MAE 598/494 Topic: Applied Computational Fluid Dynamics

Fall 2015 Tuesday/Thursday 1:30-2:45, Classroom: SCOB 250

Instructor: Huei-Ping Huang (hp.huang@asu.edu), ERC 359

Office hours: Monday 3-5 PM, Tuesday 3-5 PM, or by appointment

Course website http://www.public.asu.edu/~hhuang38/MAE598.html

Topics to cover

- Techniques for solving incompressible and compressible flow equations using commercial/industrial solvers (We will use Ansys-Fluent)

- Computer-aided analysis of fluid systems

- Applications to thermofluid system engineering

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

Planned activities

We will run two threads concurrently through the semester:

Lecture thread fills the background knowledge on fluid mechanics and numerical methods. No exam, but attendance is mandatory. Excessive absence without a proper reason will result in a failing grade.

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.) You may also purchase a student version of the license of Fluent to run on your own computer.

(This is a repeat of 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 general methods for analyzing fluid systems (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: Treatment of turbulent flow; Meshing and grid convergence
   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

Grade: Projects & homework 75%,
Class contribution ("bonus pool") 15%,
Audit of work 10%

What are "Class contribution" and "Audit of work"?

Detailed written explanations will be given before the first opportunity arises for you to earn credit from those two categories.

Roughly speaking (for now), you earn credit from "Class contribution" by

(1) Working on optional mini assignments that are sanctioned by the instructor and available for the whole class to do

(2) Directly participating in specific in-class discussion sanctioned by the instructor

The assignments in (1) will be given in class or as additional questions in the projects/homework

The tasks in both (1) and (2) will be closely related to classroom discussion, including discussion on the projects.

While an individual assignment in (1) is optional, through the semester everyone is expected to do some extra work of this kind. Thus, the category of "Class contribution" is essentially mandatory.

"Audit of work" consists of a mini project that tests your basic skill in using Ansys-Fluent. It will be administered as an in-class "instant challenge".

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

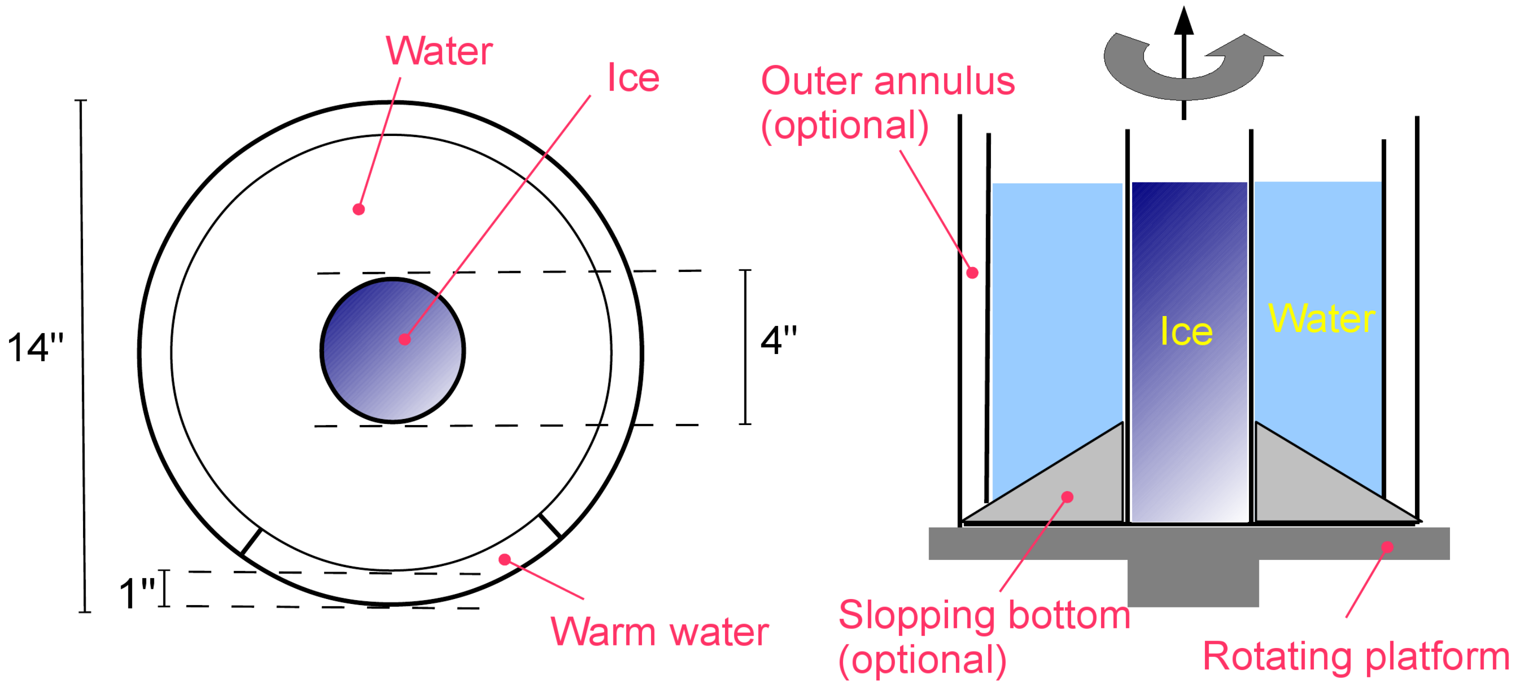
Specific rules concerning collaboration for projects/homework will be released along with the release of each assignment.

Please make sure that you are familiar with ASU policies on academic integrity

https://provost.asu.edu/academicintegrity

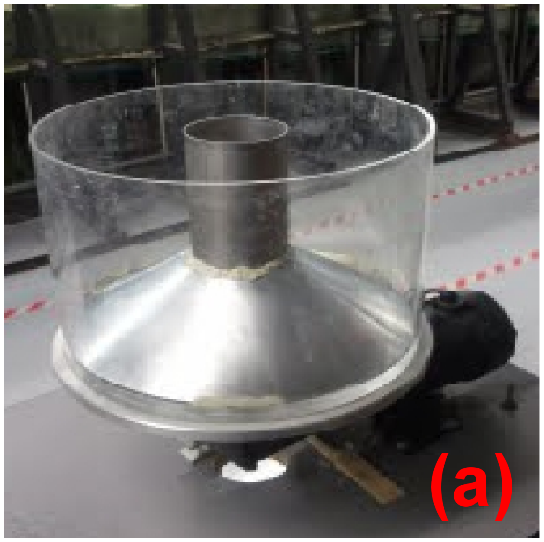
Examples of fluid simulations using Ansys-Fluent

Example 1. Currents and waves in a rotating water tank
(which emulate large-scale environmental flows)



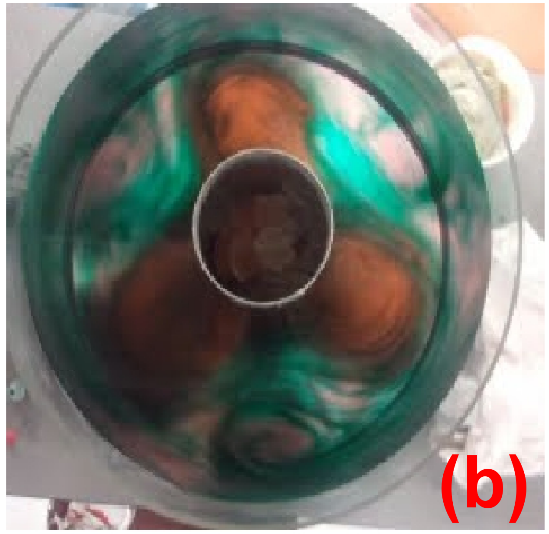
(Huang Lab)

(Huang Lab)

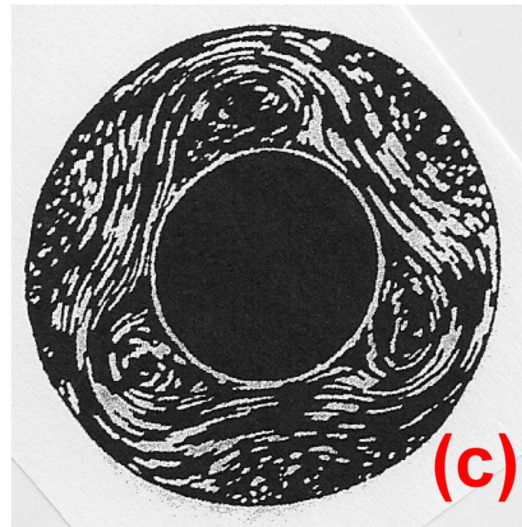


The apparatus

(Huang Lab)

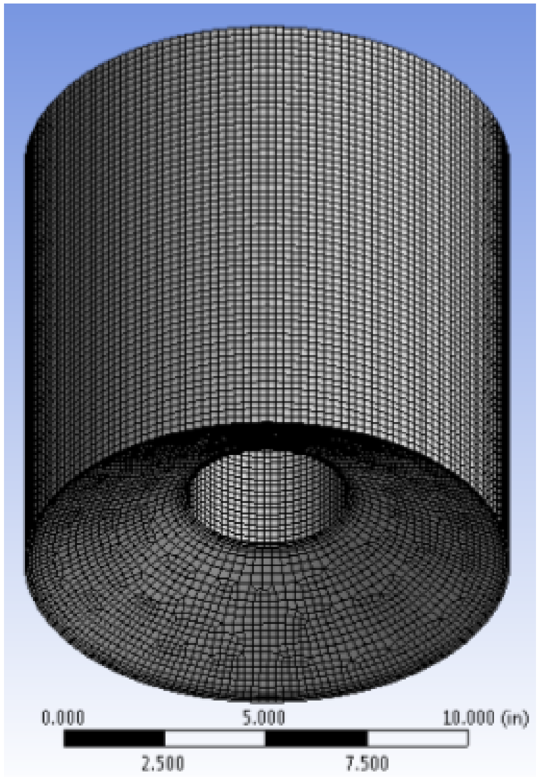


Experiment which shows a 3-wave structure (visualization by color dye)



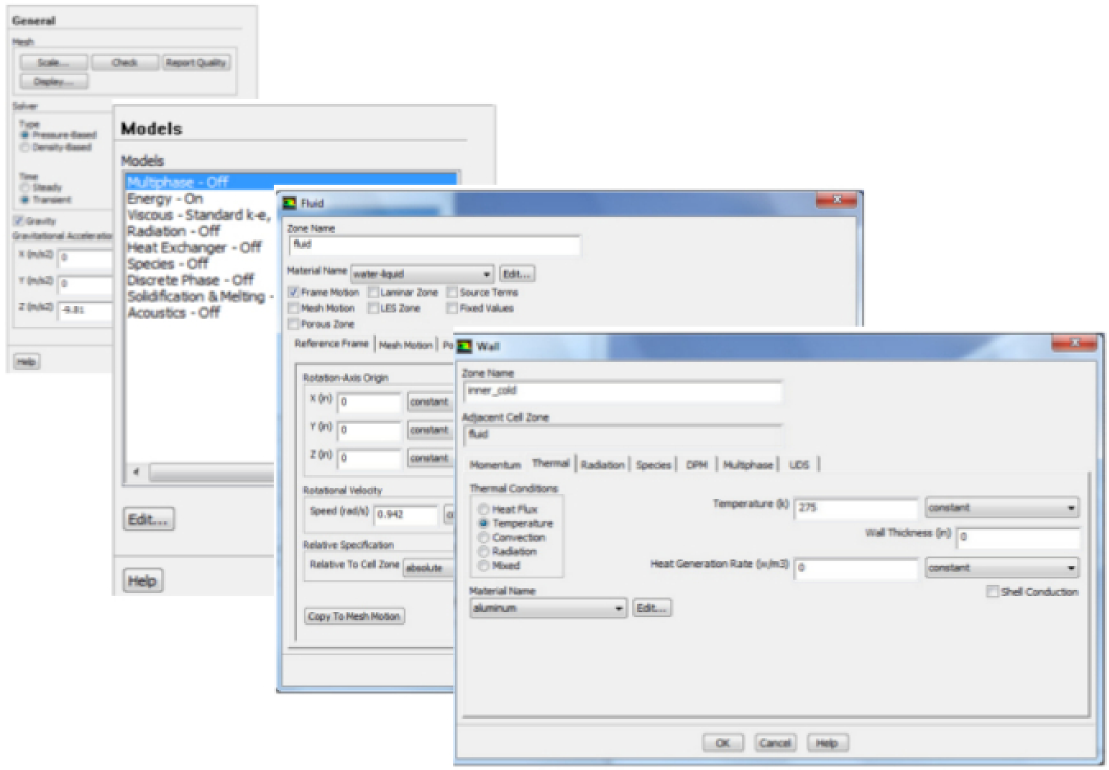
Classic result by Hide & Mason (streak photograph)

Simulate the system by Ansys-Fluent



Geometry & mesh

(N. Kulkarni , 2012, Applied Project)

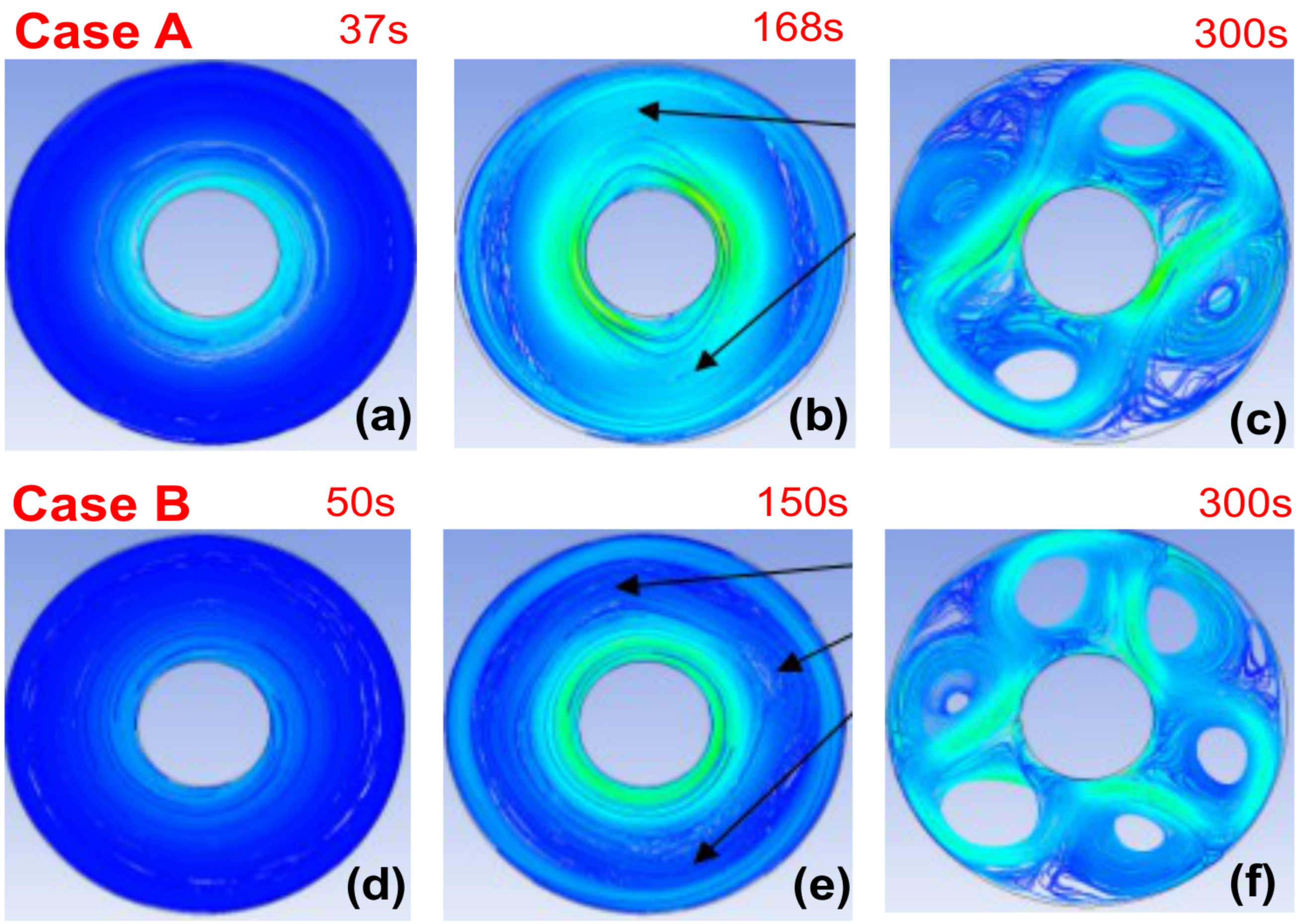


Setup of parameters in Fluent

(N. Kulkarni , 2012, Applied Project)

The simulation

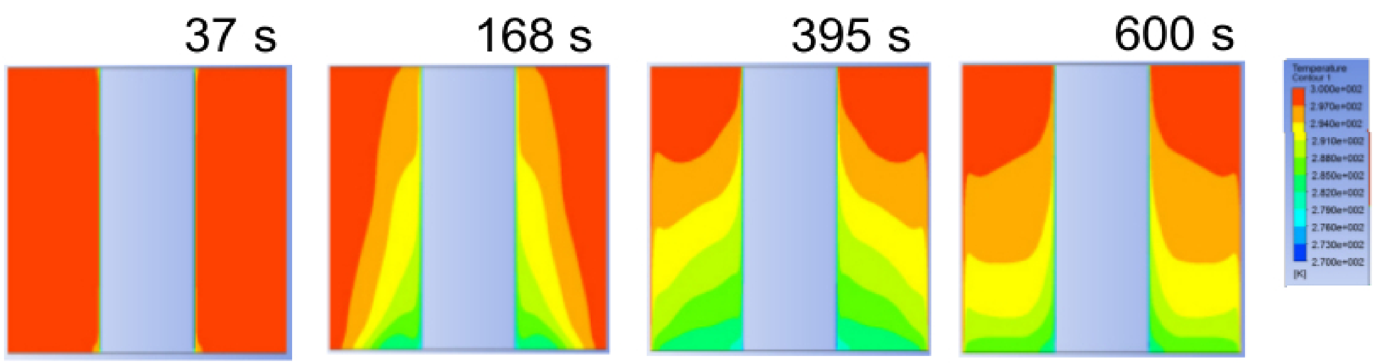
Streamlines (top view)



Case A: rotation rate = 0.942 rad/s, radial ΔT = 30K
Case B: rotation rate = 0.942 rad/s, radial ΔT = 15K

(N. Kulkarni, 2012, Applied Project)

Temperature vertical cross section



Case A: rotation rate = 0.942 rad/s, radial ΔT = 30K

(N. Kulkarni, 2012, Applied Project)

Why numerical simulation ?

- Easy and cheap to modify the apparatus
- Produces the 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 tank) for multiple experiments

and more ...

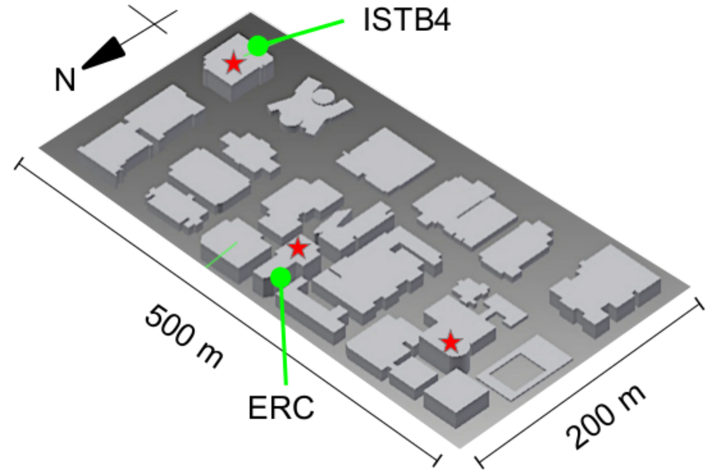
Caution: Computer model ≠ Reality

- Finite resolution
- Incomplete representation of physical processes

etc.

Example 2. Environmental flow over urban landscape
(potential application: assessment of wind power potential at rooftop)

(SE quarter of ASU campus)



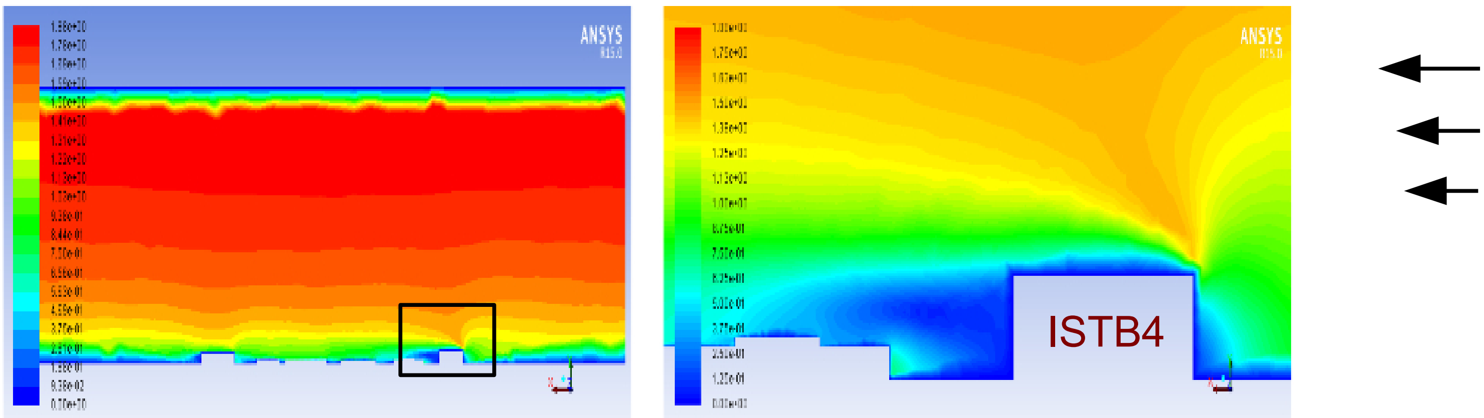
Geometry model to use in Fluent
(Digitized data from ASU library)

(X. Ying, 2015, MS thesis)

The simulation: Blow the wind from the east

(In the simulation, the buildings are scaled down 1:4; wind speed of inflow is increased accordingly by Reynolds number similarity)

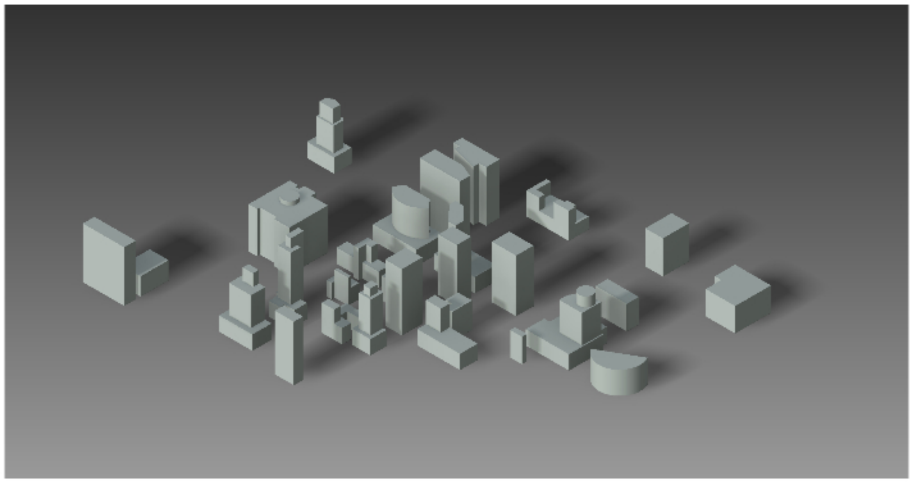
x-z cross section of u-velocity



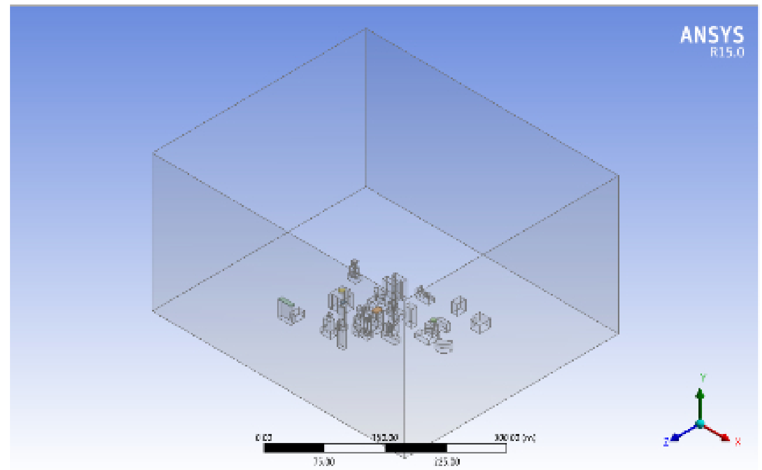
(X. Ying, 2015, MS thesis)

Apply the same framework to downtown Phoenix

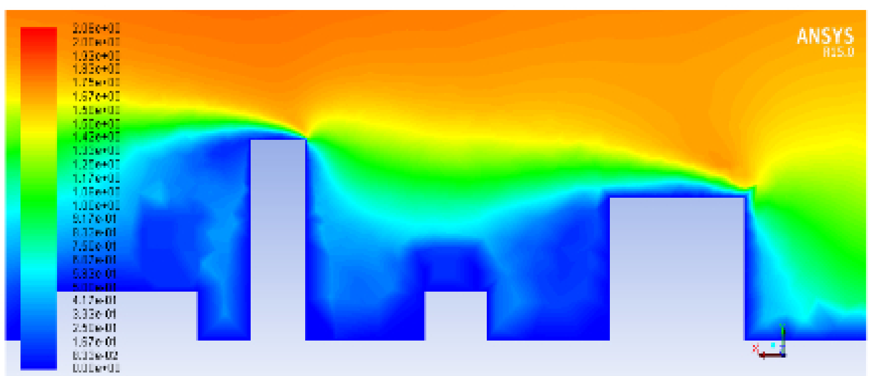
(X. Ying, 2015, MS thesis)



Geometry model (based on Google Earth and retaining only the tallest buildings)



Put in Ansys-Fluent



Simulation: u-velocity along one x-z cross section

Look forward to a fun semester!