

MAE 598/494 Applied Computational Fluid Dynamics
Fall 2017 Tuesday/Thursday 1:30-2:45, Classroom: SCOB 152

Instructor: Huei-Ping Huang (hp.huang@asu.edu), ERC 359
Office hours: Monday 4-5 PM, Tuesday 3-5 PM, or by appointment

Course website <http://www.public.asu.edu/~hhuang38/ACFD2017.html>

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 mandatory.

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-Fluent is available on the computers at GWC 481/483 computing lab. You will have access to the lab. (Detail forthcoming.)

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 most projects in this class.

Plan for the semester (Also see syllabus)

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 - regular tasks ~ 75%
Projects & homework - credit from bonus pool ~ 15%
Final exam (in-class test) 10%

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

Most projects will include a **bonus section** with one or a few problems that require extra effort to solve. For each problem, a bonus pool (with a certain fixed bonus points) will be divided by those who provide correct answers. This is modified from a system that was successfully tested in the past 2 years. Instructor will explain why the system is needed, and why it is useful.

Occasionally, a bonus question might be asked in class. A correct or useful answer will be awarded minor bonus credit.

The required work for the projects will be different for participants of MAE598 and MAE494. Details will be given in the individual assignments.

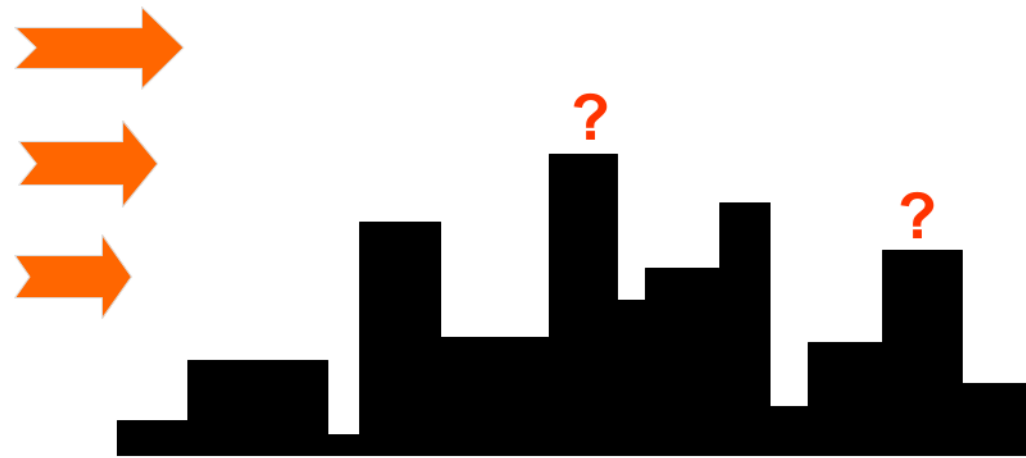
Please make sure that you are familiar with ASU policies on academic integrity: <https://provost.asu.edu/academicintegrity>

Some examples of computational fluid dynamics using commercial CFD solvers

Many commercial/industrial CFD solvers are available on the market. This class will use almost exclusively Ansys-Fluent but might include one exercise on comparing Fluent with other solvers (Comsol, AcuSolve, Abacus, Autodesk Flow Design, etc.)

Example 1 External flow - using *Autodesk Flow Design*

Assessment of rooftop wind power potential

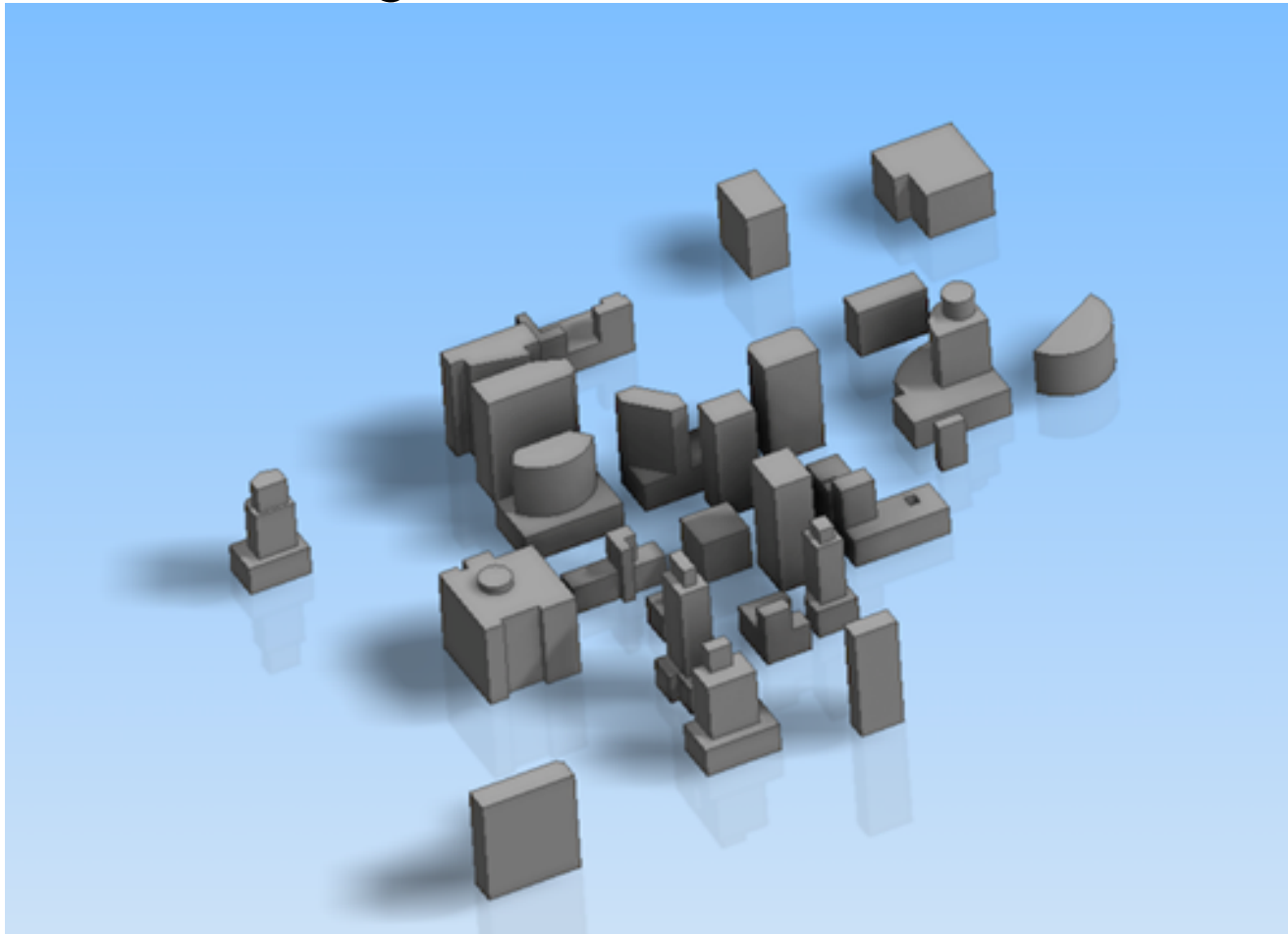


(Urban community, college campus, etc.)

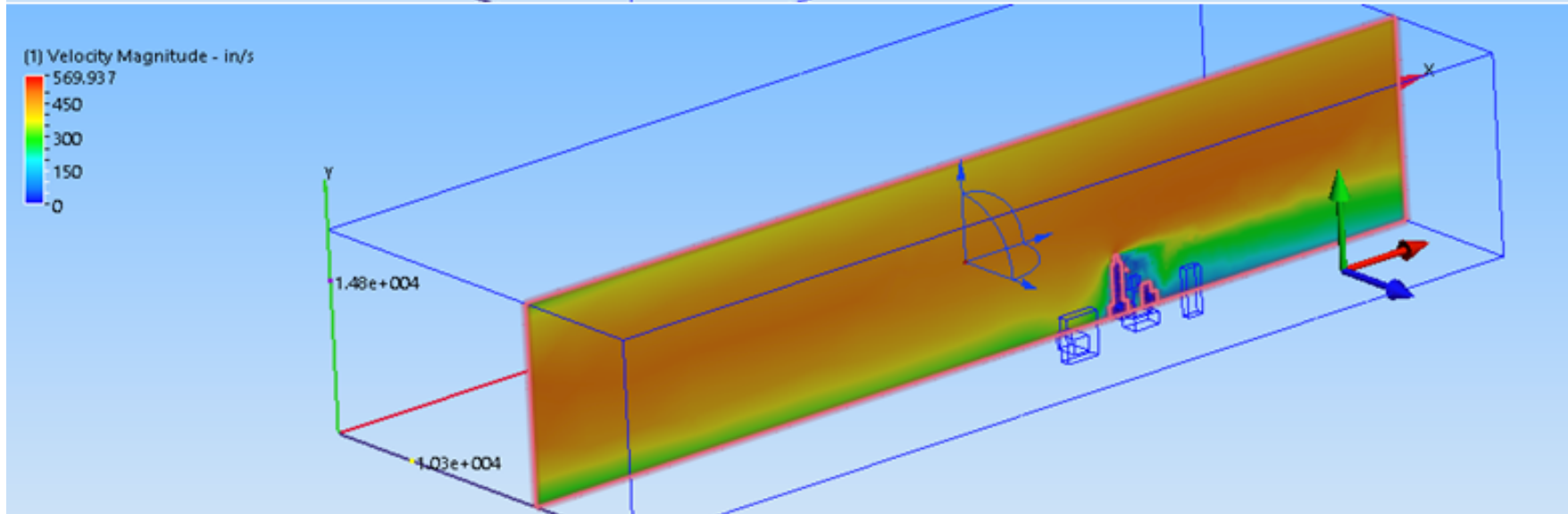
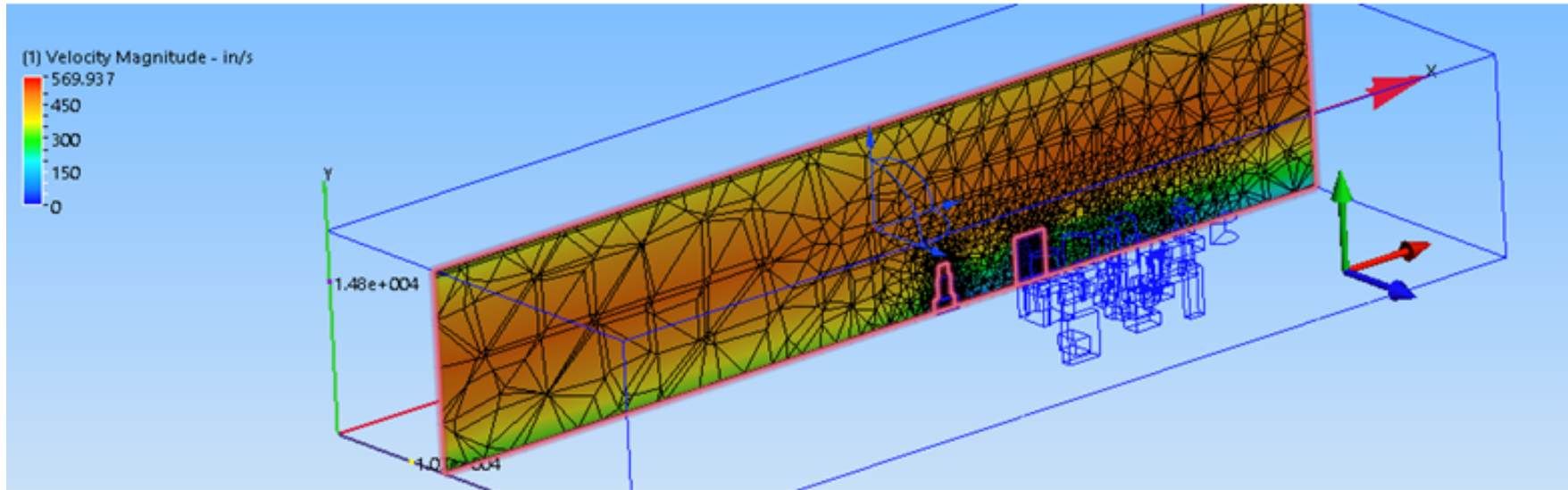
Simulation for Downtown Phoenix

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

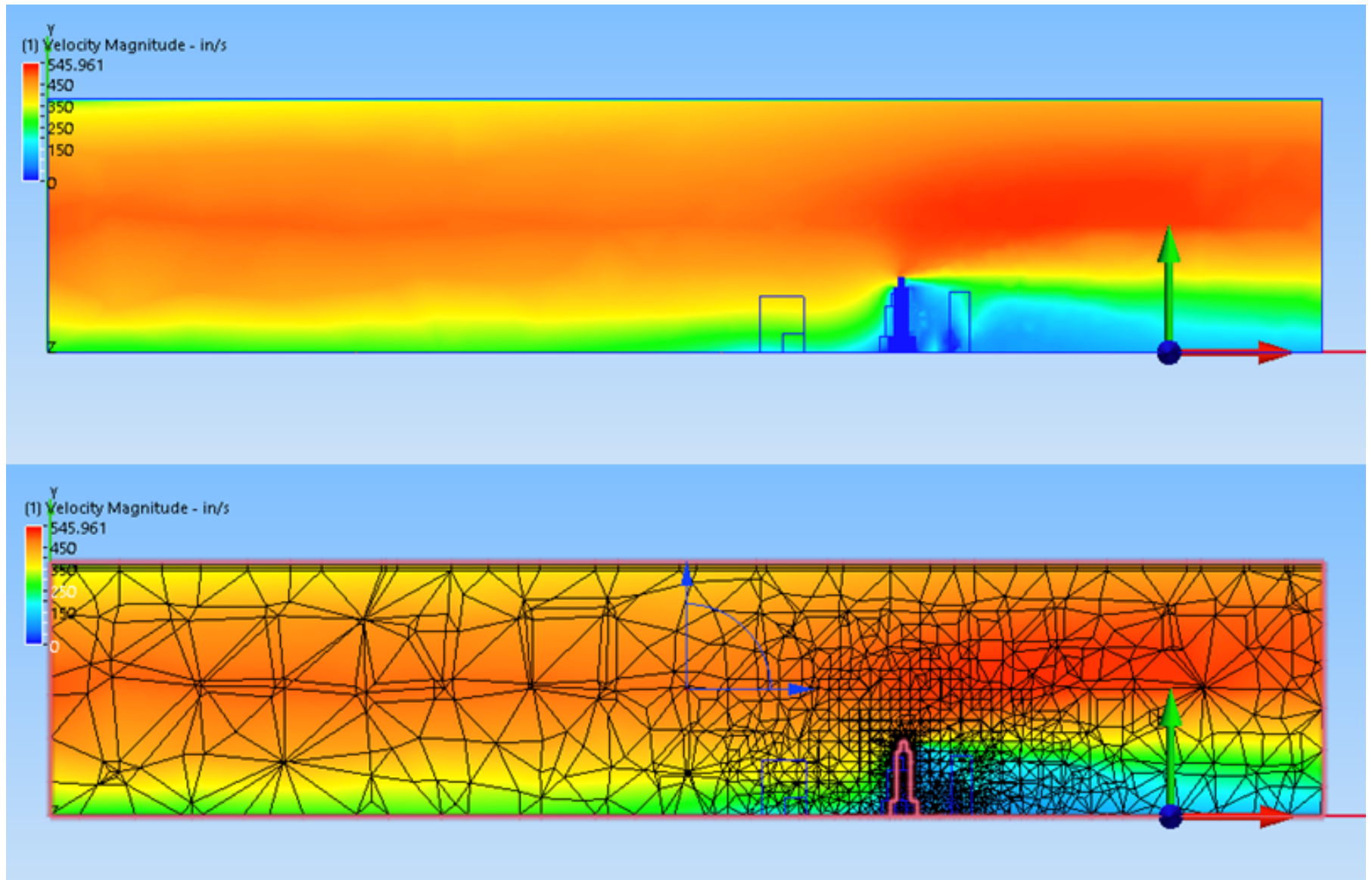
- 3-D geometric data from Google Earth
- Keep 23 tallest buildings for flow simulation



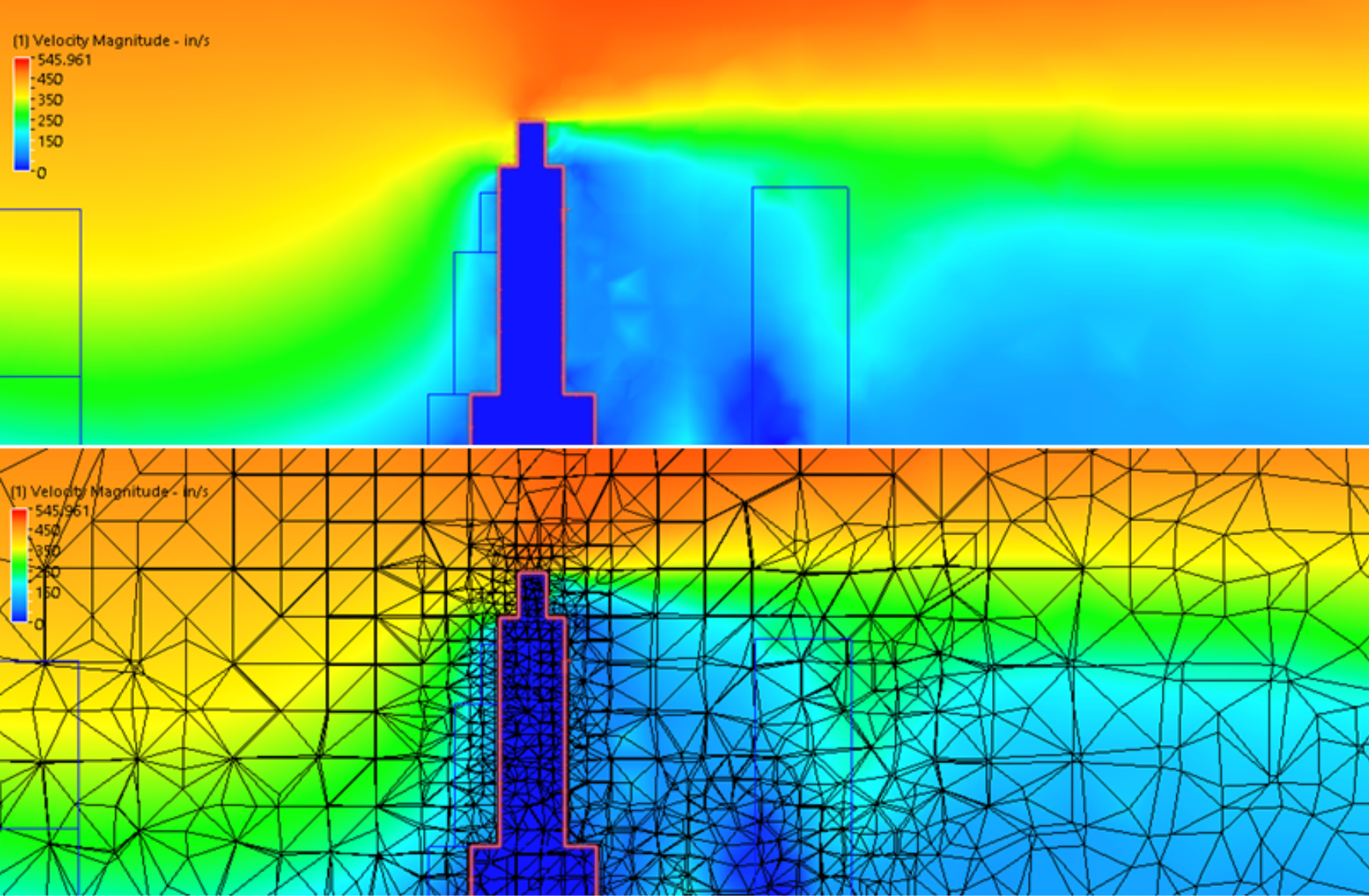
Using Autodesk Flow Design - Model setup



Simulated wind speed (simplified version with fewer buildings)

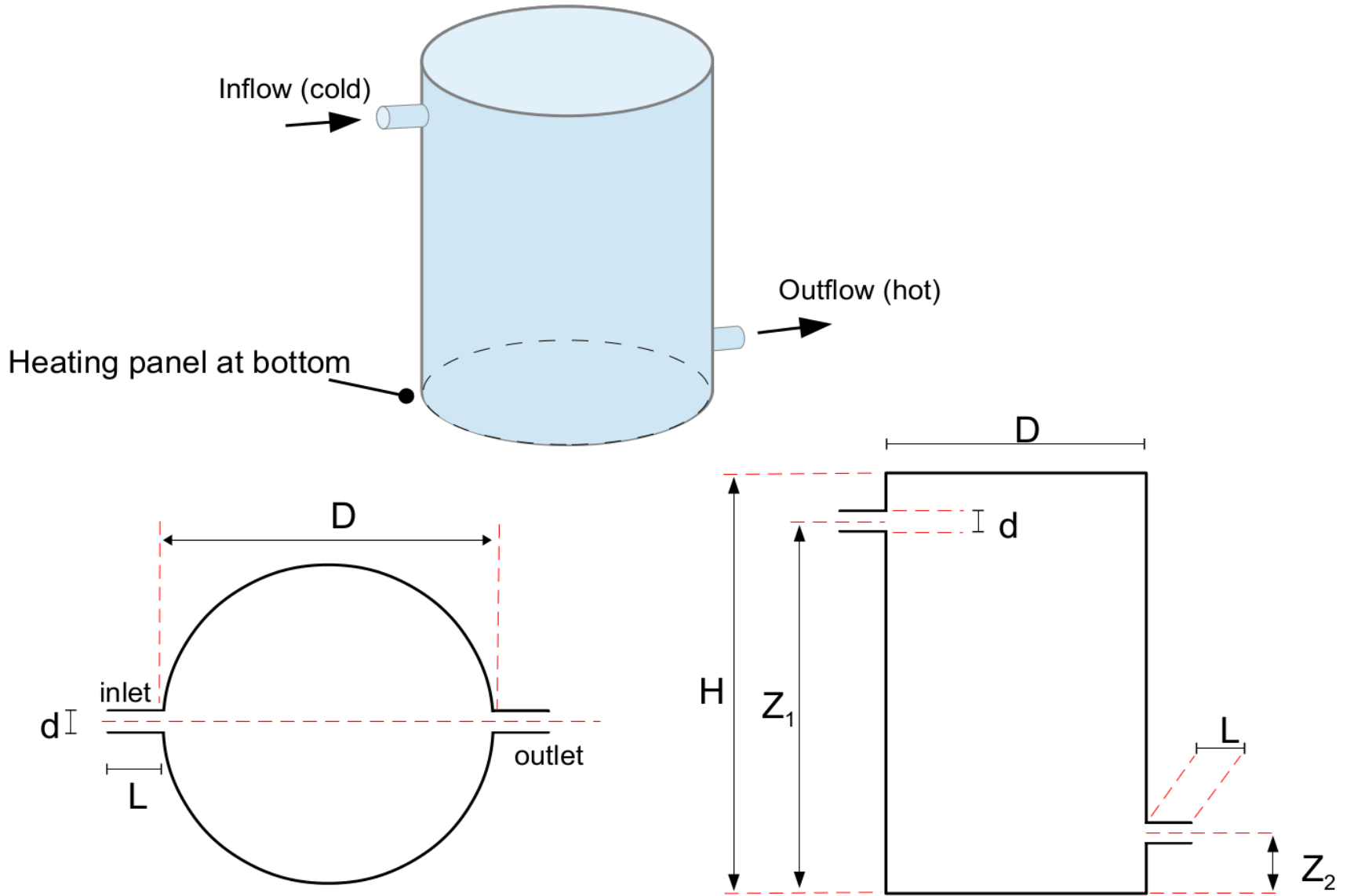


Detail around the building - wind speed



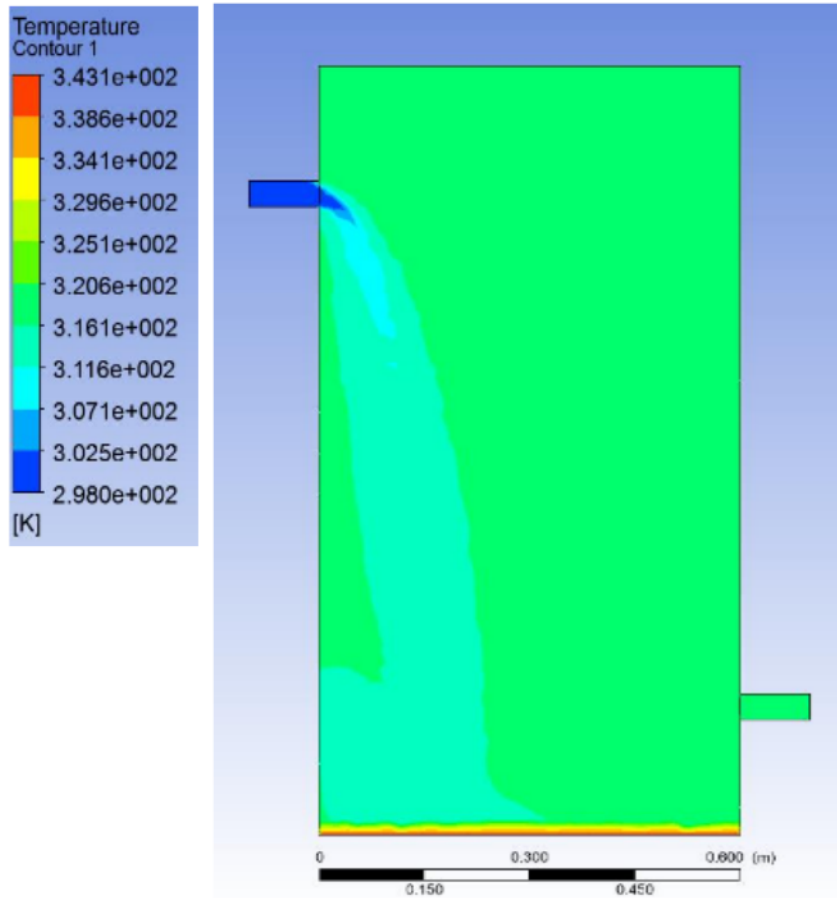
Example 2 Internal flow - using *Ansys-Fluent*

Temperature and flow within a water heater

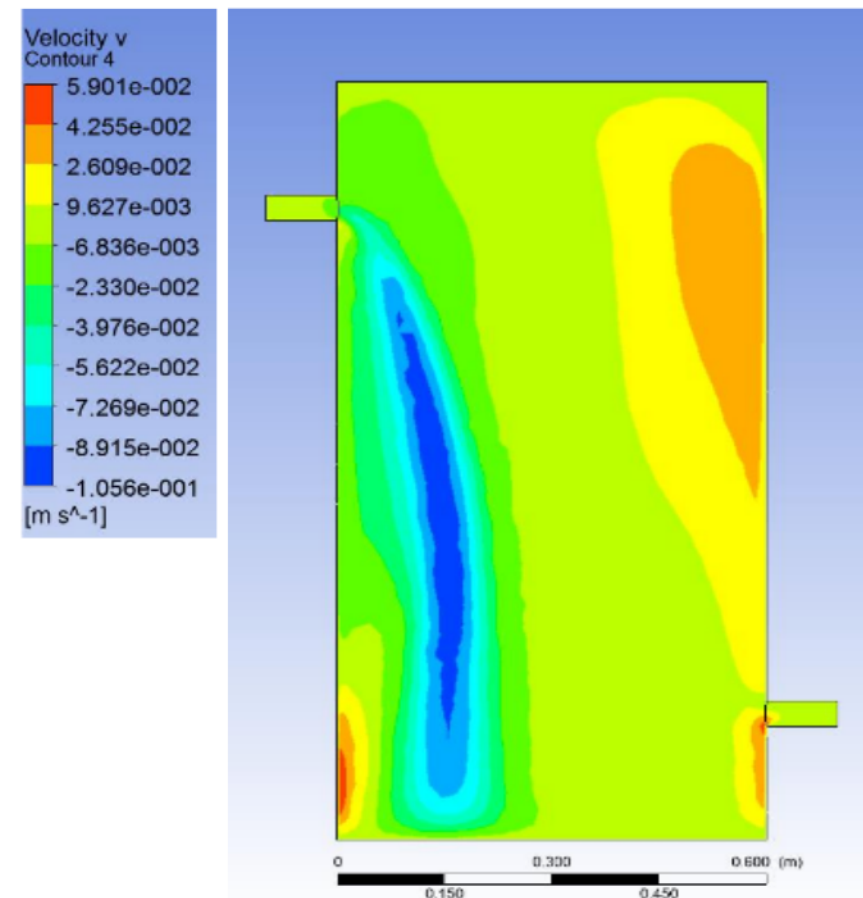


Simulated temperature & velocity

Temperature



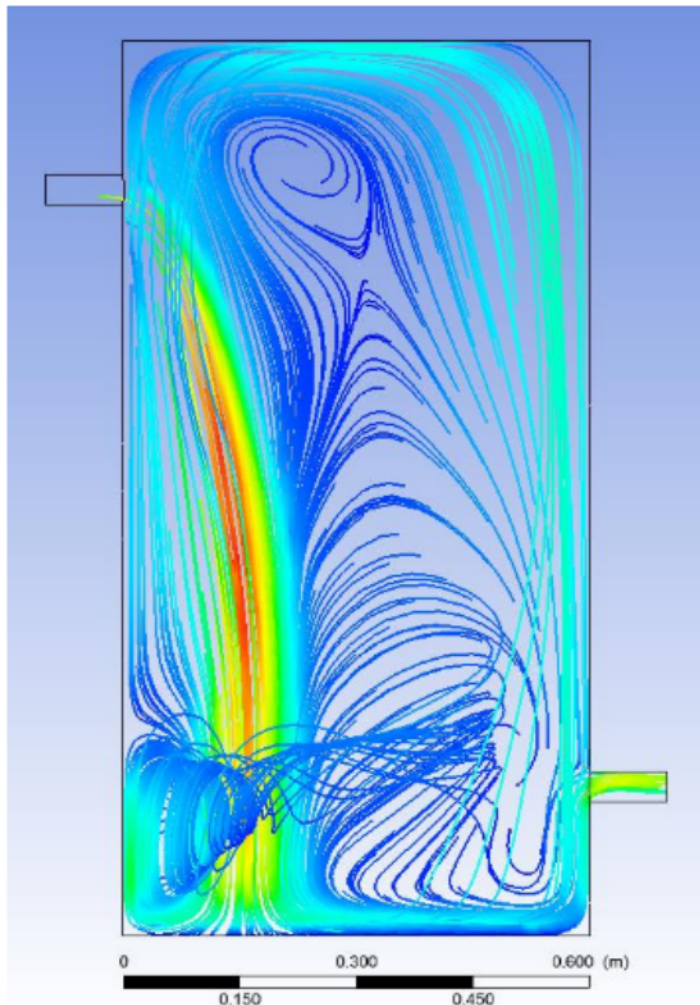
Vertical velocity



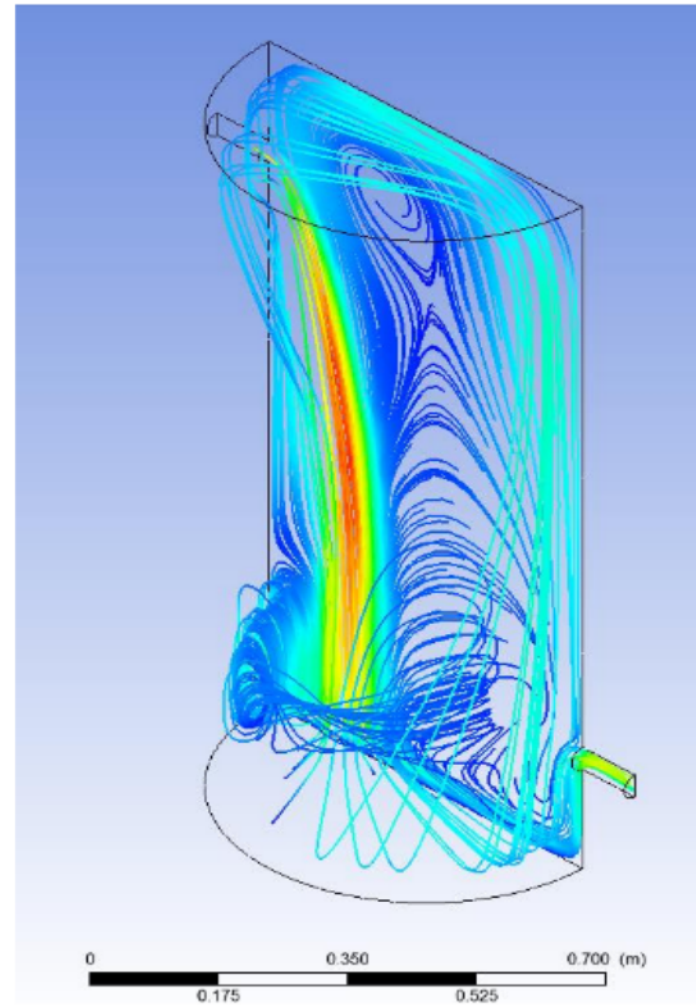
[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

Stream lines

Plane of symmetry



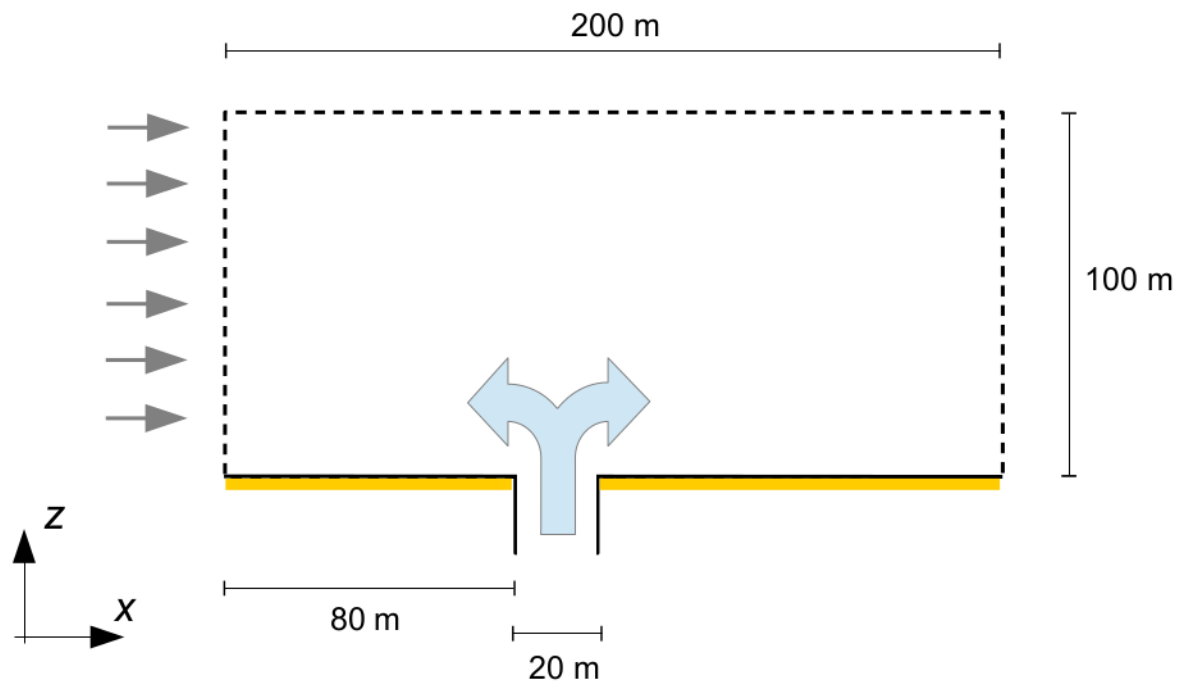
Side view



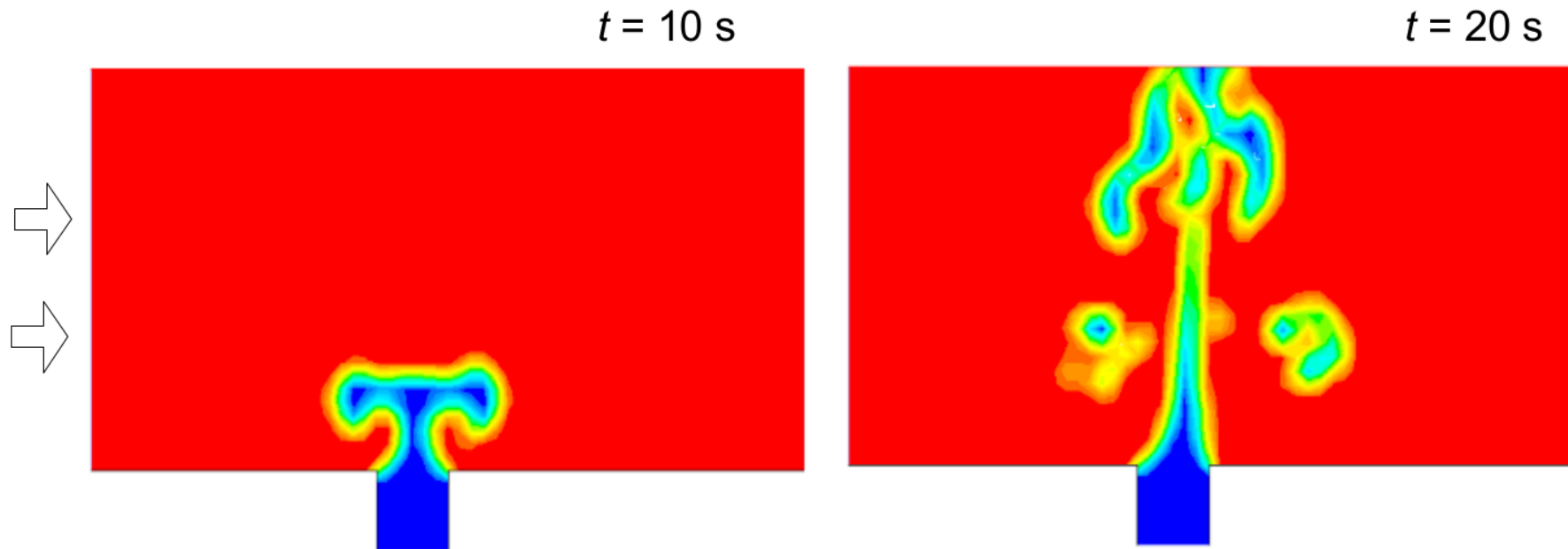
[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

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

Leak of methane from an underground reservoir into open air



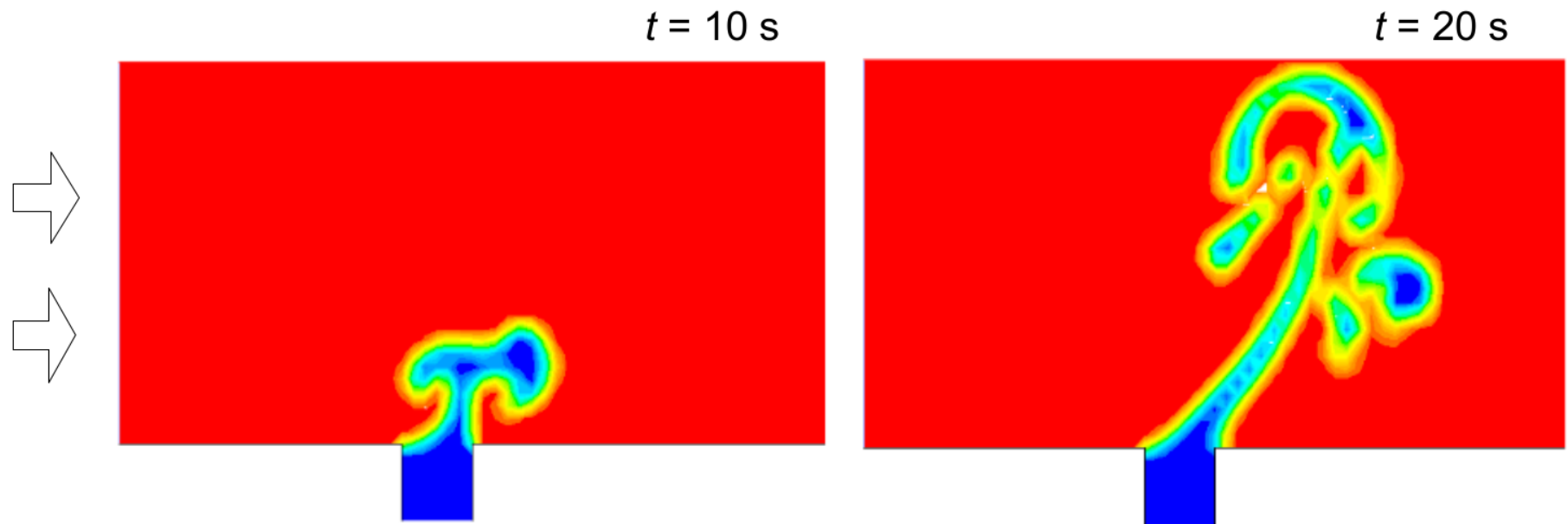
Low wind condition ($U = 0.25 \text{ m/s}$)



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

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

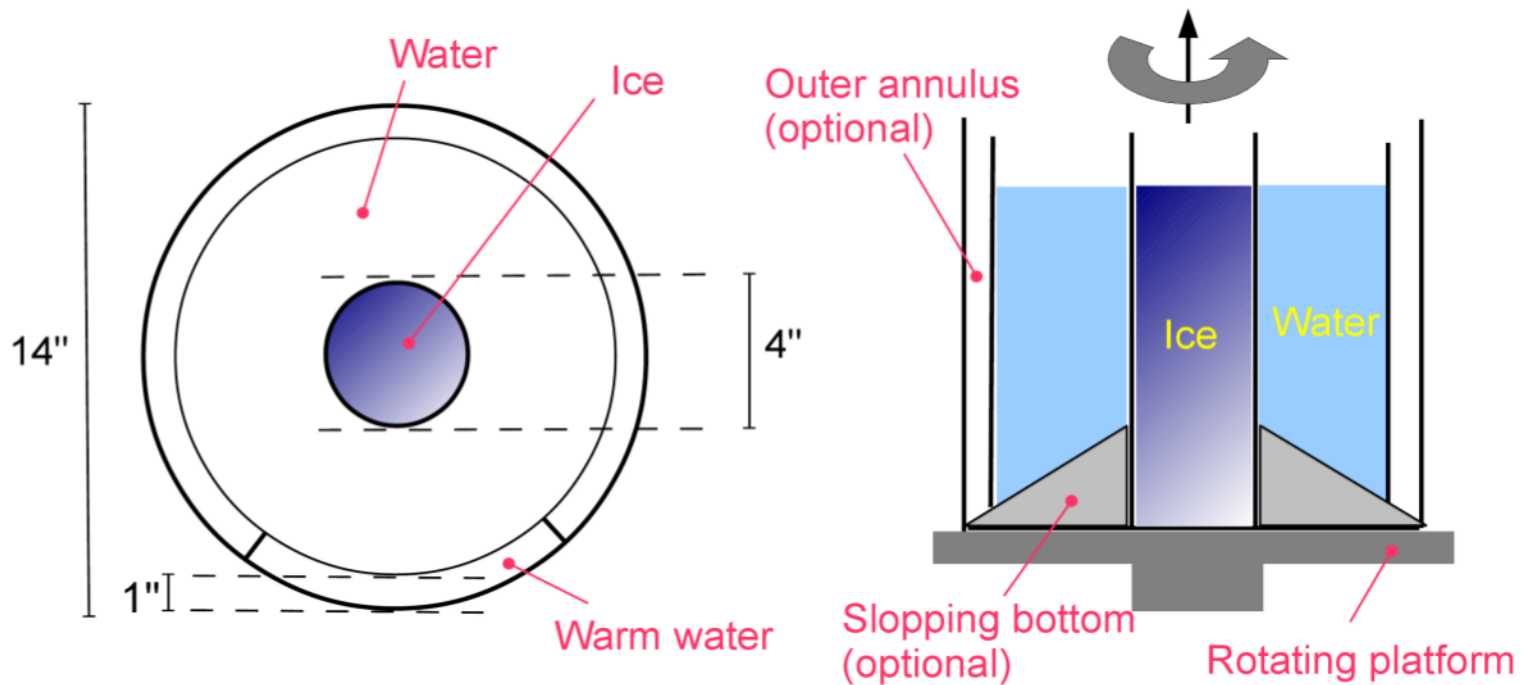
High wind condition ($U = 0.5 \text{ m/s}$)



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

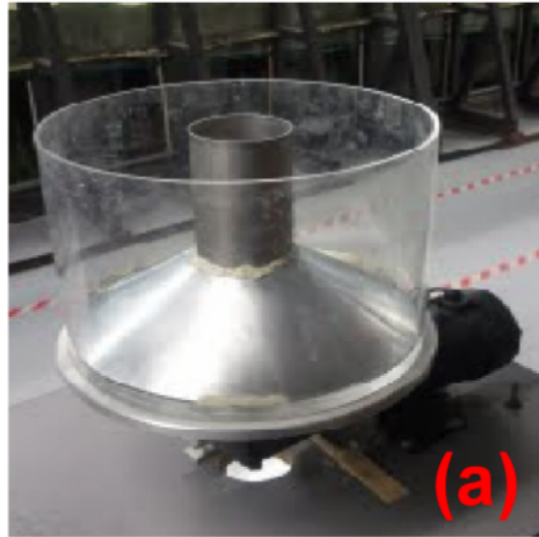
Example 4 Comparison of numerical simulation with lab experiment

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



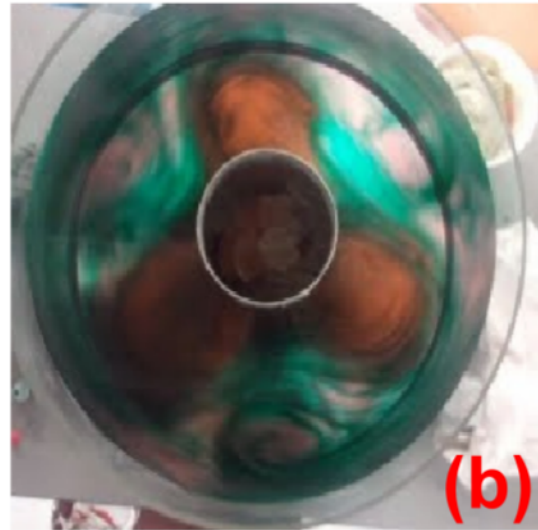
(Huang Lab)

(Huang Lab)

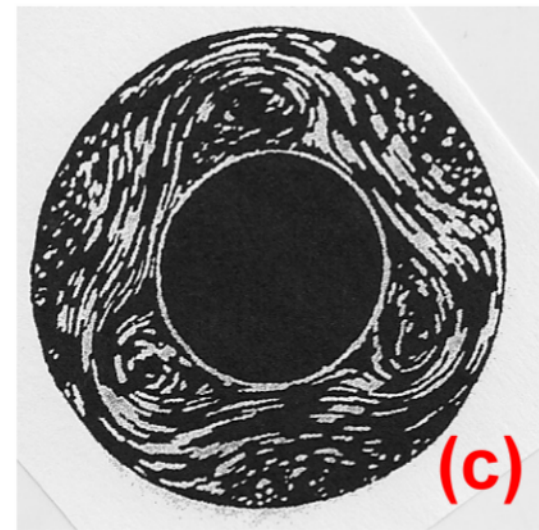


The apparatus

(Huang Lab)

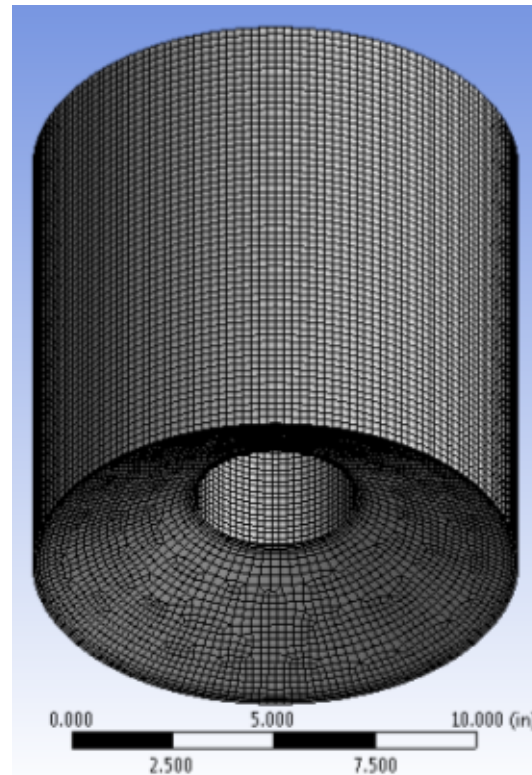


Experiment which shows
a 3-wave structure
(visualization by color dye)



Classic result by
Hide & Mason
(streak photograph)

Simulate the system using *Ansys-Fluent*

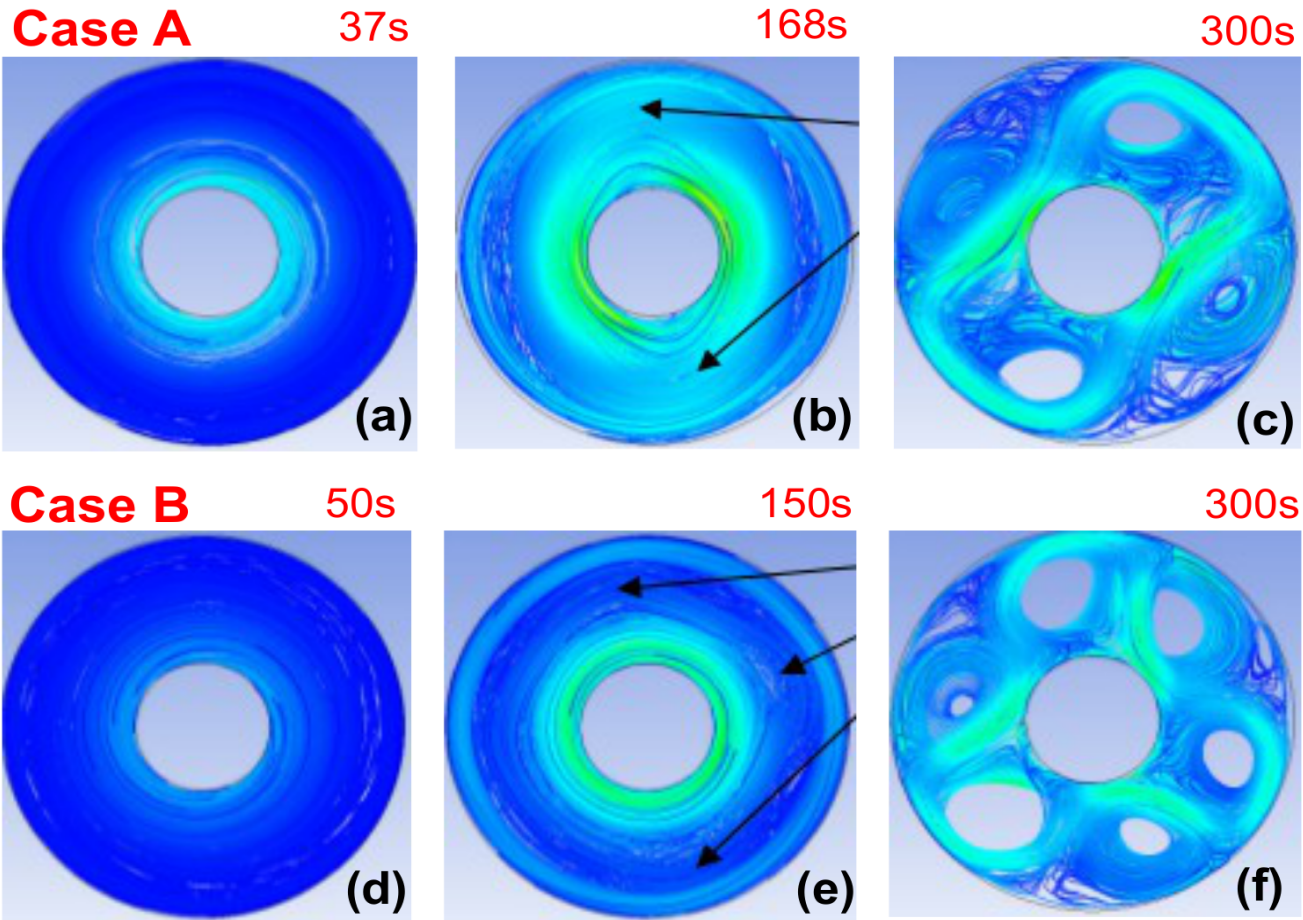


Geometry & mesh

(N. Kulkarni , 2012, Applied Project)

The simulation

Streamlines (top view)

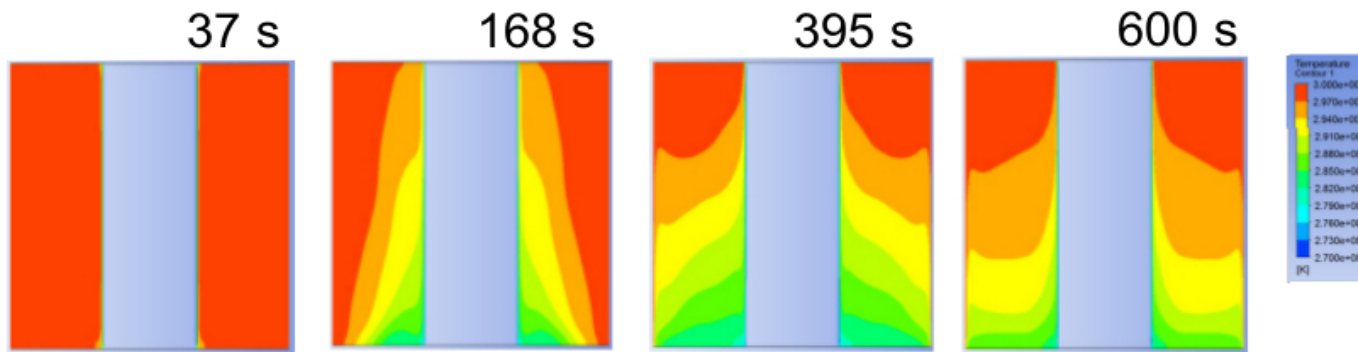


Case A: rotation rate = 0.942 rad/s, radial $\Delta T = 30K$

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.

Facilities & software

Ansys-Fluent will be the main tool for all projects

- GWC 481/483 Computing Lab
Ansys-Fluent, Solidworks, etc., available on each computer
Needs to apply for access
- Student version of Ansys can be downloaded from company website
Has limits on # of nodes and # of modules to open at once
- While Ansys-Fluent has its own post-processing tools, the output of simulation can also be analyzed with external software (e.g., Matlab)