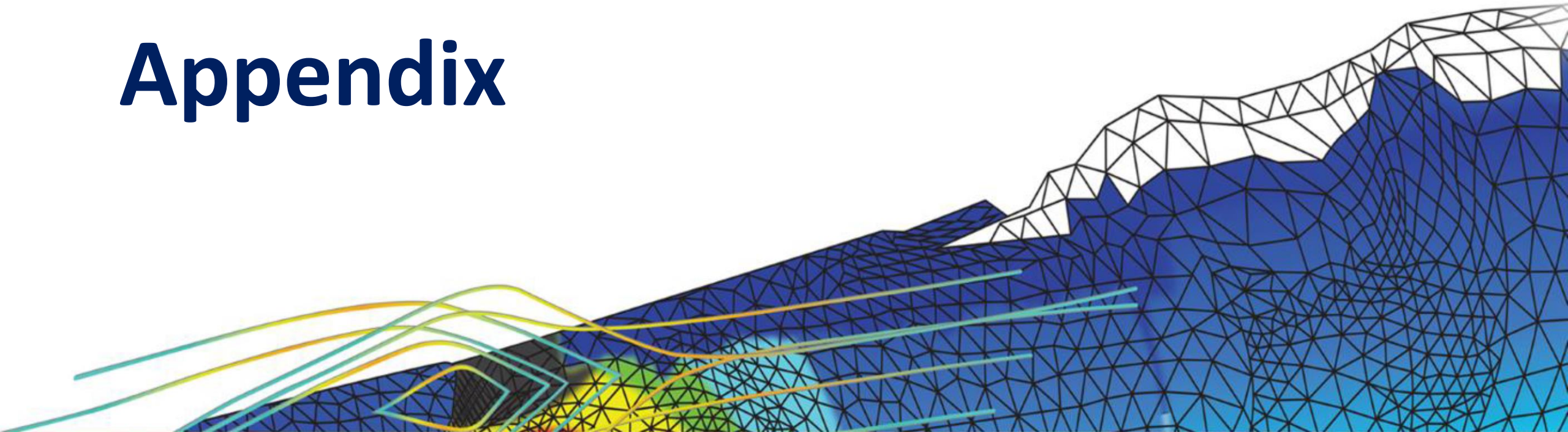1.236
0.927
0.618
0.309
0.000
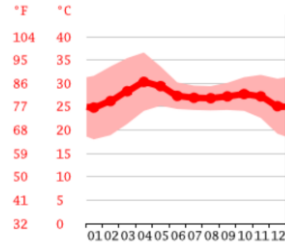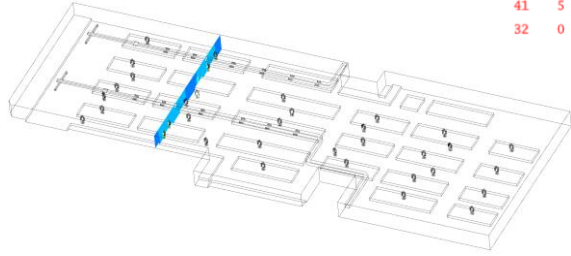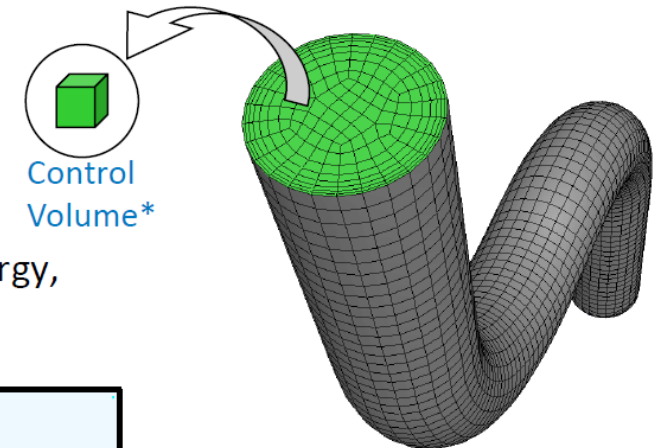
[m s^-1]
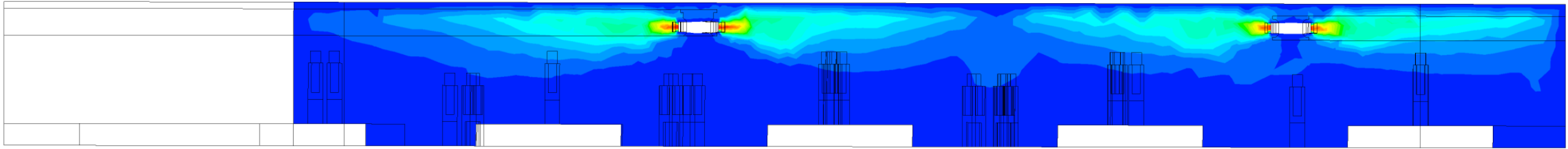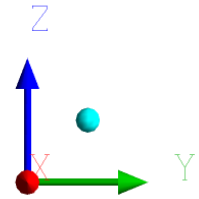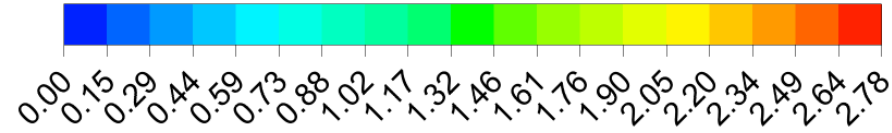
Thank You

Appendix

WHAT IS CFD?

How Does CFD Work?

ANSYS CFD solvers are based on the finite volume method

- Domain is discretized into a finite set of control volumes

- General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes

Control Volume*

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi\nabla\phi\cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

Equation	ϕ
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

- Partial differential equations are discretized into a system of algebraic equations

- All algebraic equations are then solved numerically to render the solution field

CFD IS MULTIDISCIPLINARY