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*

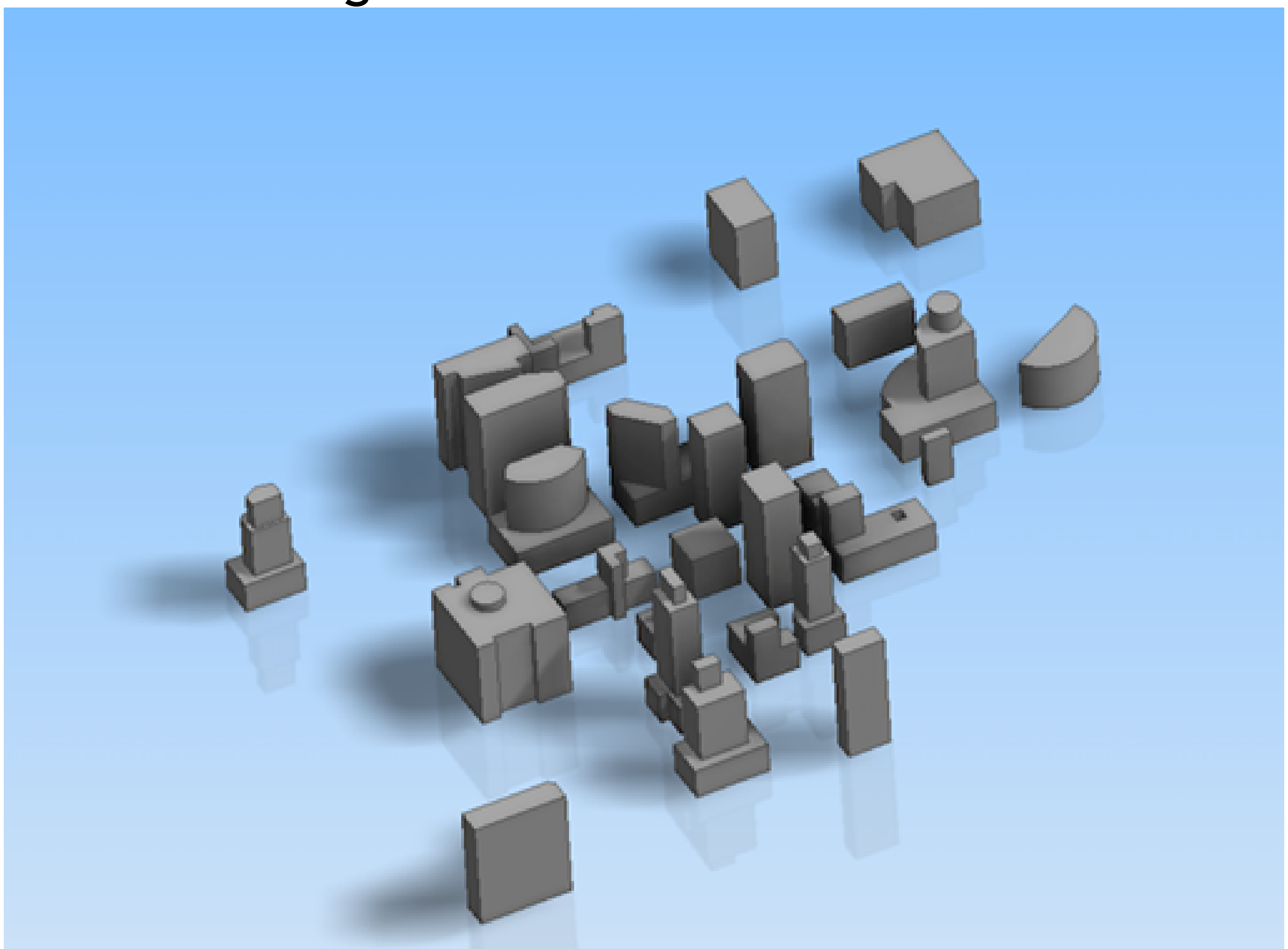
Assessment of rooftop wind power potential

(Urban community, college campus, etc.)

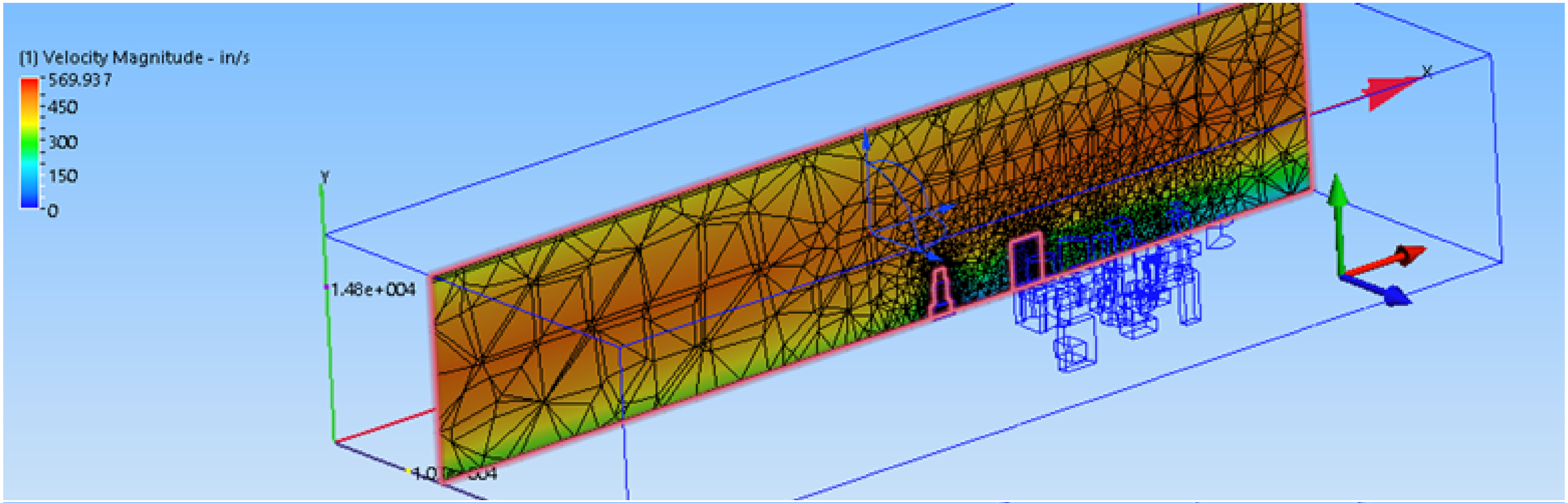
Simulation for Downtown Phoenix

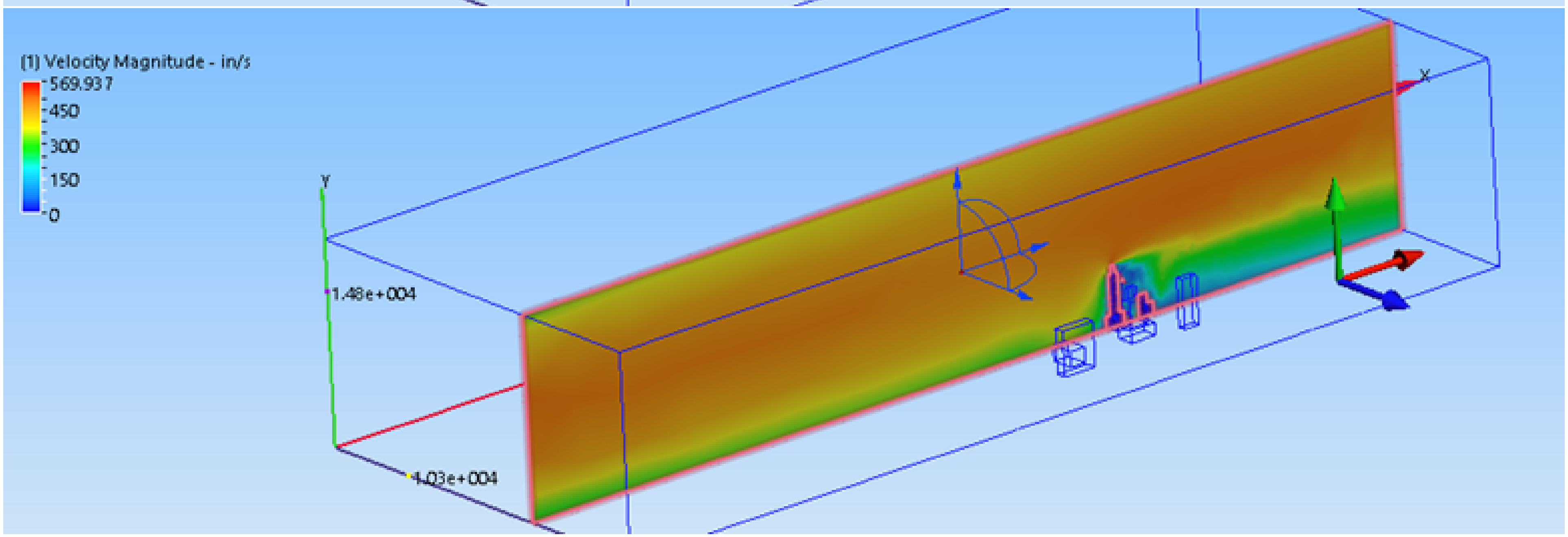
[Research project, 2015, X. Ying (Huang group)]

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

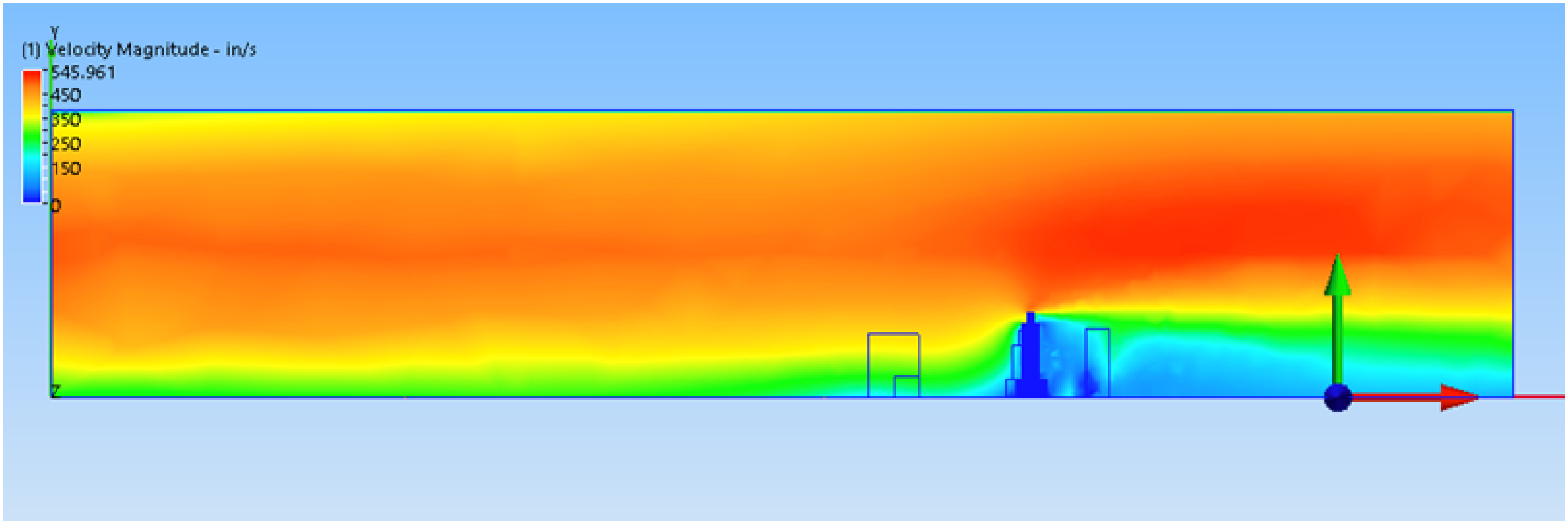


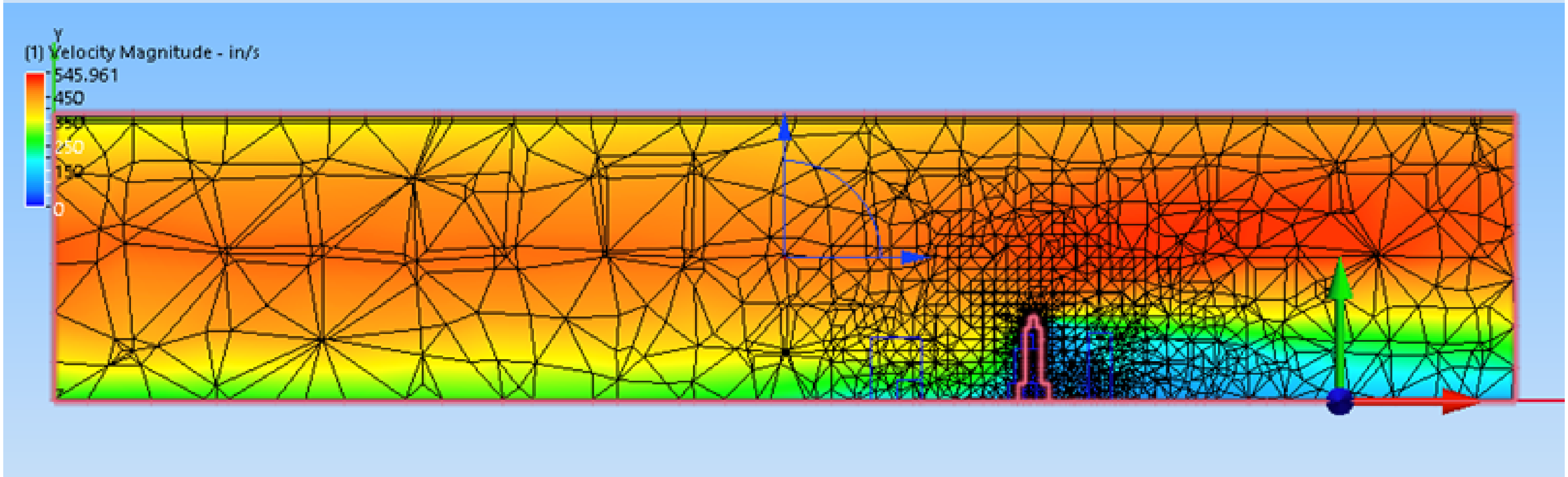
Using *Autodesk Flow Design* - Model setup



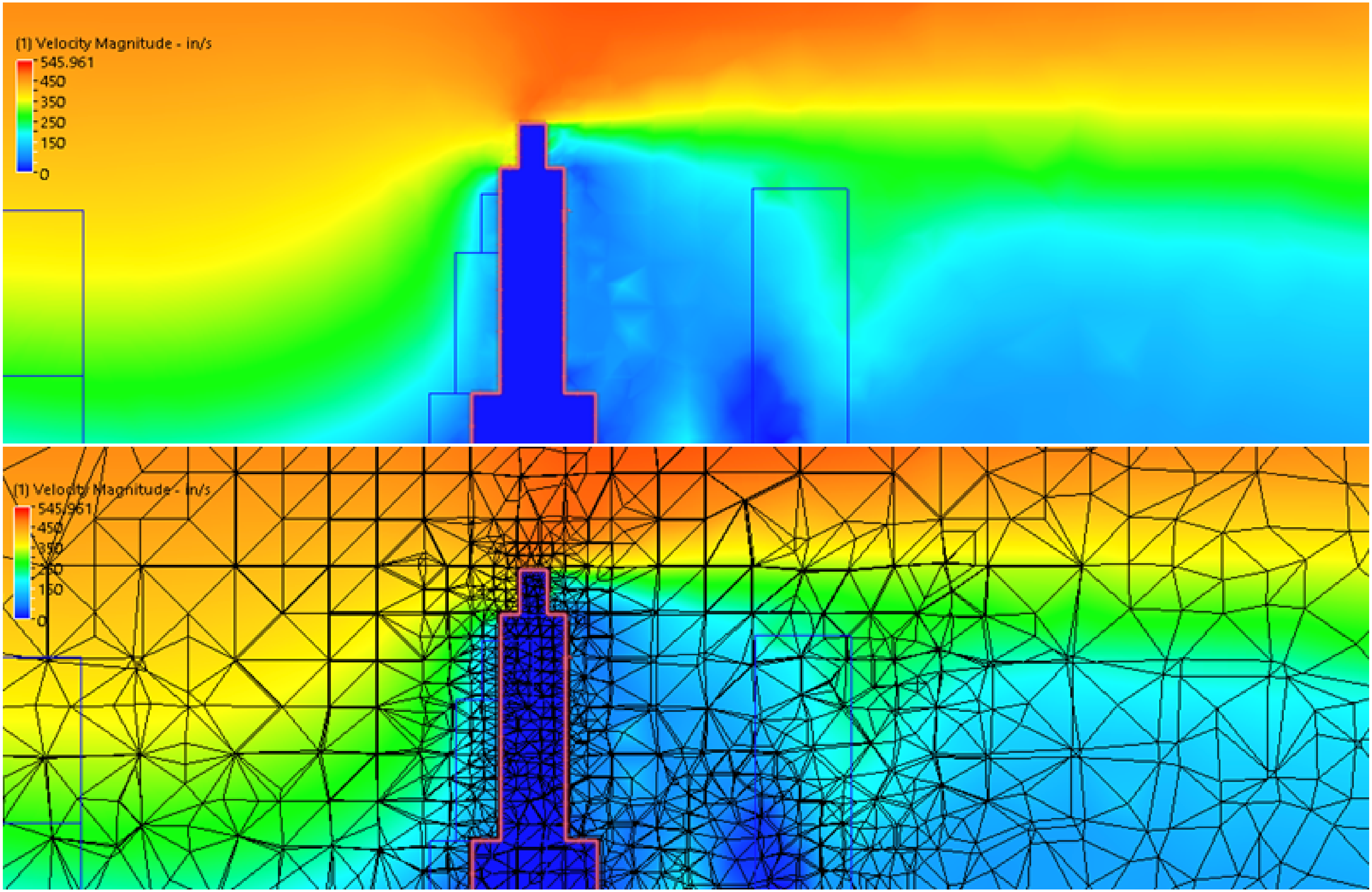


Simulated wind speed (simplified version with fewer buildings)



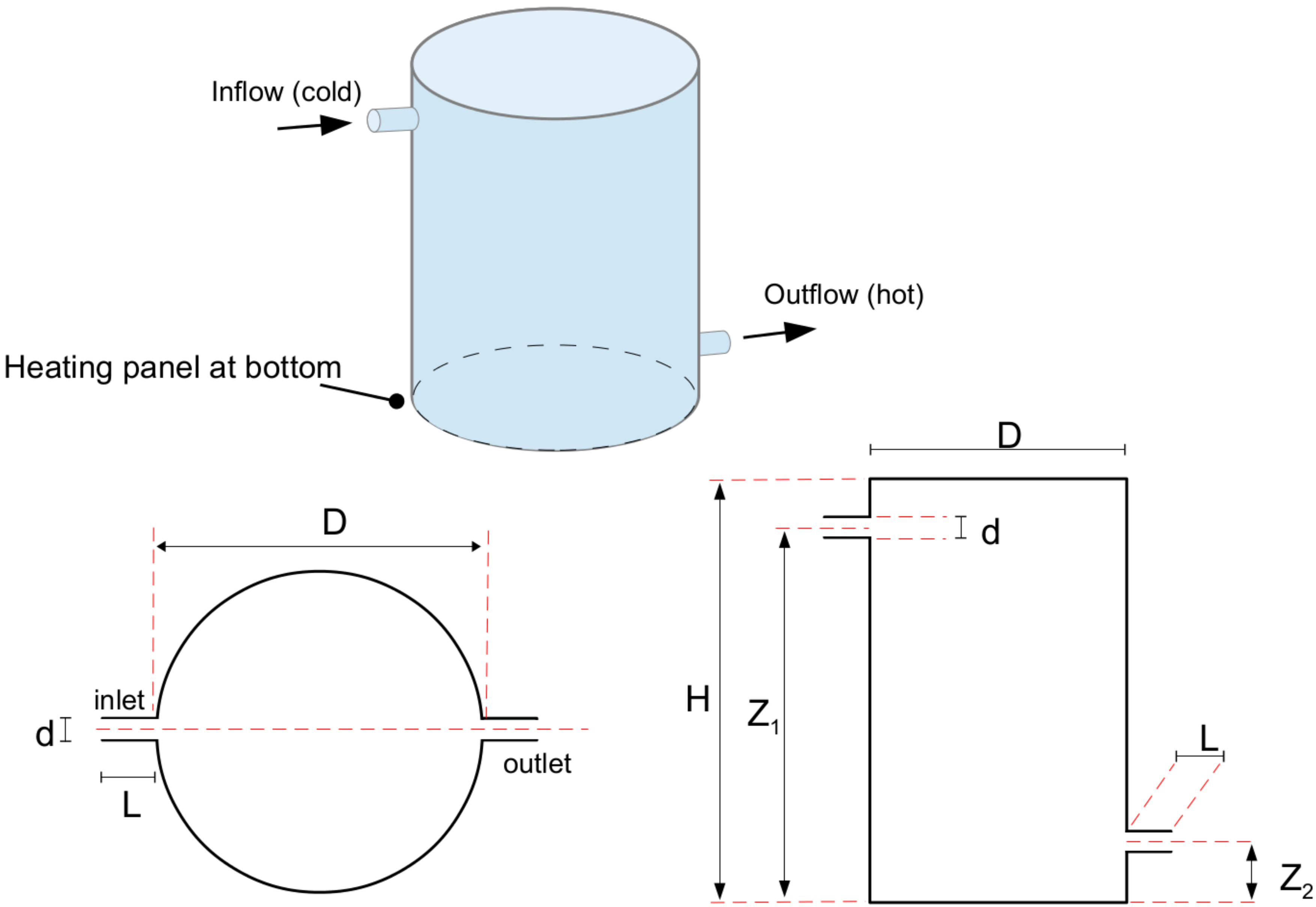


Detail around the building - wind speed

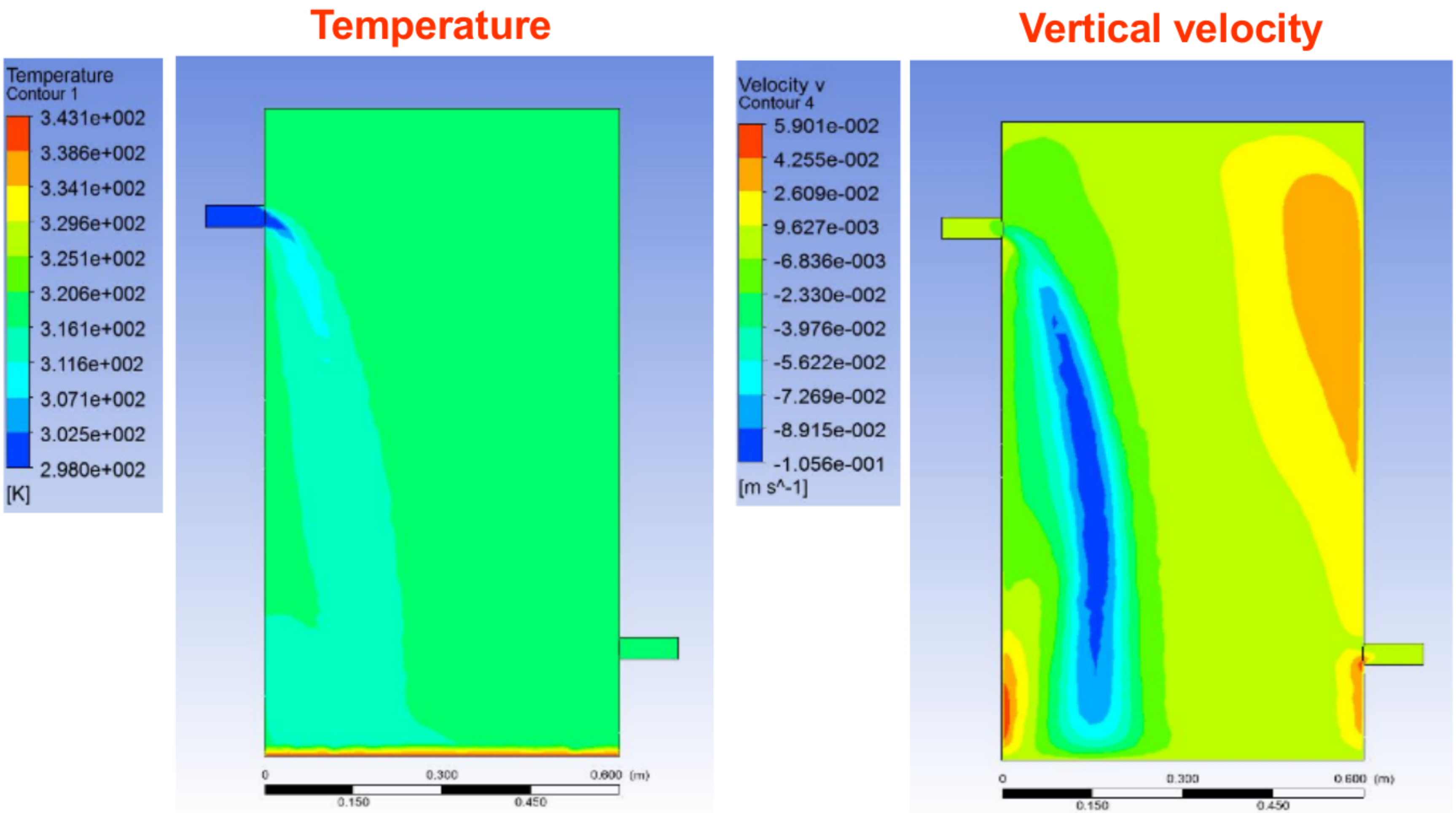


Example 2 Internal flow - using *Ansys-Fluent*

Temperature and flow within a water heater



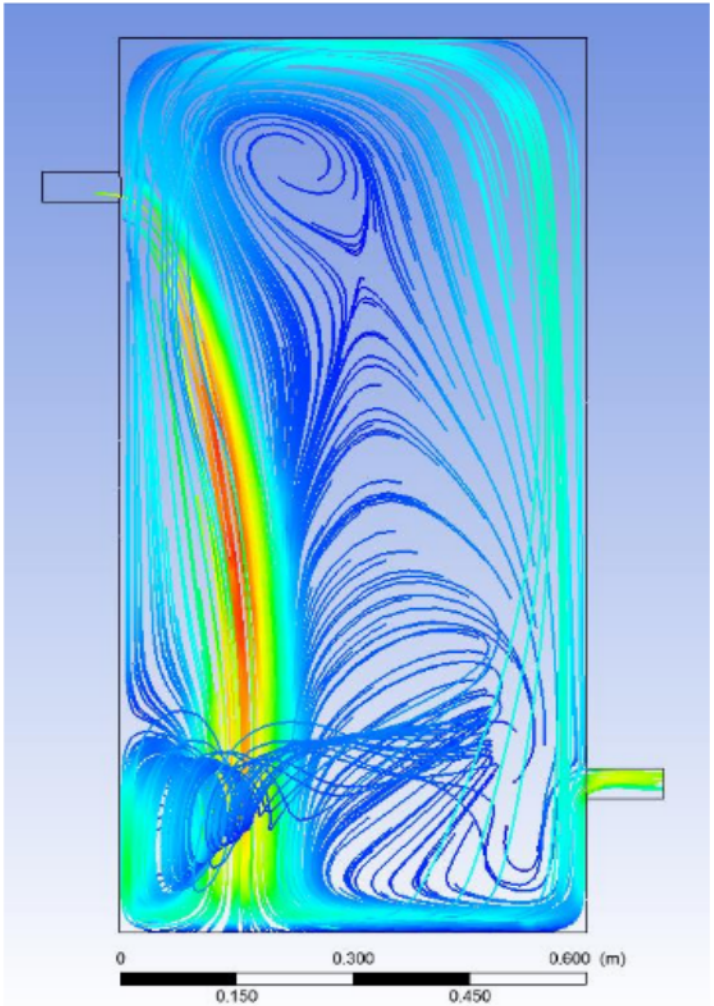
Simulated temperature & velocity



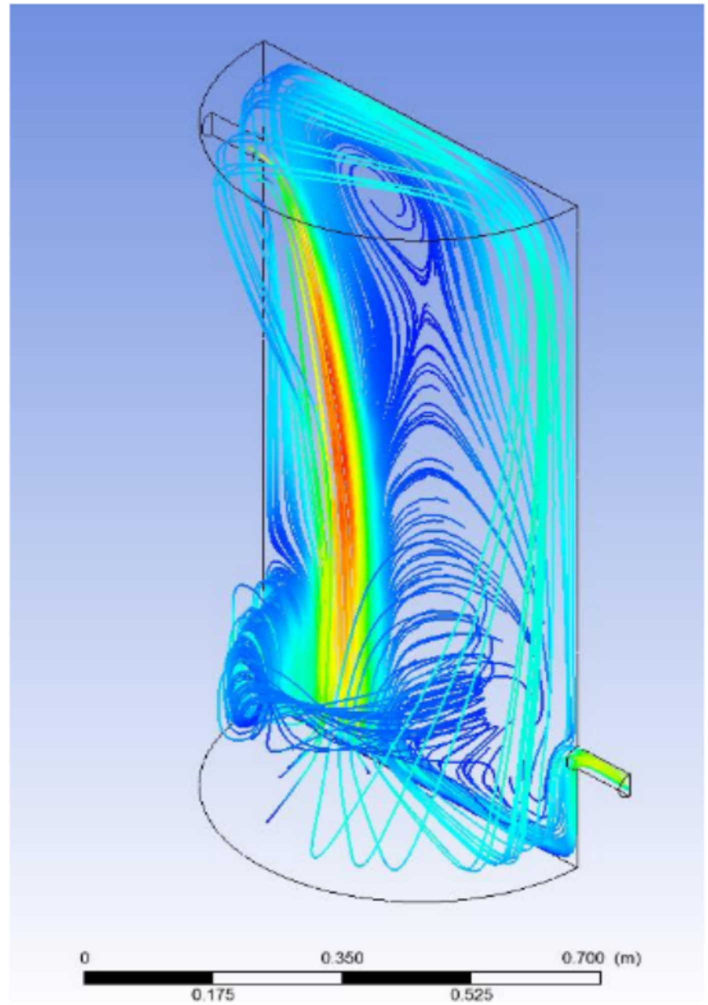
[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

Stream lines

Plane of symmetry



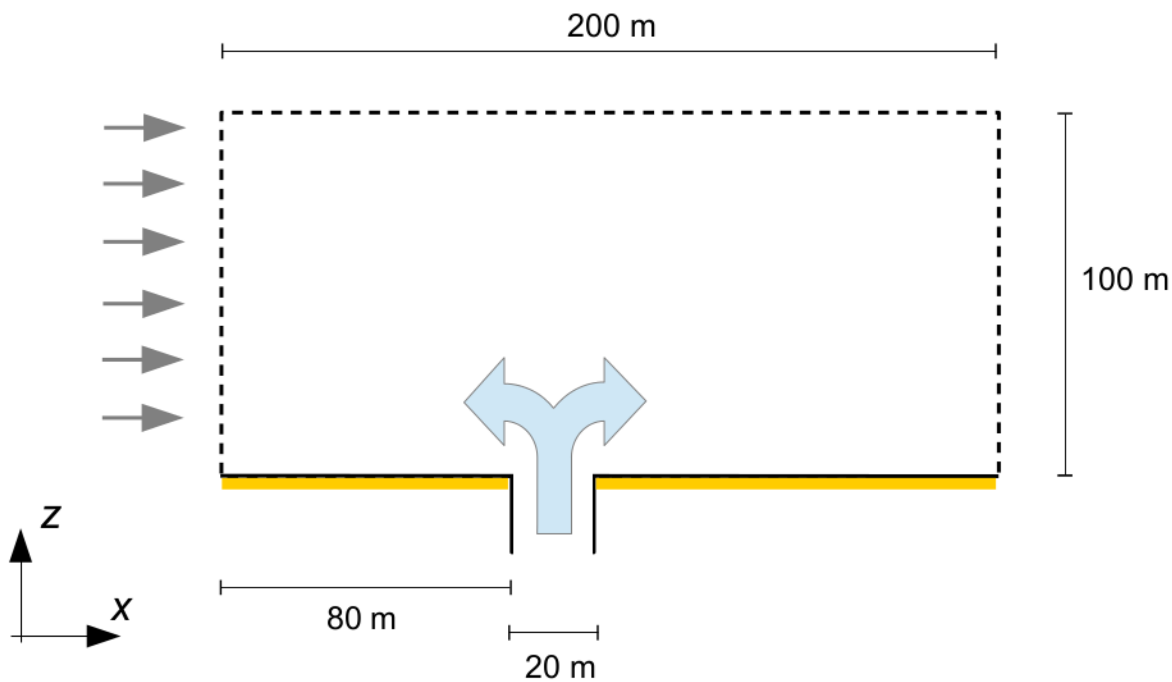
Side view



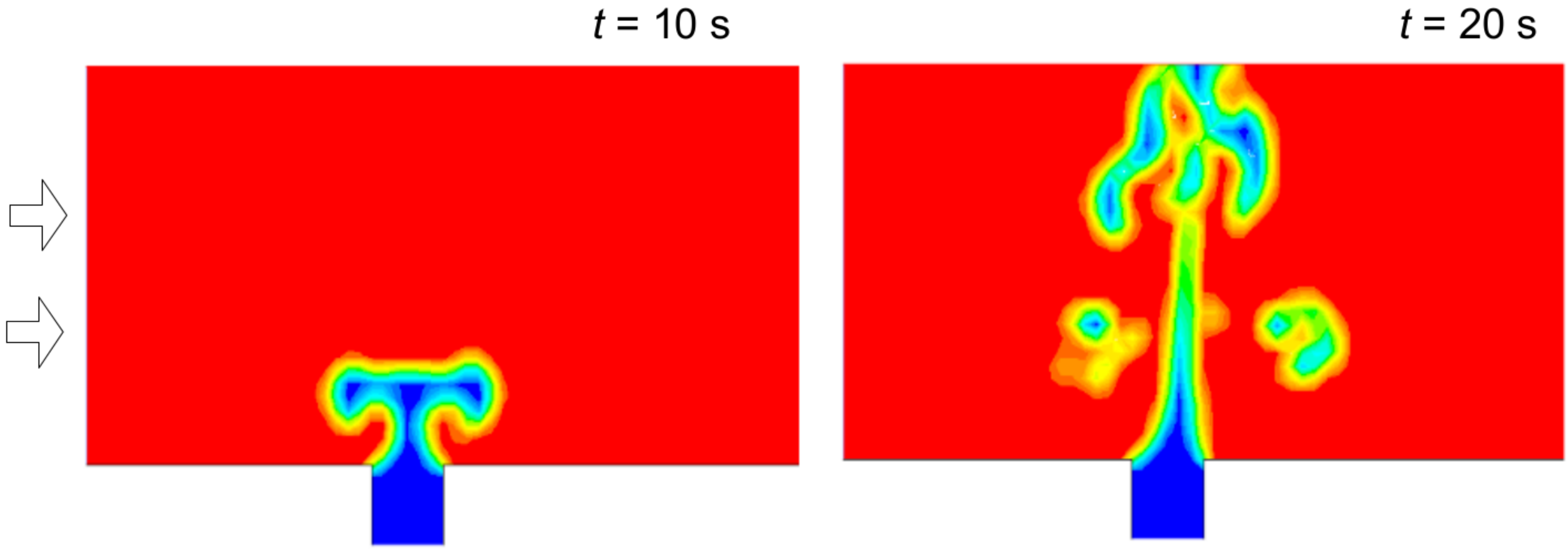
[Applied CFD course project, Fall 2015, A. Aguinaga (Instructor: HP Huang)]

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

Leak of methane from an underground reservior into open air



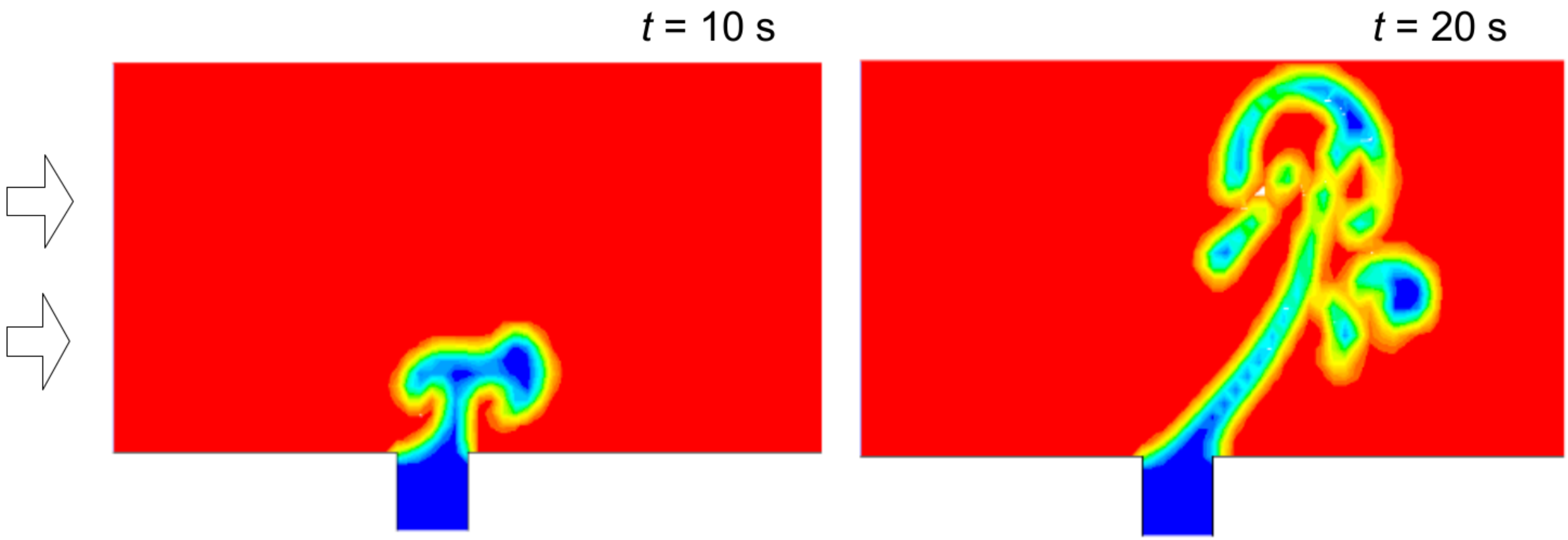
Low wind condition (U = 0.25 m/s)



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

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

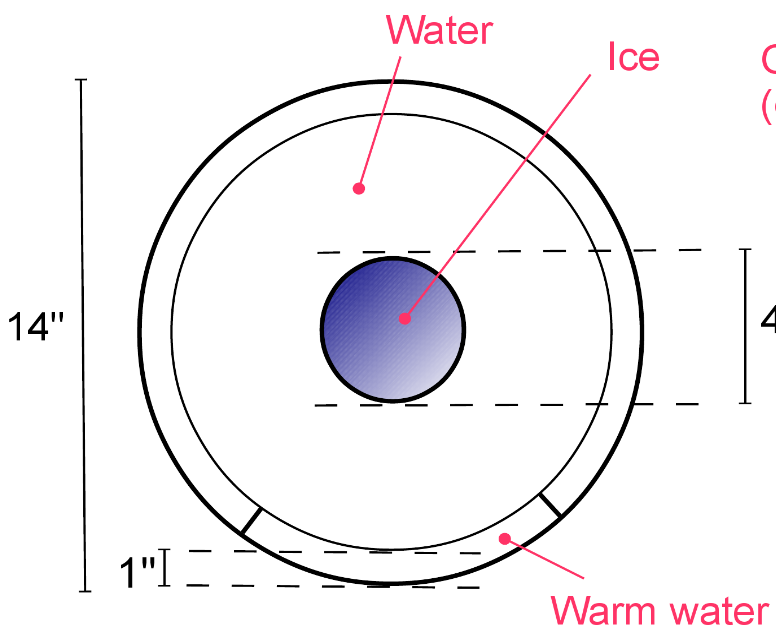
High wind condition (U = 0.5 m/s)

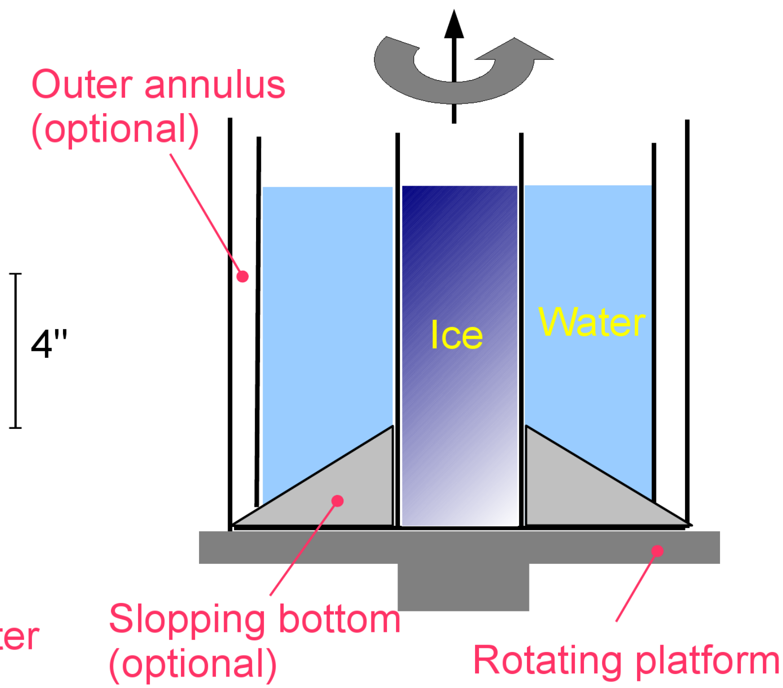


[Applied CFD course project, Fall 2015, Z. Damania (Instructor: HP Huang)]

Example 4 Comparison of numerical simulation with lab experiment

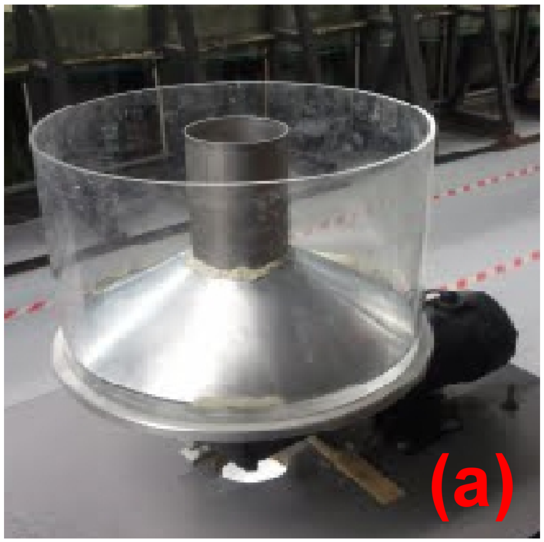
Currents and waves in a rotating water tank
(which emulate large-scale environmental flows)



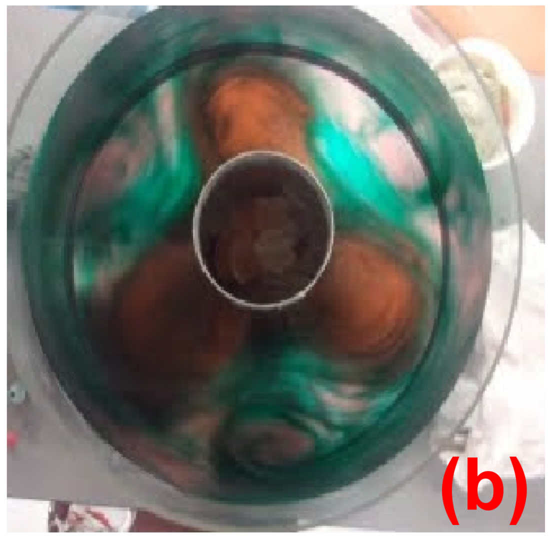


(Huang Lab)

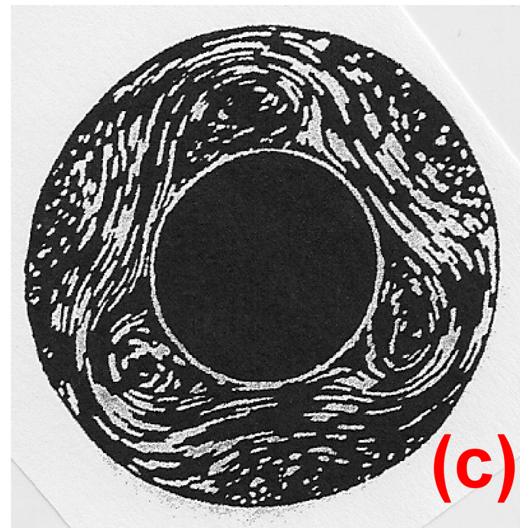
(Huang Lab) (Huang Lab)



The apparatus

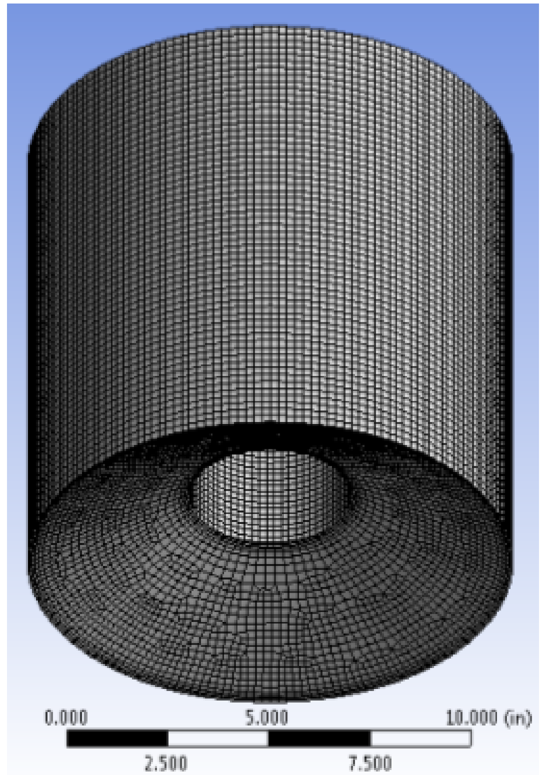


Experiment which shows
a 3-wave structure
(visualization by color dye)



Classic result by
Hide & Mason
(streak photograph)

Simulate the system using *Ansys-Fluent*

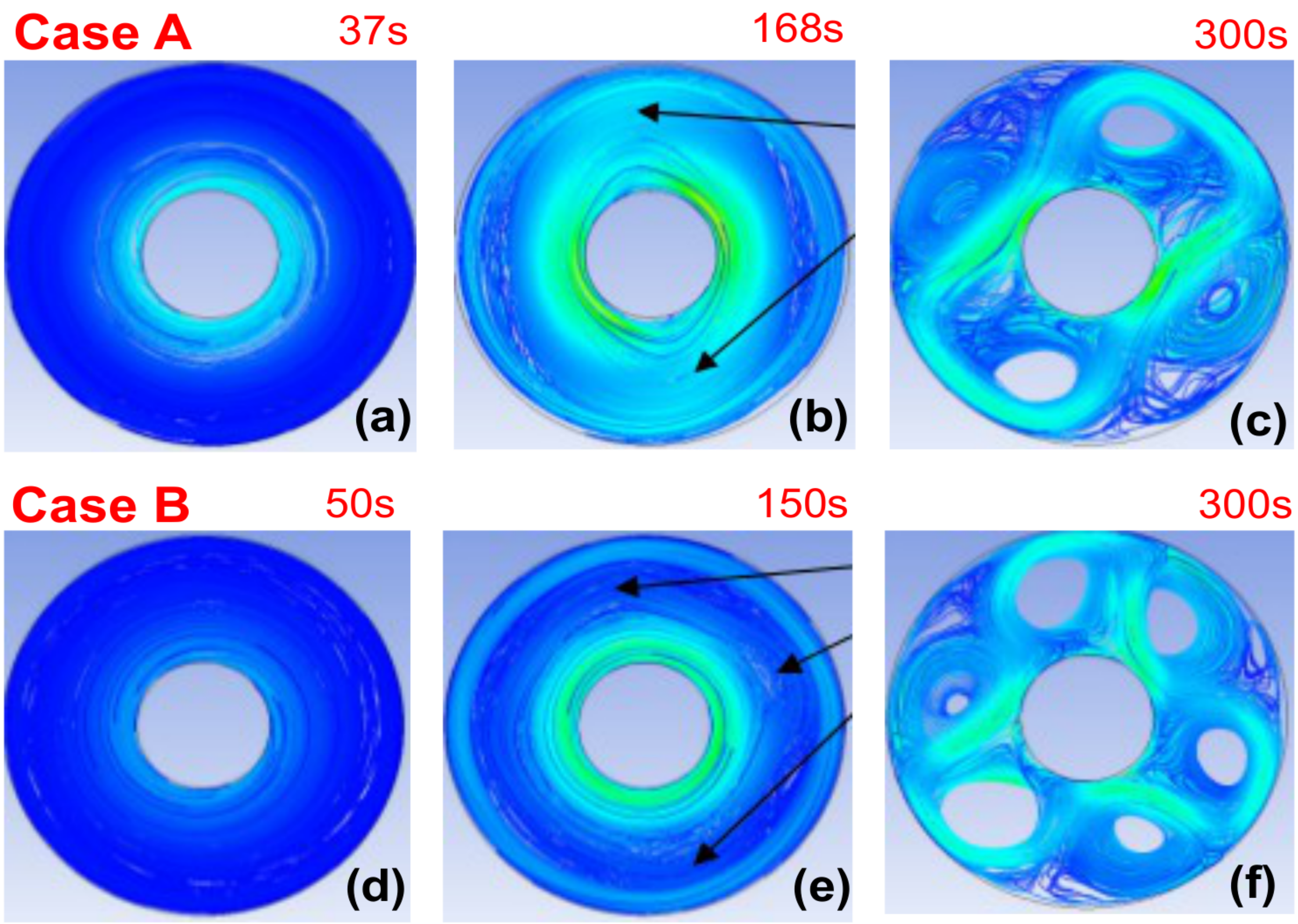


Geometry & mesh

(N. Kulkarni , 2012, Applied Project)

The simulation

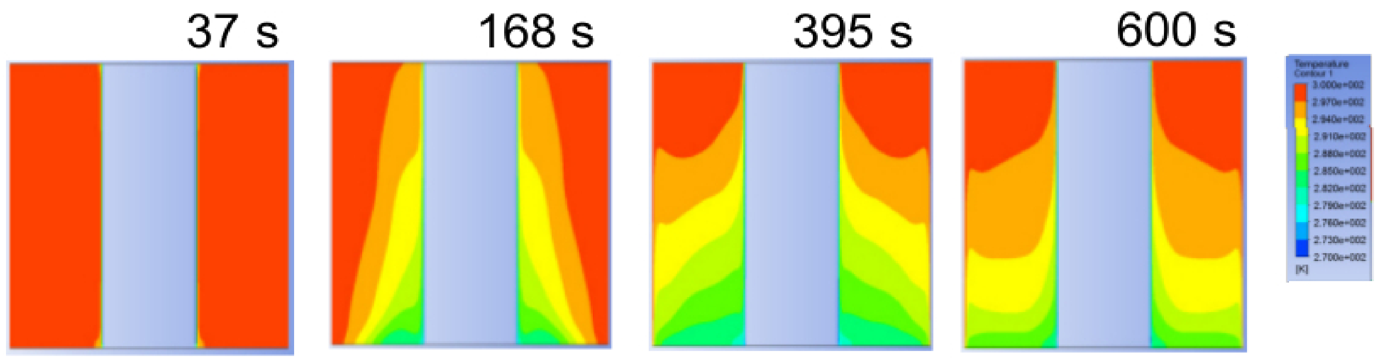
Streamlines (top view)



Case A: rotation rate = 0.942 rad/s, radial 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 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 purchased individually
  Has limits on # of nodes and # of modules to open at once