

**MAE 560/460 Applied Computational Fluid Dynamics**  
Fall 2020 Tue/Thu 1:30-2:45, ASU Sync/In-Person, Classroom: LIB C12

Instructor: Huei-Ping Huang (hp.huang@asu.edu), ERC 359  
Office hours: Monday 3-5 PM, Tuesday 3-5 PM, or by appointment

Office hours will be hybrid with both in-person  
and remote components. Detail forthcoming.

Instructor will email the class on logistic details for  
class attendance (in-person vs. remote).

## Two websites for the class

**1. Main course website <http://www.public.asu.edu/~hhuang38/ACFD2020.html>**

schedule update, critical announcements, projects, supplementary resources, archive of slides and backup videos

**2. Canvas website for MAE560/460 (access through MyASU)**

submission of project reports and answer for exam, grades

## **Topics to cover**

- **Techniques for solving incompressible and compressible flow equations using commercial/industrial solvers (We will use **Ansys-Fluent** as the principal tool)**
- **Computer-aided analysis of fluid systems**
- **Applications to thermofluid system engineering**

**No required textbook. Instructor will provide tutorials and lecture notes as needed.**

**Attendance is strongly encouraged.**

## Planned activities

We will run two threads concurrently through the semester:

*Lecture* thread fills the background knowledge on fluid mechanics and numerical methods.

*Project* thread focuses on the execution and analysis of specific projects using Ansys-Fluent.

Ansys provides student version of their software for free, with some restrictions (limit of number of nodes, etc.). The student version of Ansys-Fluent is good enough for all projects in this class.

**Pending:** Ansys-Fluent is available on the computers in GWC 481/483 computing lab. Access (including remote login) to the lab is being worked on.

## Plan for the semester

### *I. Lectures*

1. Survey of basic fluid mechanics and thermodynamics as preparation for the projects (4 weeks)
2. Survey of numerical methods for deepening the understanding of the functionality of Ansys-Fluent or similar industrial software (6 weeks)
3. Discussion on more advanced topics in CFD (e.g., turbulence modeling) (3 weeks)

### *II. Projects*

1. Tutorials for Ansys-Fluent (2 weeks)
2. Main projects (12 weeks)

*At least four of the following projects (3 weeks each) will be chosen for this semester. The list is tentative and the detail of the individual project is subject to further adjustments.*

Project 1: Fluid system with heat transfer

Project 2: External flow (calculation of drag and lift; Reynolds number dependence)

Project 3: Compressible flow system

Project 4: Low Reynolds number flow; Microfluidics

Project 5: Moving boundary and moving grid

Project 6: Flow with an interface (two-phase or multi-phase)

**Grade:** Projects & homework 90%  
Final exam 10%

Specific rules concerning collaboration on projects/homework will be released along with the release of each assignment.

The required work for the projects will be different for participants of MAE560 and MAE460. Details will be given in the individual assignments.

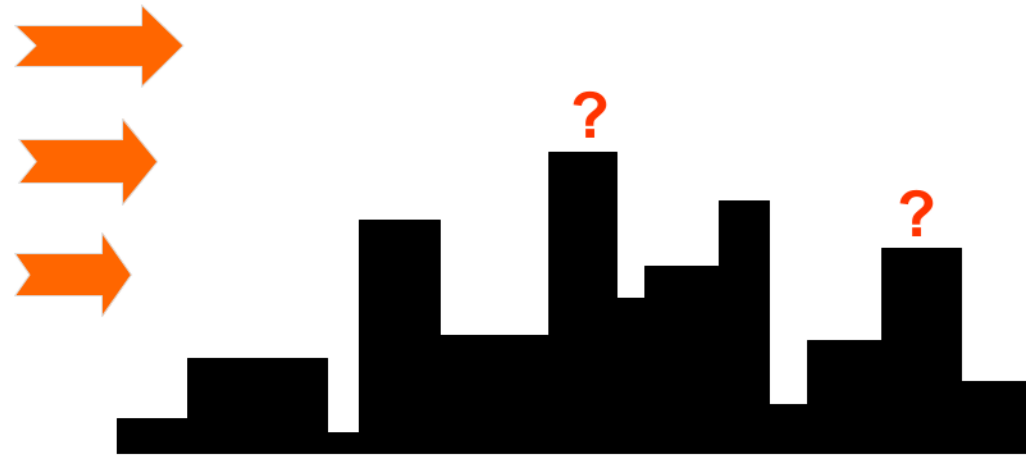
**Please make sure that you are familiar with ASU policies on academic integrity:** <https://provost.asu.edu/academicintegrity>

## Some examples of computational fluid dynamics using commercial CFD solvers

Many commercial/industrial CFD solvers are available on the market. This class will use Ansys-Fluent but might include a short survey on other CFD solvers (Comsol, AcuSolve, Abacus, Autodesk Flow Design, etc.) if time permits.

# Example 1 External flow - using *Autodesk Flow Design*

## *Assessment of rooftop wind power potential*



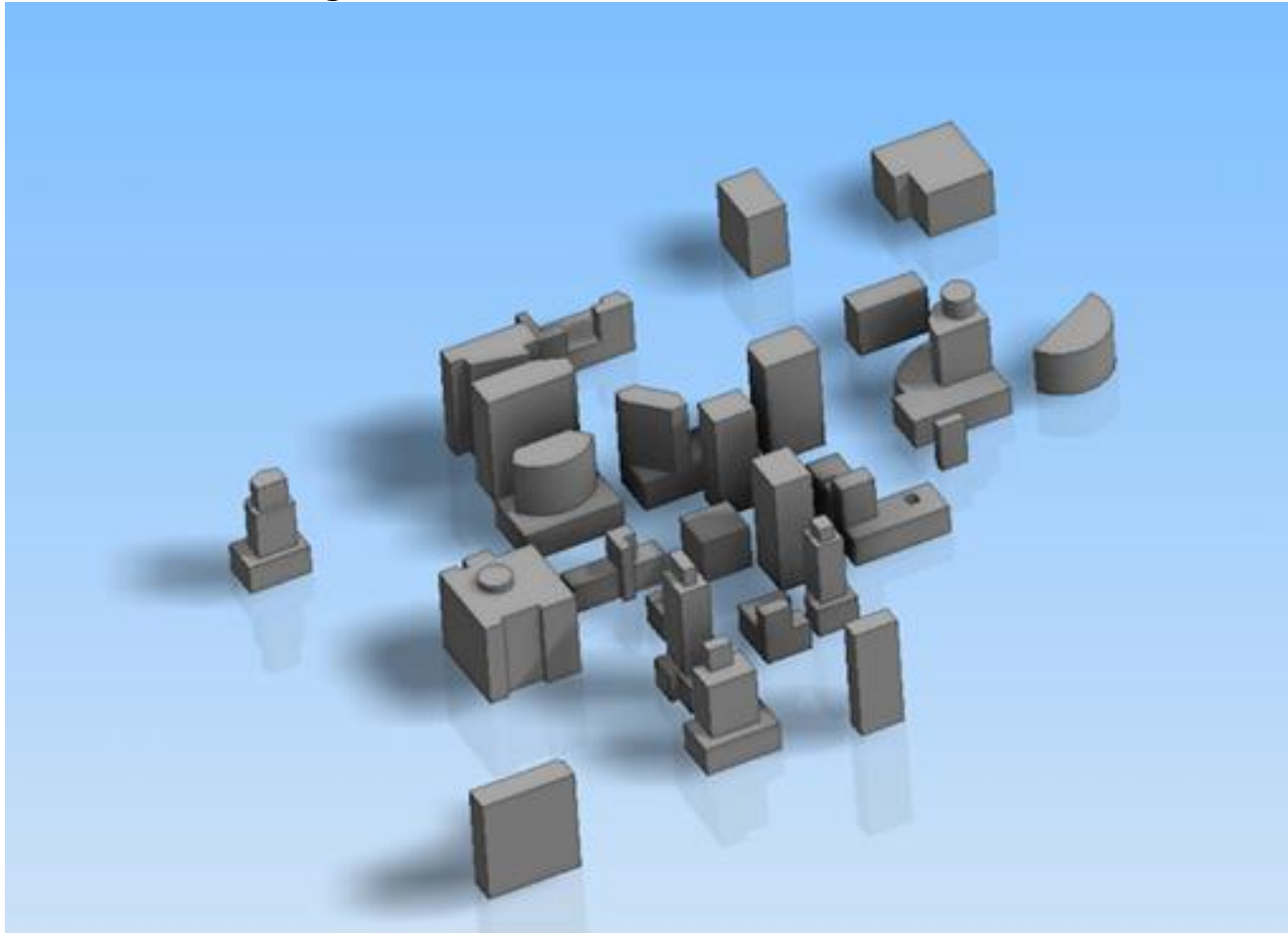
(Urban community, college campus, etc.)



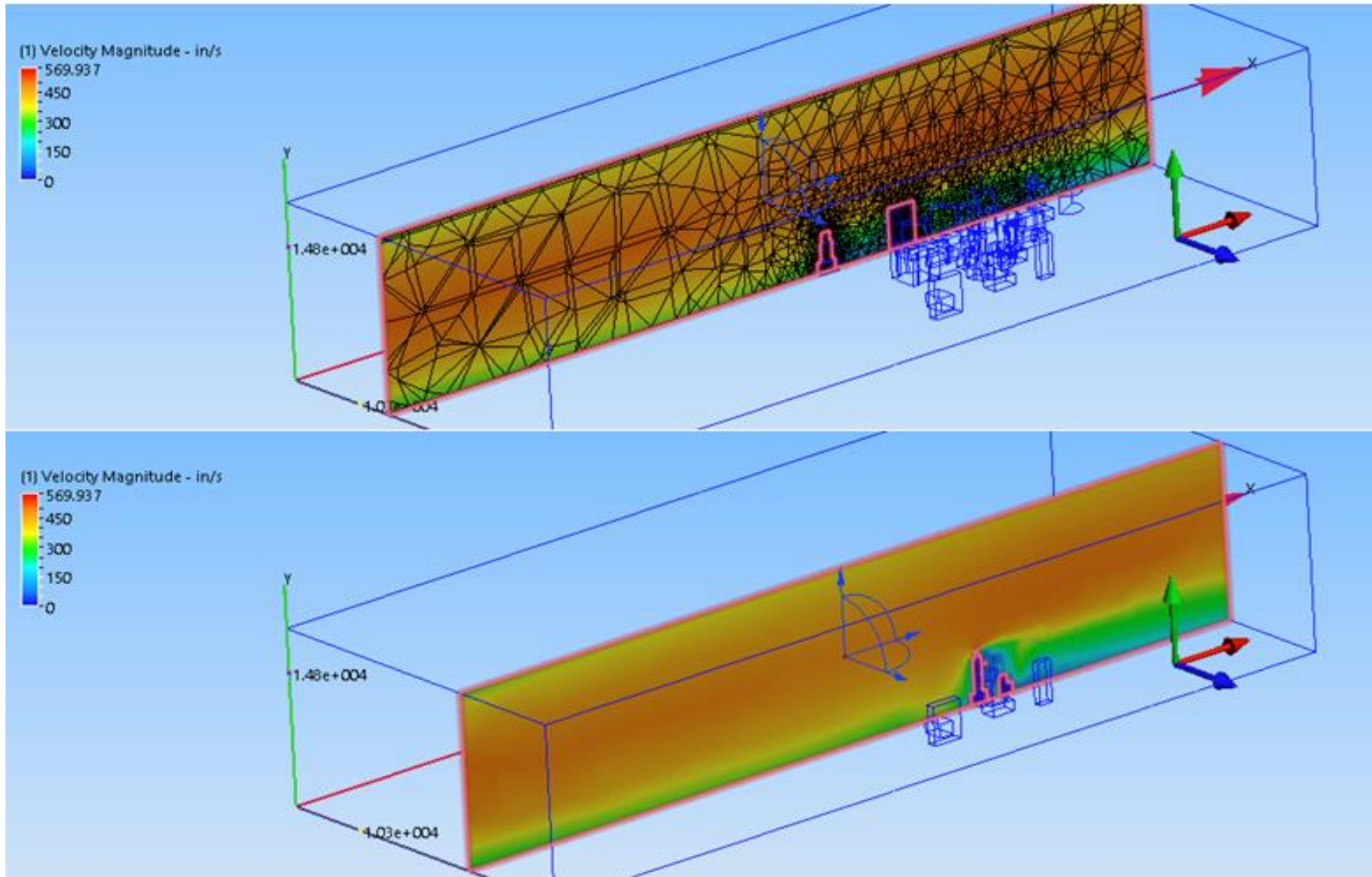
## Simulation for Downtown Phoenix

[Research project, 2015, X. Ying (Huang group)]

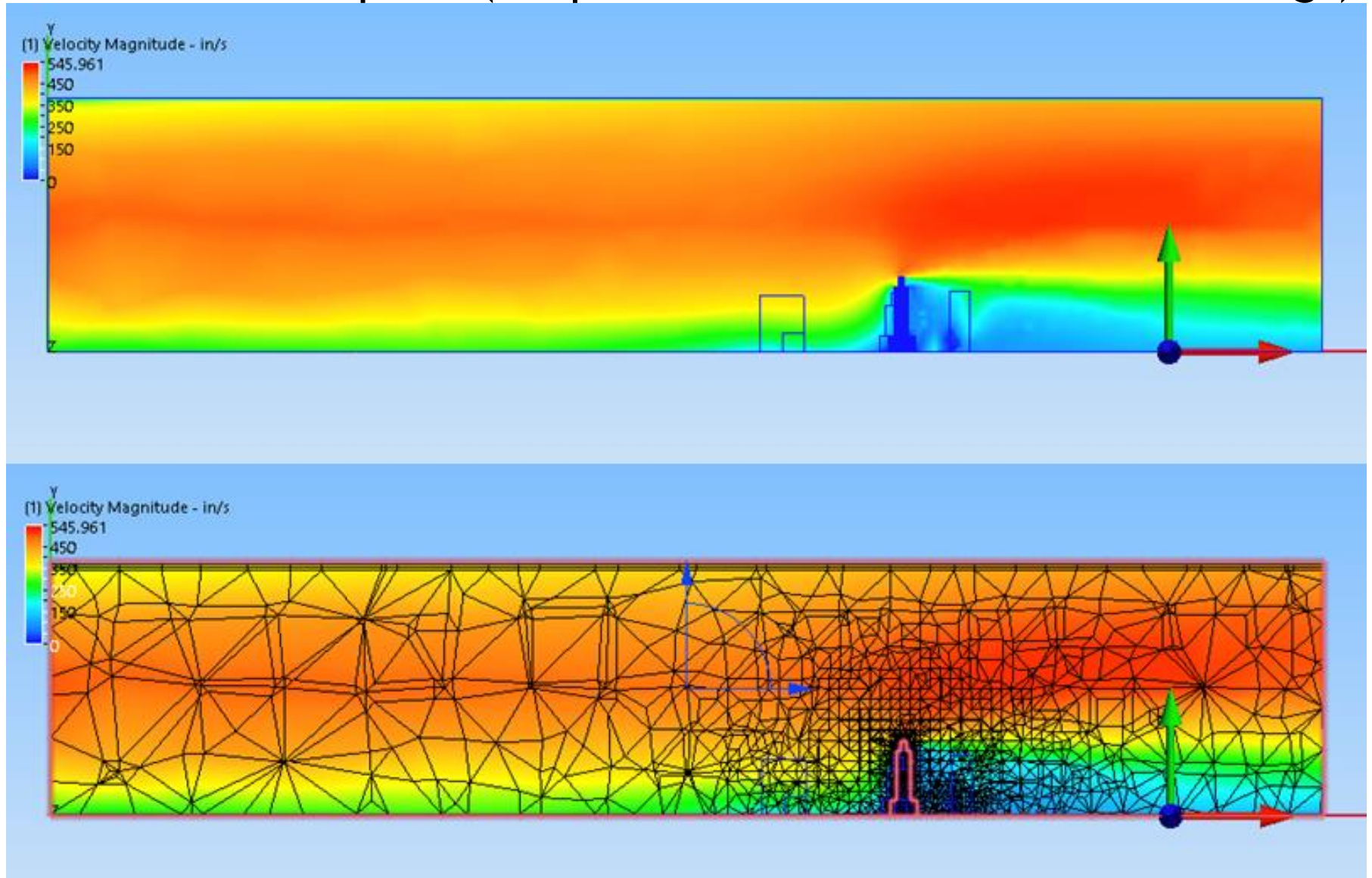
- 3-D geometric data from Google Earth
- Keep 23 tallest buildings for flow simulation



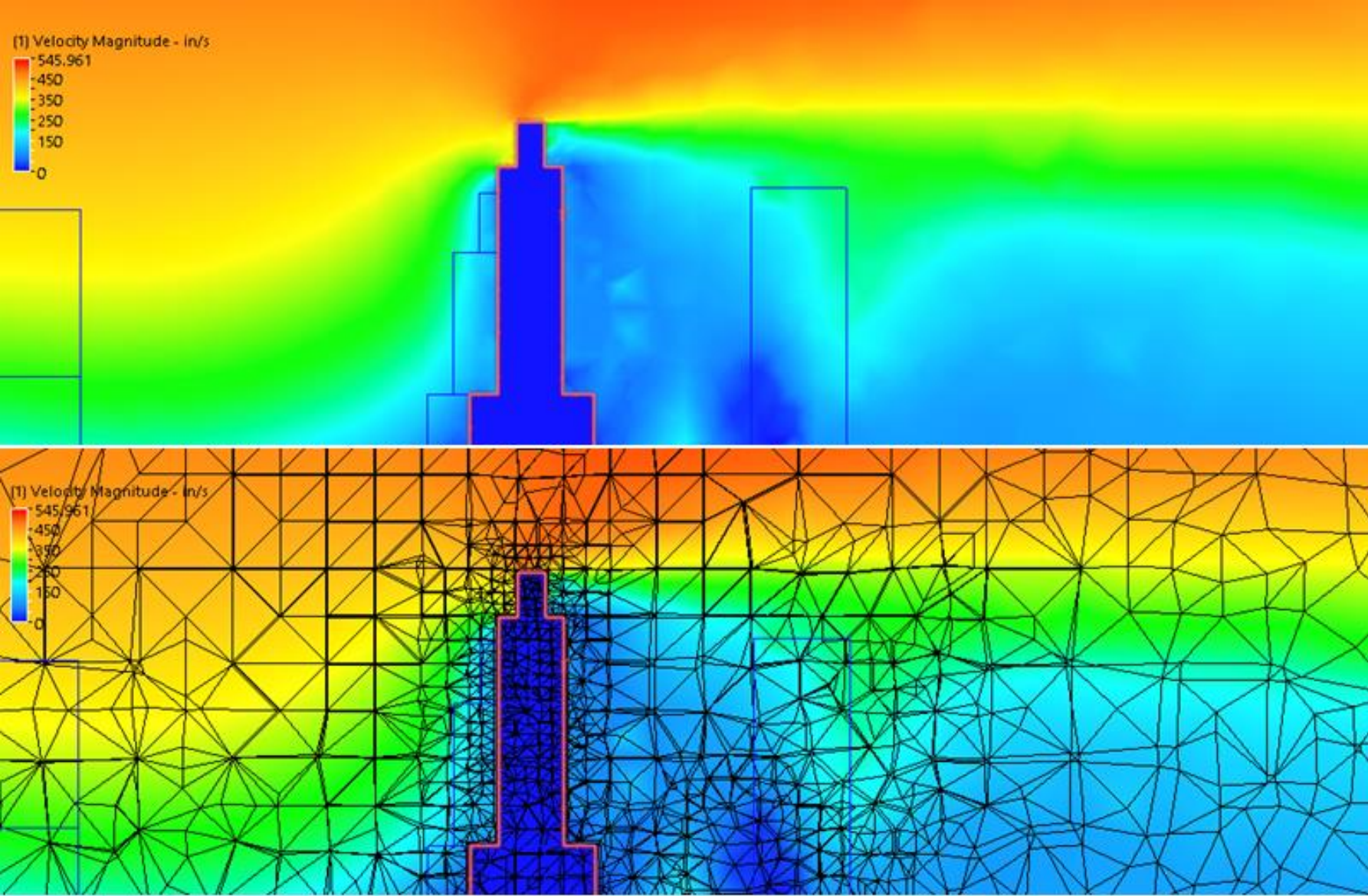
# Using Autodesk Flow Design - Model setup



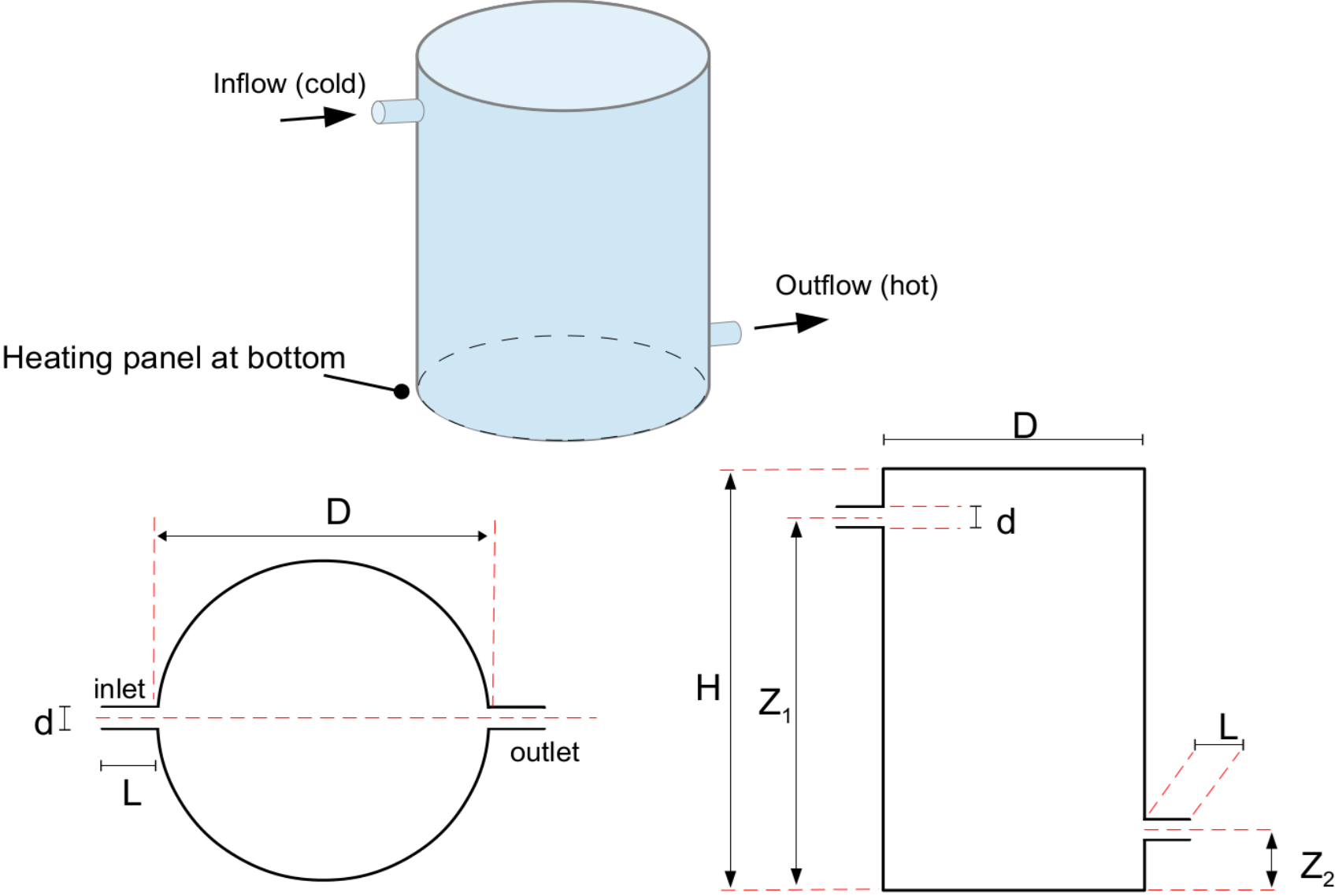
# Simulated wind speed (simplified version with fewer buildings)



# Detail around the building - wind speed

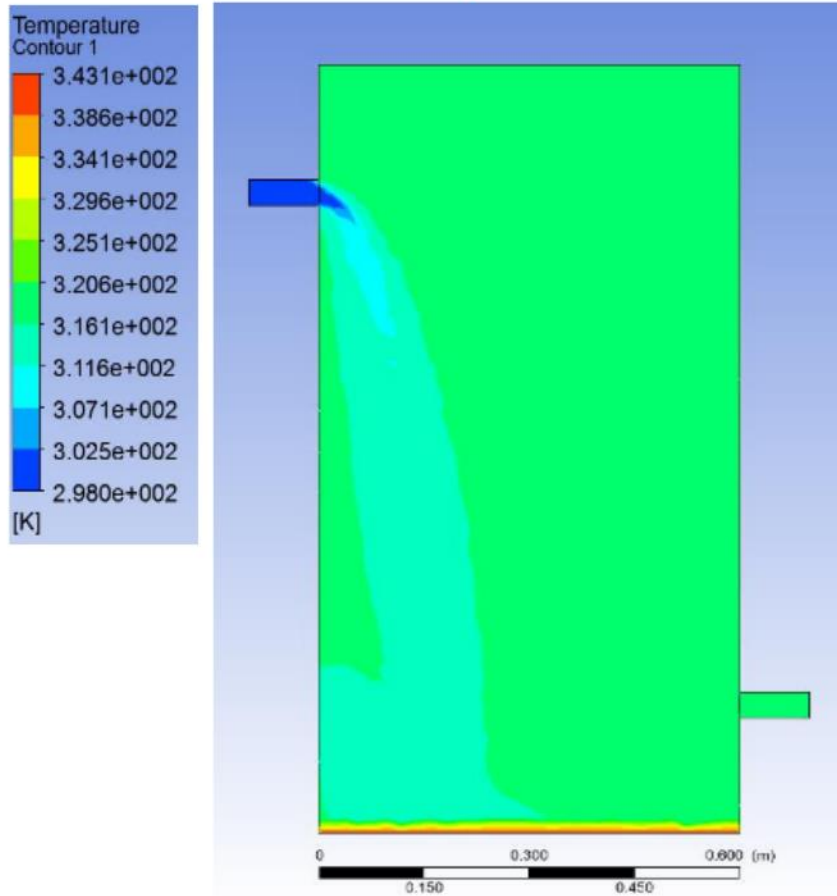


# Example 2 Internal flow - using *Ansys-Fluent*

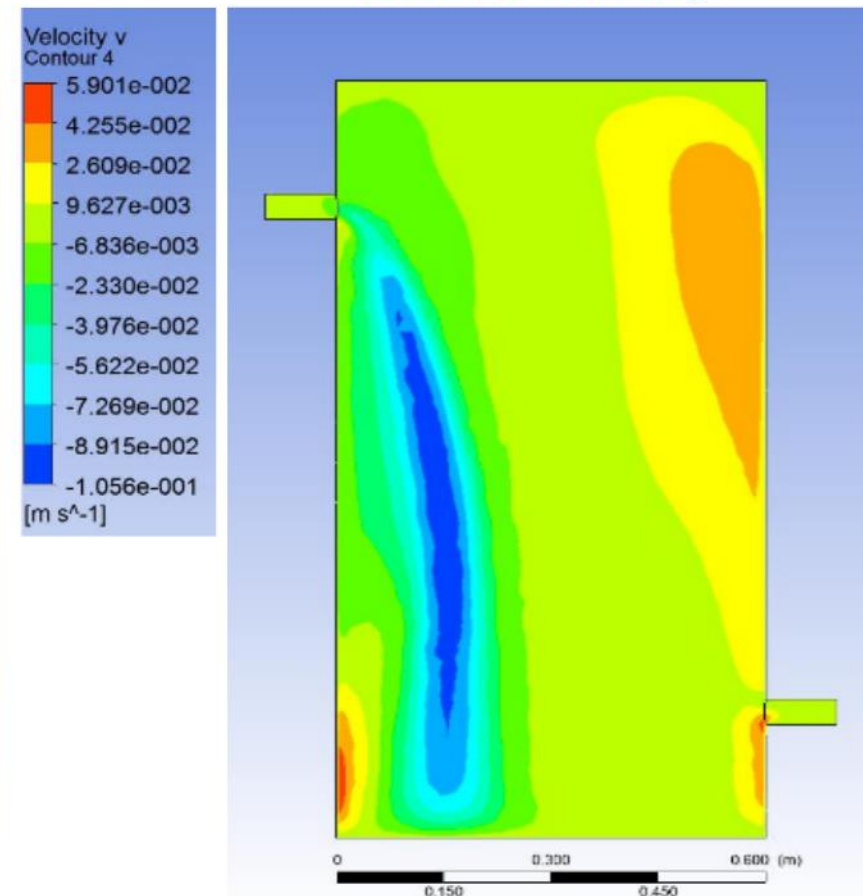


# Simulated temperature & velocity

## Temperature



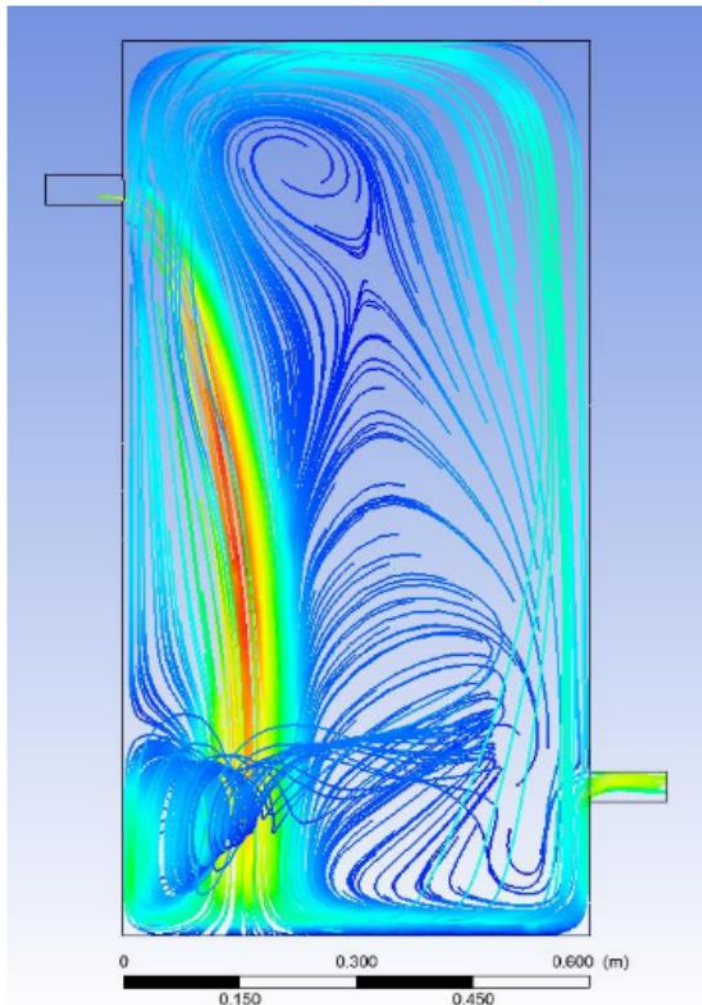
## Vertical velocity



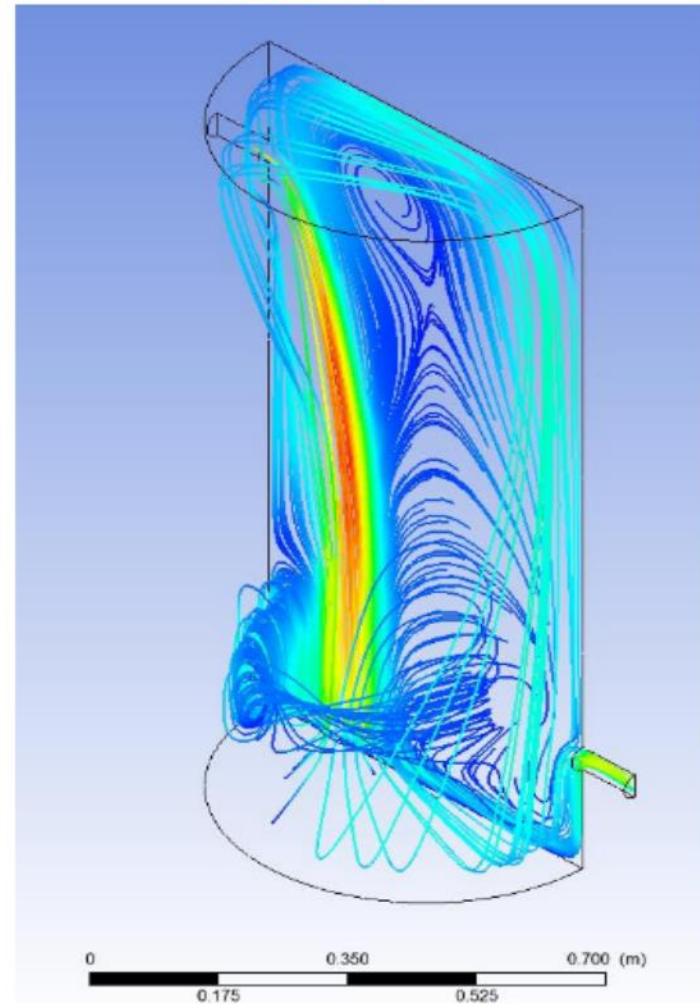
[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

# Stream lines

Plane of symmetry



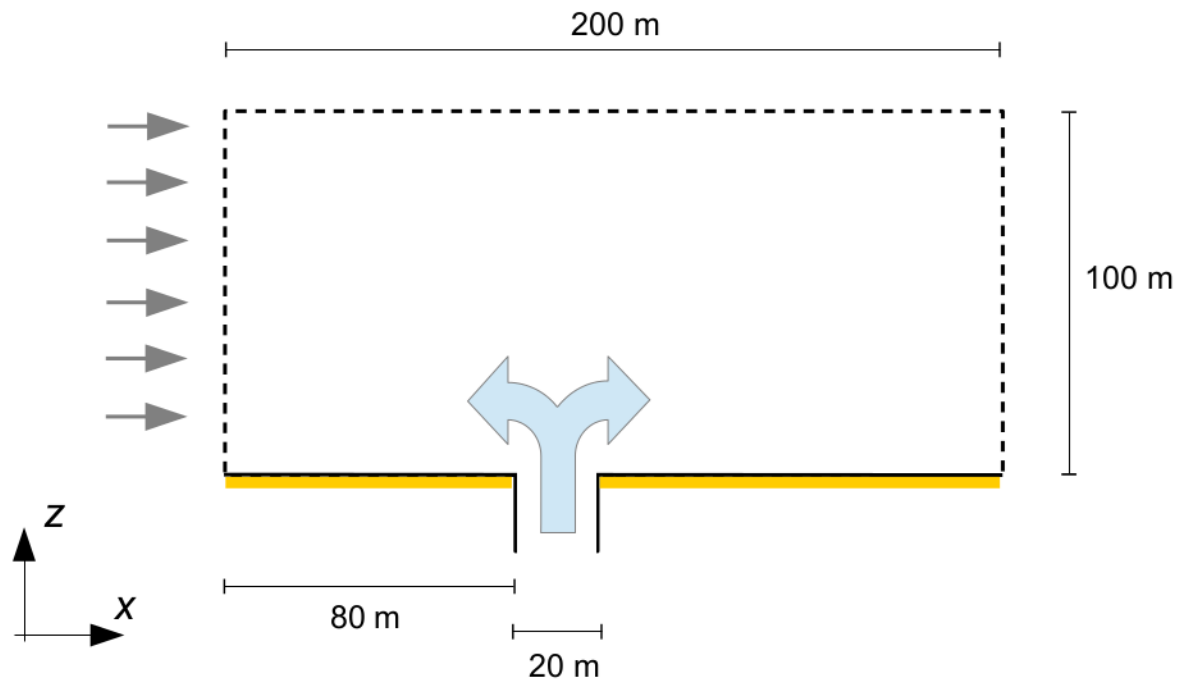
Side view



[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

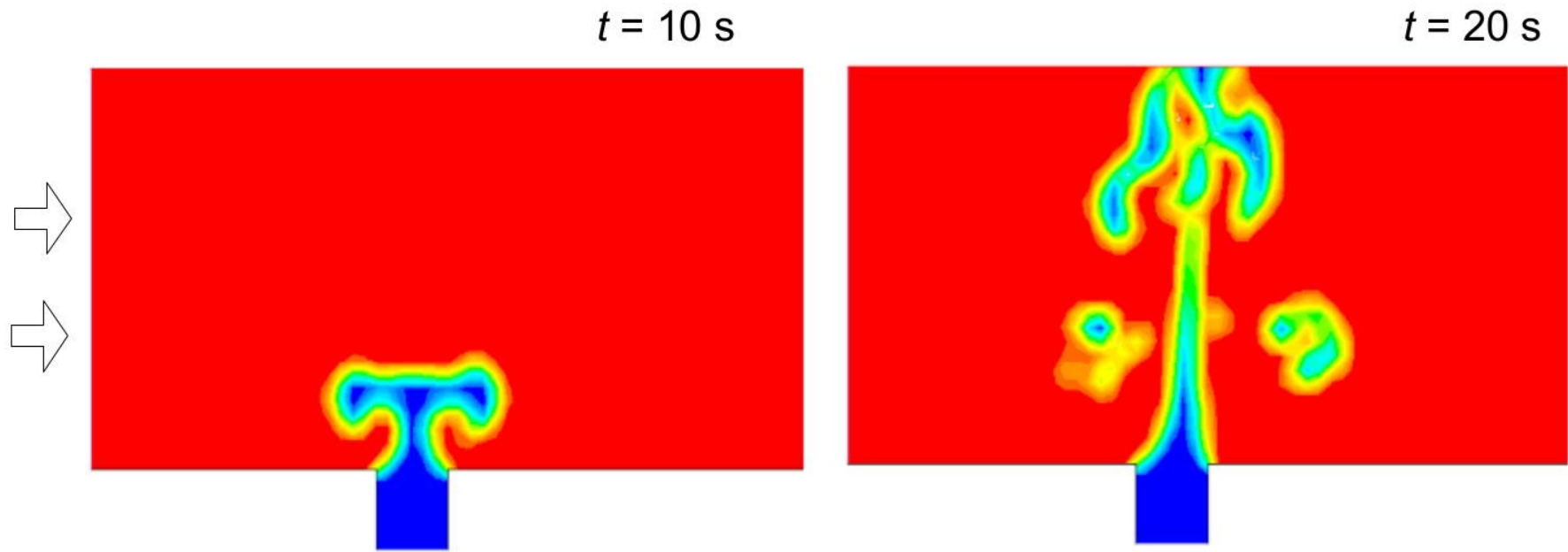
Example 3 Two-phase flow in an open domain  
- using *Ansys-Fluent* (VOF method)

*Leak of methane from an underground reservoir into open air*





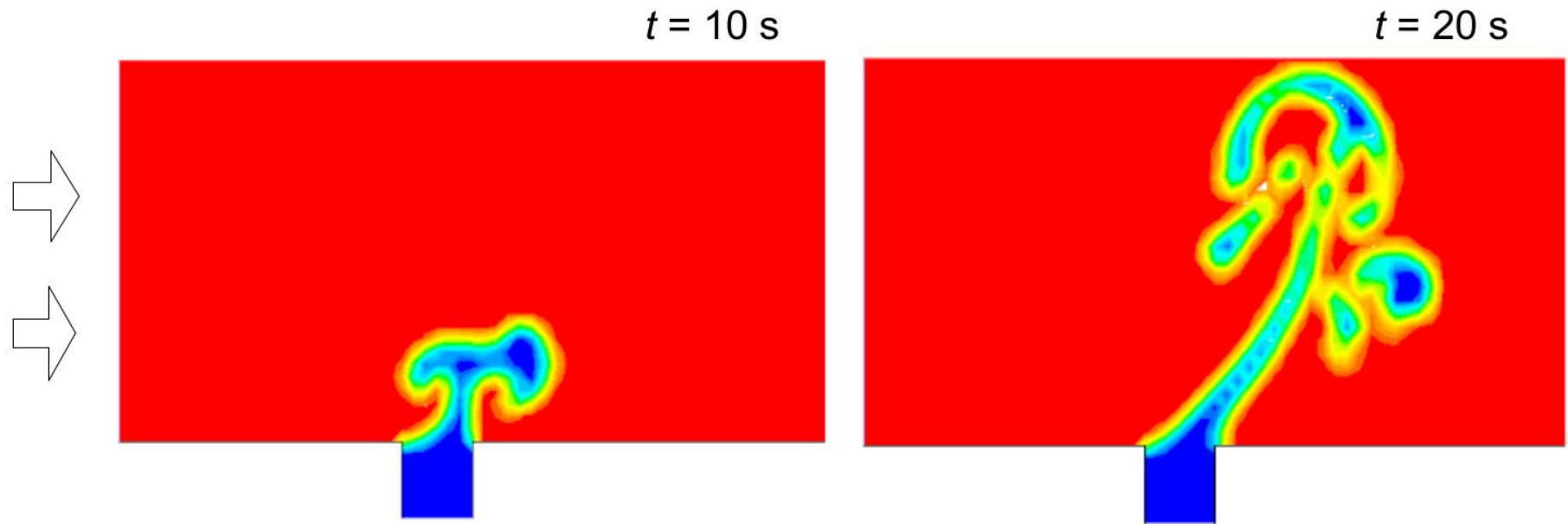
Low wind condition ( $U = 0.25 \text{ m/s}$ )



blue: methane, red = air, yellow/green = mixture

[Applied CFD course project, Fall 2015, Z. Damania (Instructor: HP Huang)]

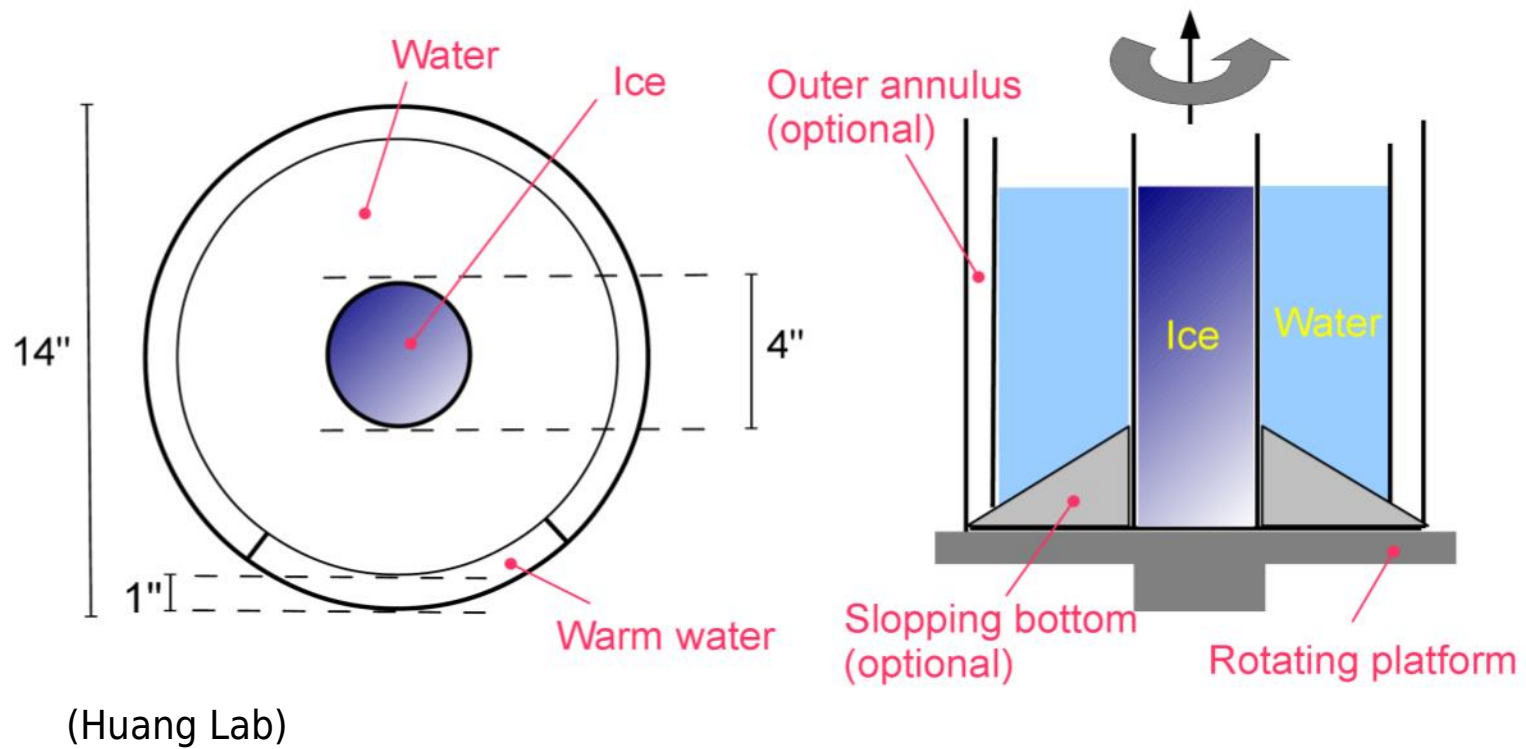
High wind condition ( $U = 0.5 \text{ m/s}$ )



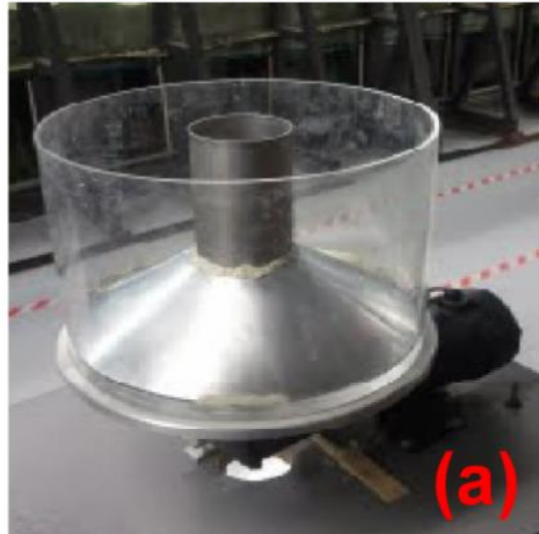
[Applied CFD course project, Fall 2015, Z. Damania (Instructor: HP Huang)]

## Example 4 Comparison of numerical simulation with lab experiment

### *Currents and waves in a rotating water tank* (which emulate large-scale environmental flows)

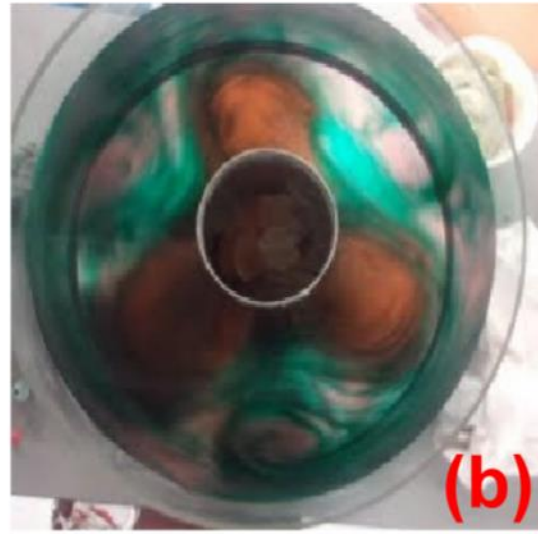


(Huang Lab)

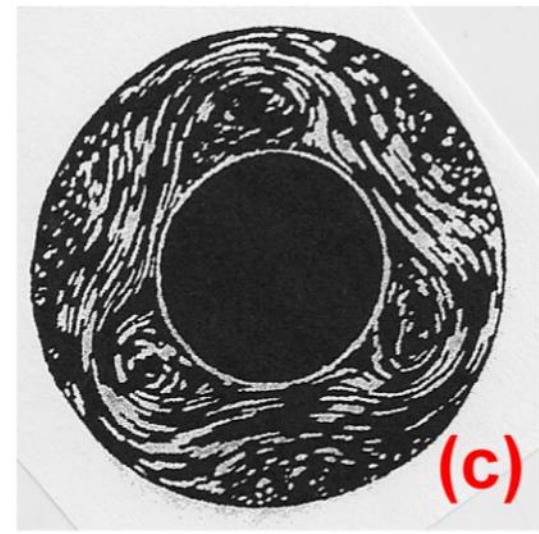


The apparatus

(Huang Lab)



Experiment which shows  
a 3-wave structure  
(visualization by color dye)



Classic result by  
Hide & Mason  
(streak photograph)

Simulate the system using *Ansys-Fluent*

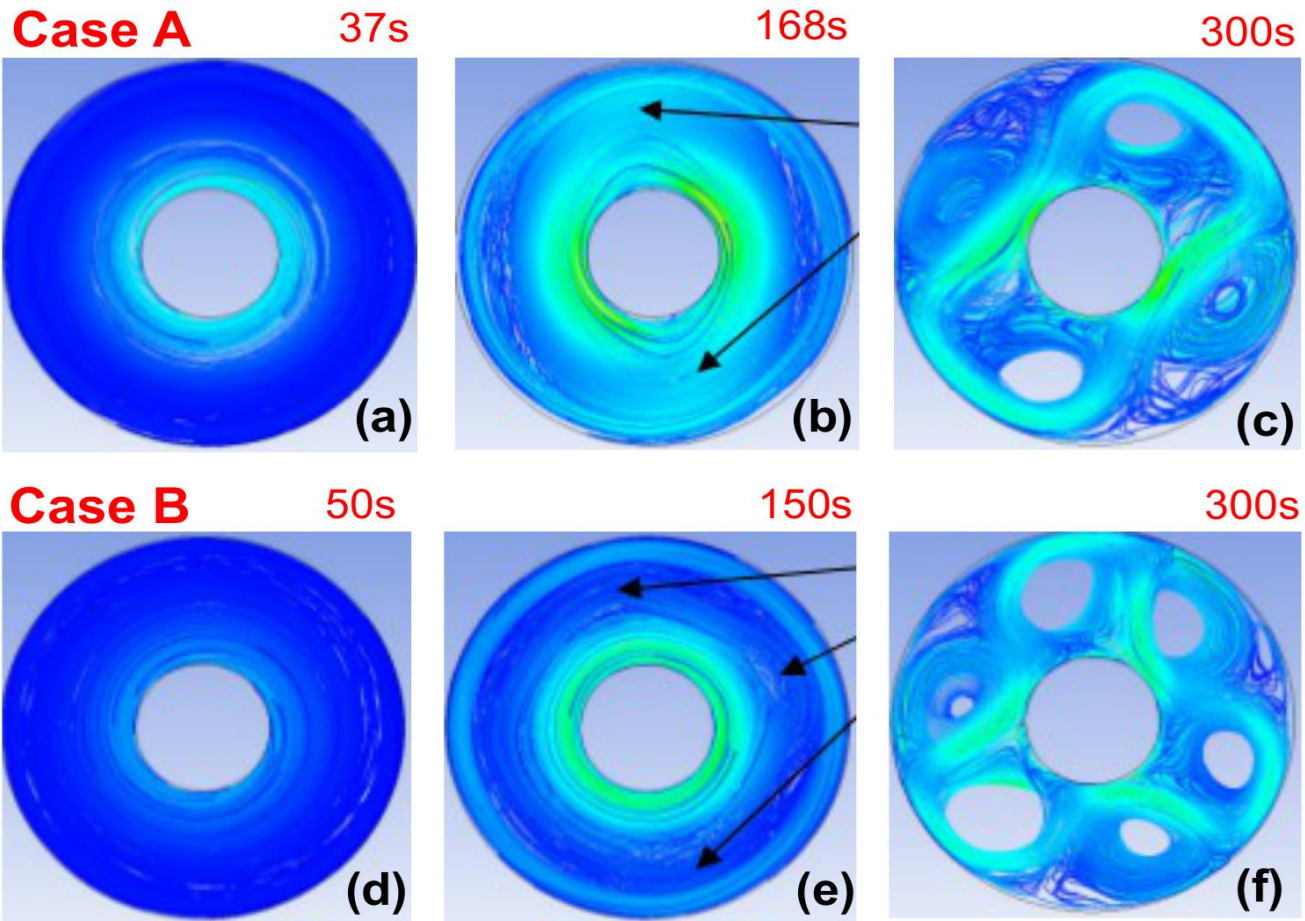


Geometry & mesh

(N. Kulkarni , 2012, Applied Project)

# The simulation

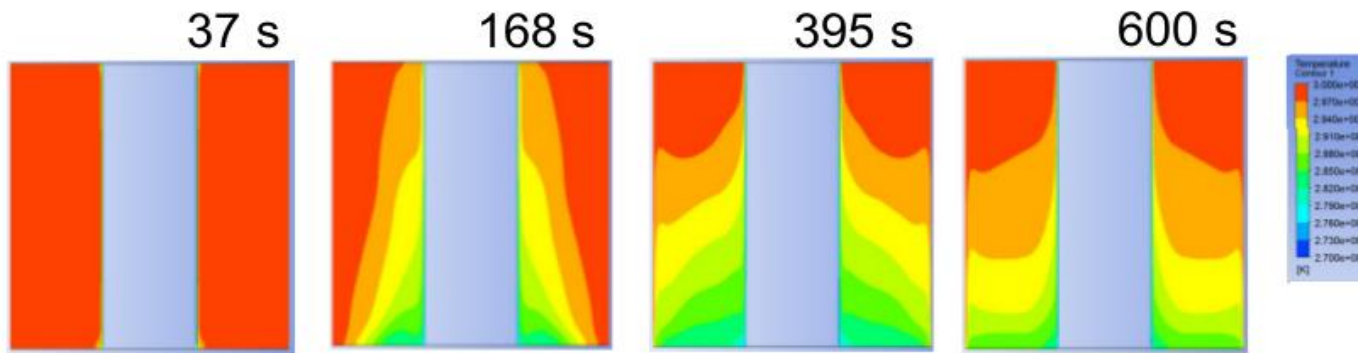
Streamlines (top view)



Case A: rotation rate = 0.942 rad/s, radial  $\Delta T = 30K$   
Case B: rotation rate = 0.942 rad/s, radial  $\Delta T = 15K$

(N. Kulkarni, 2012, Applied Project)

## Temperature: vertical cross section



Case A: rotation rate = 0.942 rad/s, radial  $\Delta T = 30\text{K}$

(N. Kulkarni, 2012, Applied Project)

## Why numerical simulation ?

- Easy and cheap to modify the apparatus
  - Produces full 3-D fields (for velocity, temperature, etc.) which are otherwise hard to measure in the lab
  - Easy to adjust the external parameters (e.g., rotation rate of the water tank) for multiple experiments
- and more ...

### Caution: Computer model $\neq$ Reality

- Finite resolution
- Inaccuracies in numerical schemes
- Incomplete representation of physical processes etc.



## Other software/tools

- While Ansys-Fluent has its own post-processing tools, the output of simulation can also be analyzed using external software (e.g., Matlab)
- Ansys-Fluent has a built-in module for CAD (DesignModeler) that it is not necessary to use an external CAD software to complete the projects. Nevertheless, students may choose to use their own favorite CAD tools (e.g., Solidworks) to build the geometry and import it to Ansys.