

# Lecture 1

8/24

\* First email + memo on Ansys sent to class

↳ Install it NOW!

\* Alternative: Access to ECCG 150 Computing Lab has been set up.

↳ NOT GWC 481

- 24/7 access by ISAAC as originally announced
- Use your ASURITE to log in

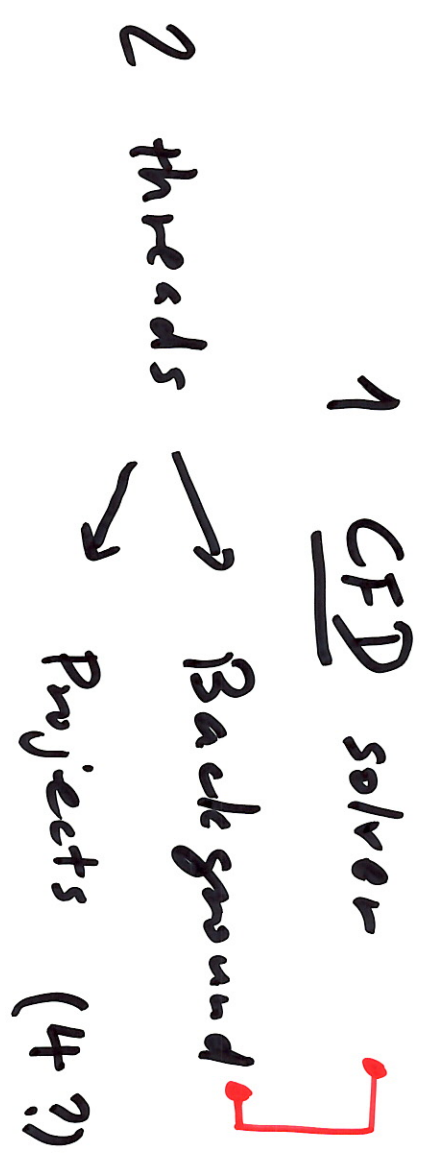
\* 2021 R2 older versions 2020 R1 } Prior releases  
2020 R2 }

\* Compatibility across versions

\* Mac users: Win Emulator

Applied

Computational Fluid Dynamics



Final  
 # Exam 10%  
 90%

(Solidworks etc.)

Design Modeler

(CAD)

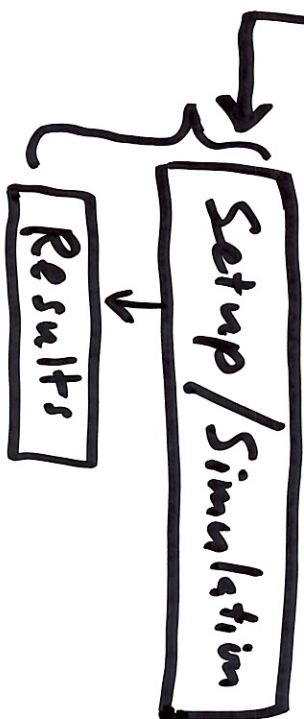
(geometry)

Mesh

Fluent  
(Fluid Sim.)  
(thermo fluid)

Ansys  
many modules

To launch Ansys :



Workbench

CFD-Post

(Matlab  
Excel  
etc.)

# CFD solver

Governing eq. Fluid/Thermal (P.D.E.s)

N-S eq. Fluid

$$\frac{\partial u}{\partial t} = -u \frac{\partial u}{\partial x} + \dots$$

$$\frac{\partial v}{\partial t} = -u \frac{\partial v}{\partial x} + \dots$$

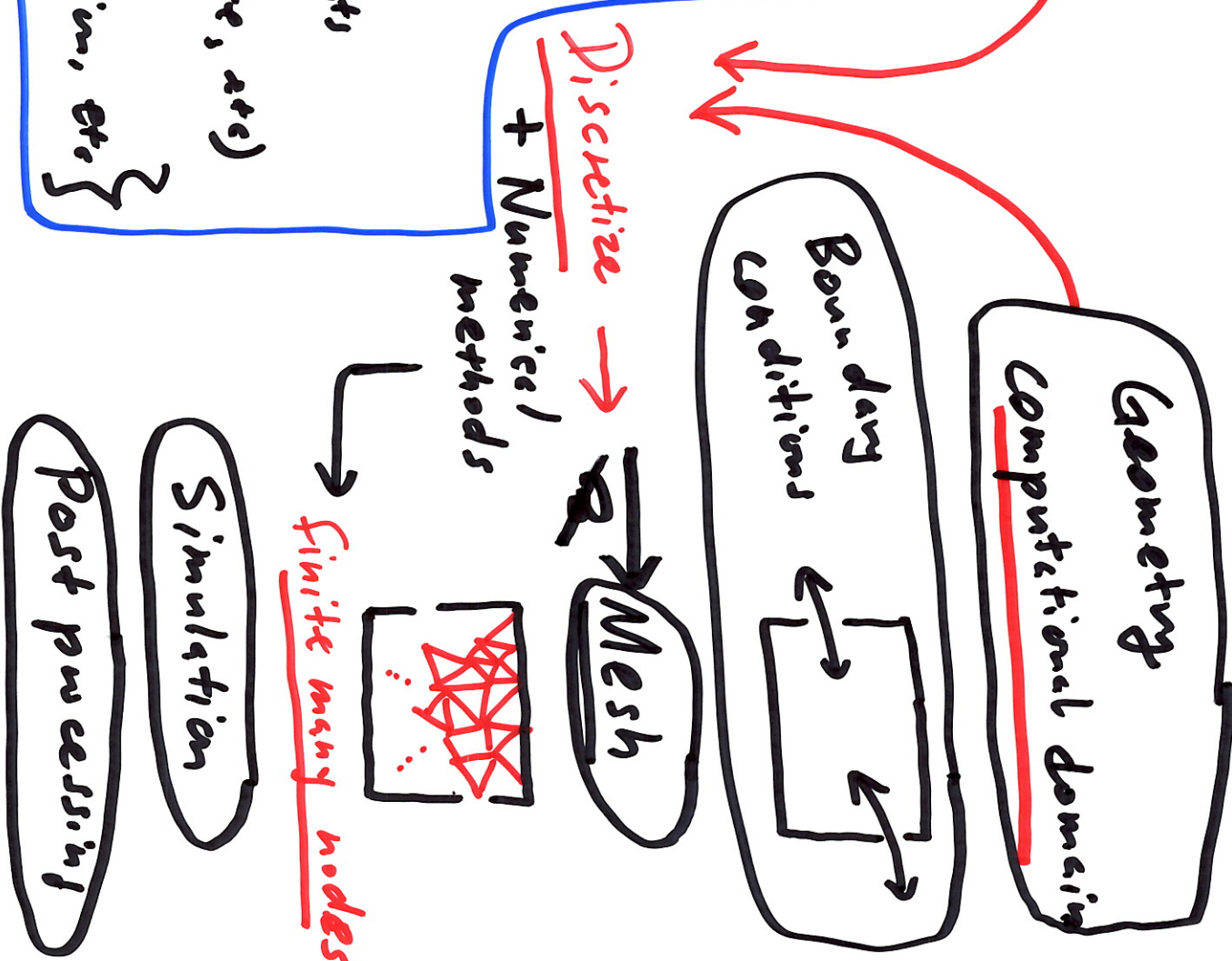
$$\frac{\partial p}{\partial t} = \dots$$

$$\frac{\partial T}{\partial t} = \dots$$

$$\frac{\partial q}{\partial t} = \dots$$

} Momentum  
 } mass  
 } (Thermal)  
 } Energy  
 } Transport  
 } of constituents  
 (Chemical, particulate, etc)

Eq. of state  
 Eg. for chemical reaction, etc.





Numerical methods:

Base bone  
Fluid

Ansys-Fluent: Finite Volume

Spatial discretization

Others: Finite Difference

Spectral

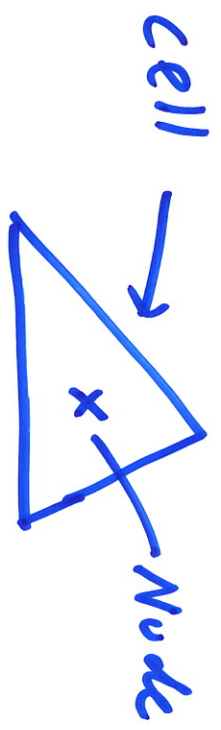
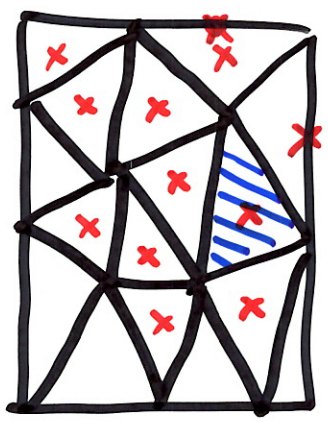
many others

Ansys student

Limit

~ 500K

nodes



Discretization in time:

"Transient"

various schemes to choose



{  
 Upwind  
 1st, 2nd order  
 etc. etc.

---

Codes? (C C++ ?)

Ansys

Not visible to users

Black box

Verification  
Validation }

this  
beyond class

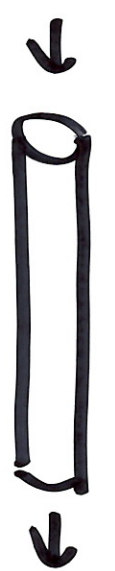
(will be  
some  
compromise)

Grid  
convergence

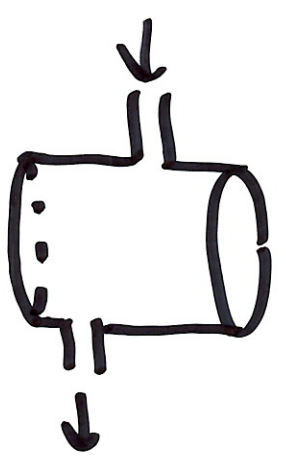
# Homework & Projects

~~For~~ Tutorials → Hw1

## Projects (4 ?)



\* Internal flow w/ heat transfer



\* Flow w/ interface (two-phase)



\* External flow



(\* Compressible flow ?)

\* Engineering design