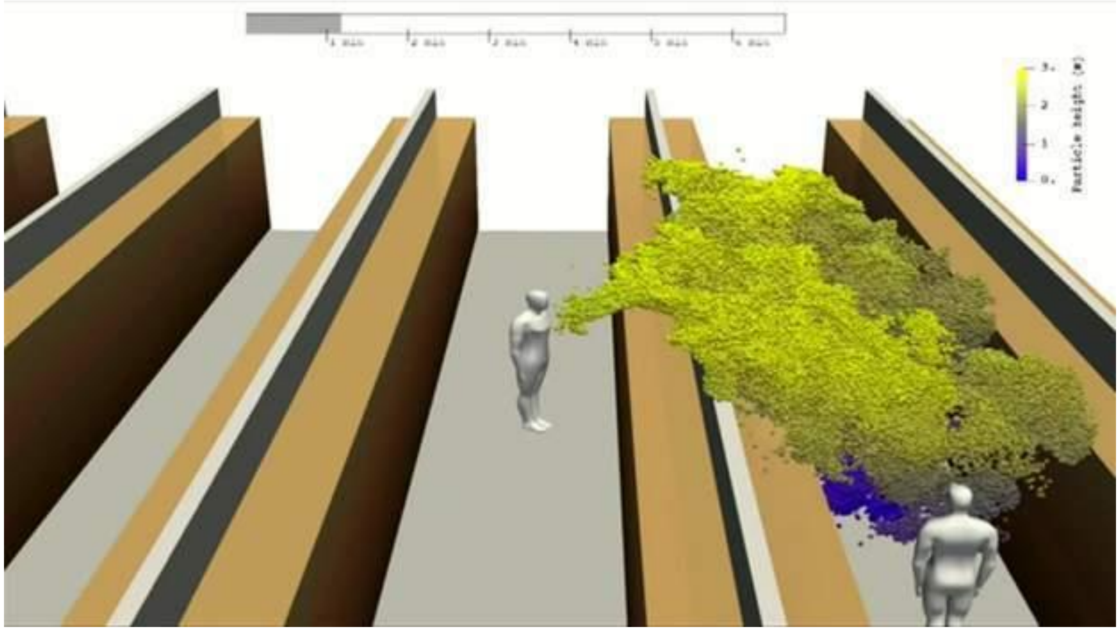
4A2 Computational Fluid Dynamics (CFD)

Jie Li CUED 2021

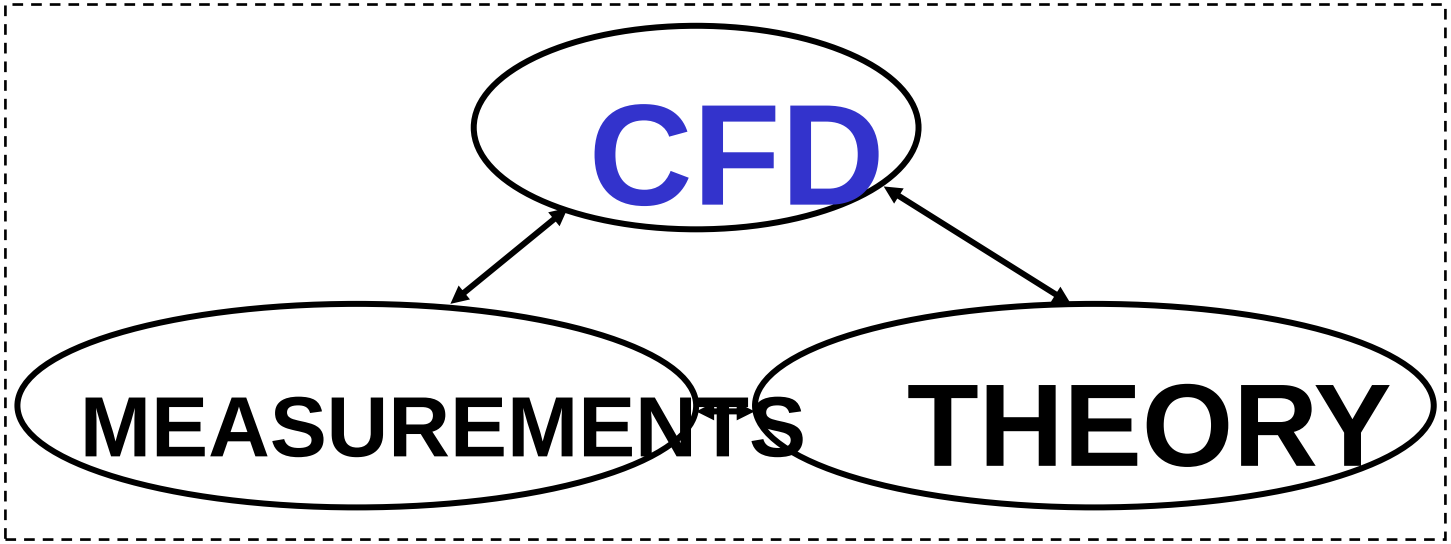
- Computation + Fluid Dynamics
- Numerical Methods (lectures)
- Euler Solver for Compressible Flows (coursework)



Simulation: how people coughing can spread coronavirus in confined spaces

Courtesy of Aalto University, Finland

TRIAD (Available Tools)



- Experimental fluid dynamics (17th century)
- Theoretical fluid dynamics (18th and 19th centuries)
- CFD (advent of computer, numerical algorithms – 1960s)

CFD Advantages

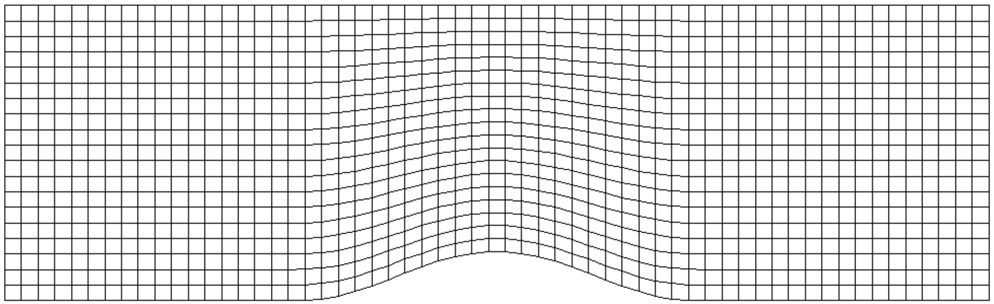
- provide a significant amount of detail about a flow situation
- provide an effective means for the rapid evaluation of what-if design scenarios
- Geometry changes easy
- Safe for dangerous experiments

CFD Problems

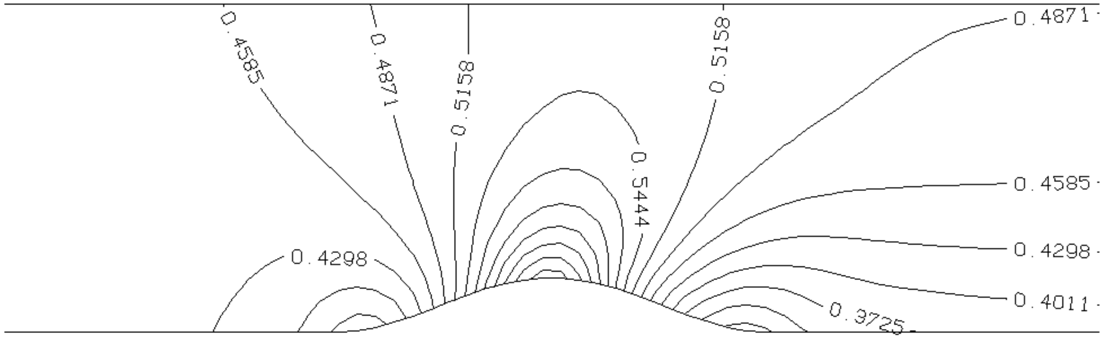
- Turbulence modelling a big problem
- Geometry handling and meshing can be time consuming
- Needs careful validation

Assessment is 100% by Coursework

Write a program for solving 2D Euler Eqns



Mesh



Mach Contours

Course Information

Documents (on Dept. Linux Teaching System)

- **ls /public/teach/4A2/**
- **cp -r /public/teach/4A2/Reading .**
- **cp -r /public/teach/4A2/2021_4A2 .**

Online Demonstration Sessions (Oct 14 - Dec 1)

- Mondays (2pm - 4pm)
- Tuesdays (2pm - 4pm)
- Wednesdays (2pm - 4pm)
- Thursdays (2pm - 4pm)

Submission of 2 Reports

- Interim report: before 4pm **Thursday 11th Nov**
- Final report: before 4pm **Friday 10th Dec**

Access to Linux Teaching System

From Linux Platform

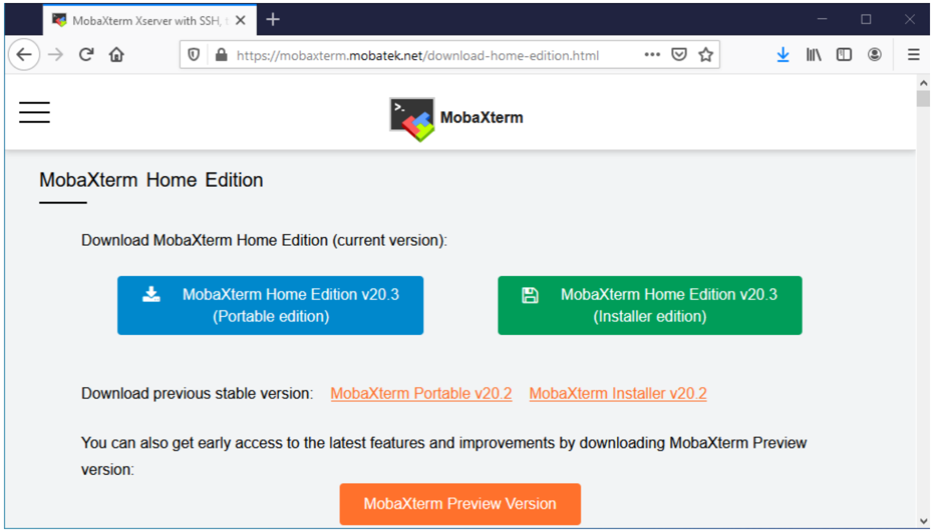
- **ssh –X CRSid@gate.eng.cam.ac.uk**
- **ssh –X CRSid@ts-access**

DATA Transfer

- **scp CRSid@gate.eng.cam.ac.uk:2021_4A2/SaveSrc.tar.gz .**
- **scp SaveSrc.tar.gz CRSid@gate.eng.cam.ac.uk:2021_4A2/.**

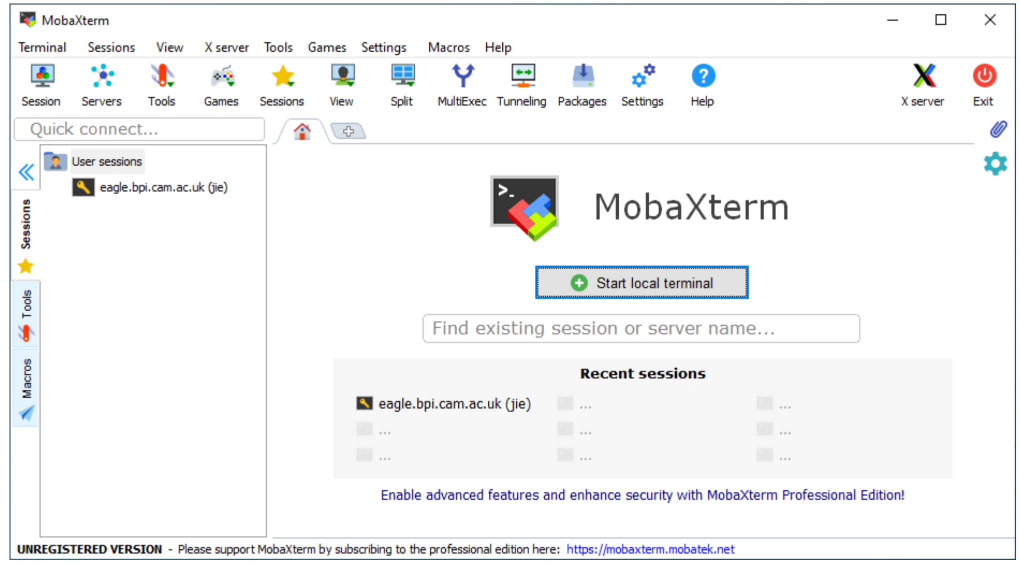
For Windows Platform, first install MobaXterm

https://mobaxterm.mobatek.net/download-home-edition.html

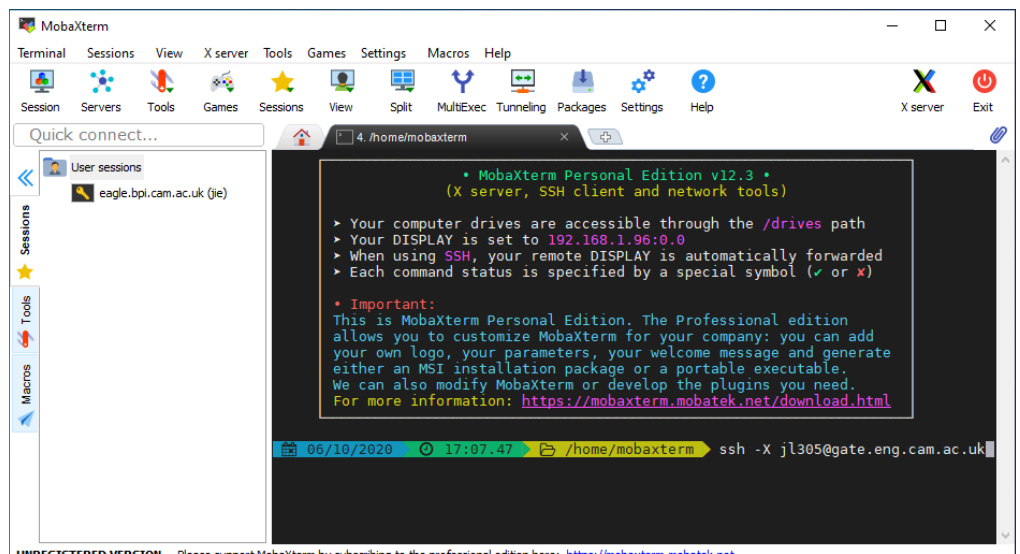


Use of MobaXterm

Launch MobaXterm



Start local terminal



Content of Lectures

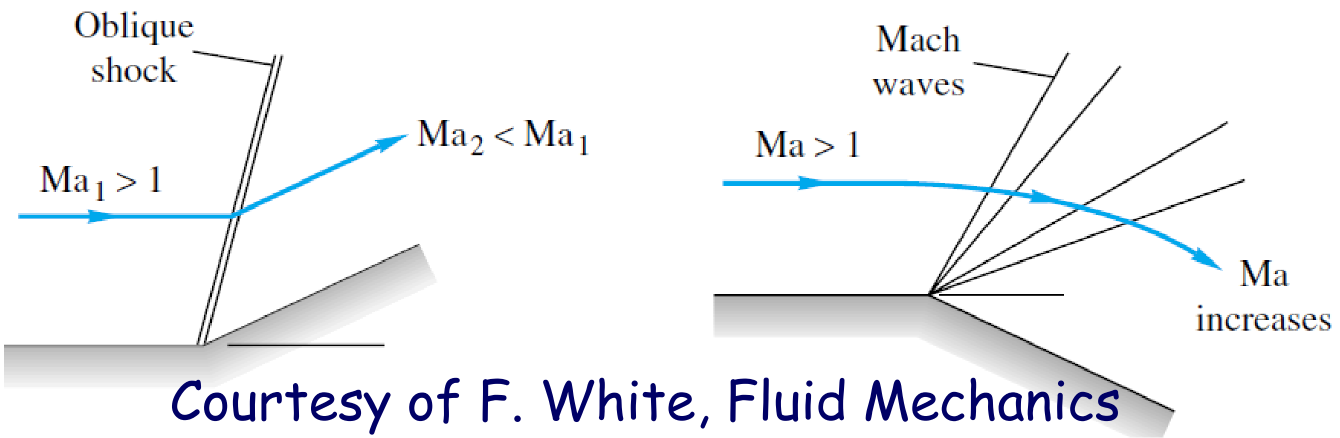
- Introduction to CFD
- Writing a Basic Euler Solver
- Advanced concepts and Test Cases
- Numerical Basics
- *Total Variation Diminishing (TVD) Methods*
- *High Resolution Methods*

Aspects of CFD

Journal of Computational Physics (*Journal*)

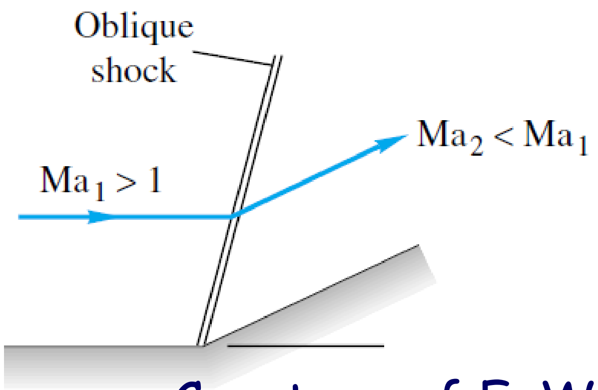
- Compressible Flows (shock capturing)
- Incompressible Flows (pressure based methods)
- Mesh Generators
- Structured/Unstructured mesh methods
- Numerical Methods (*FDM*, *FVM*, *FEM*…)
- Turbulence Modelling
- Eulerian/Lagrangian (moving/fixed methods)
- Adaptive Mesh Methods
- Parallel Computing

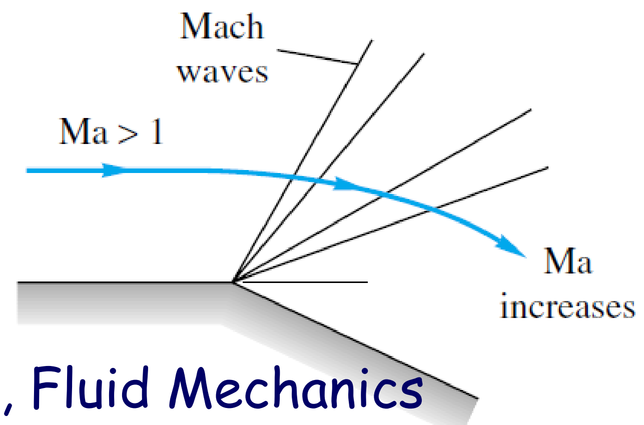
Oblique Shock & Prandtl-Meyer Expansion Wave (Compressible Flows)



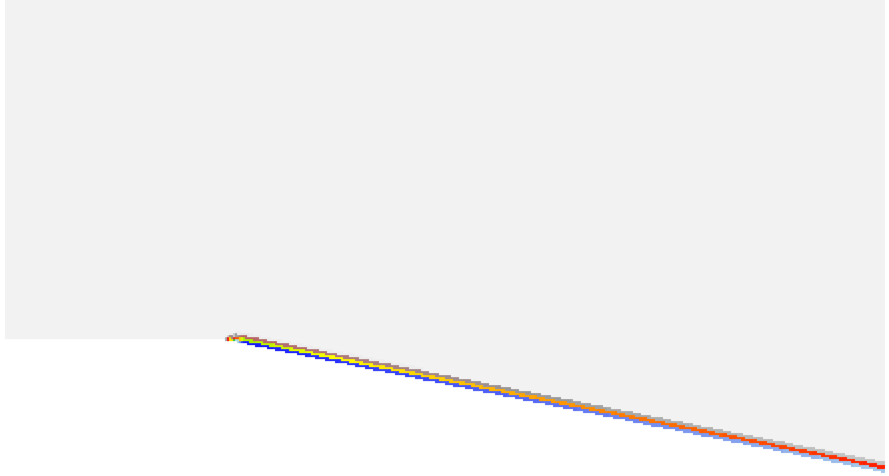
Courtesy of F. White, Fluid Mechanics

Oblique Shock & Prandtl-Meyer Expansion Wave (Compressible Flows)





Courtesy of F. White, Fluid Mechanics



Simulation of expansion wave

Prandtl-Meyer Expansion Wave

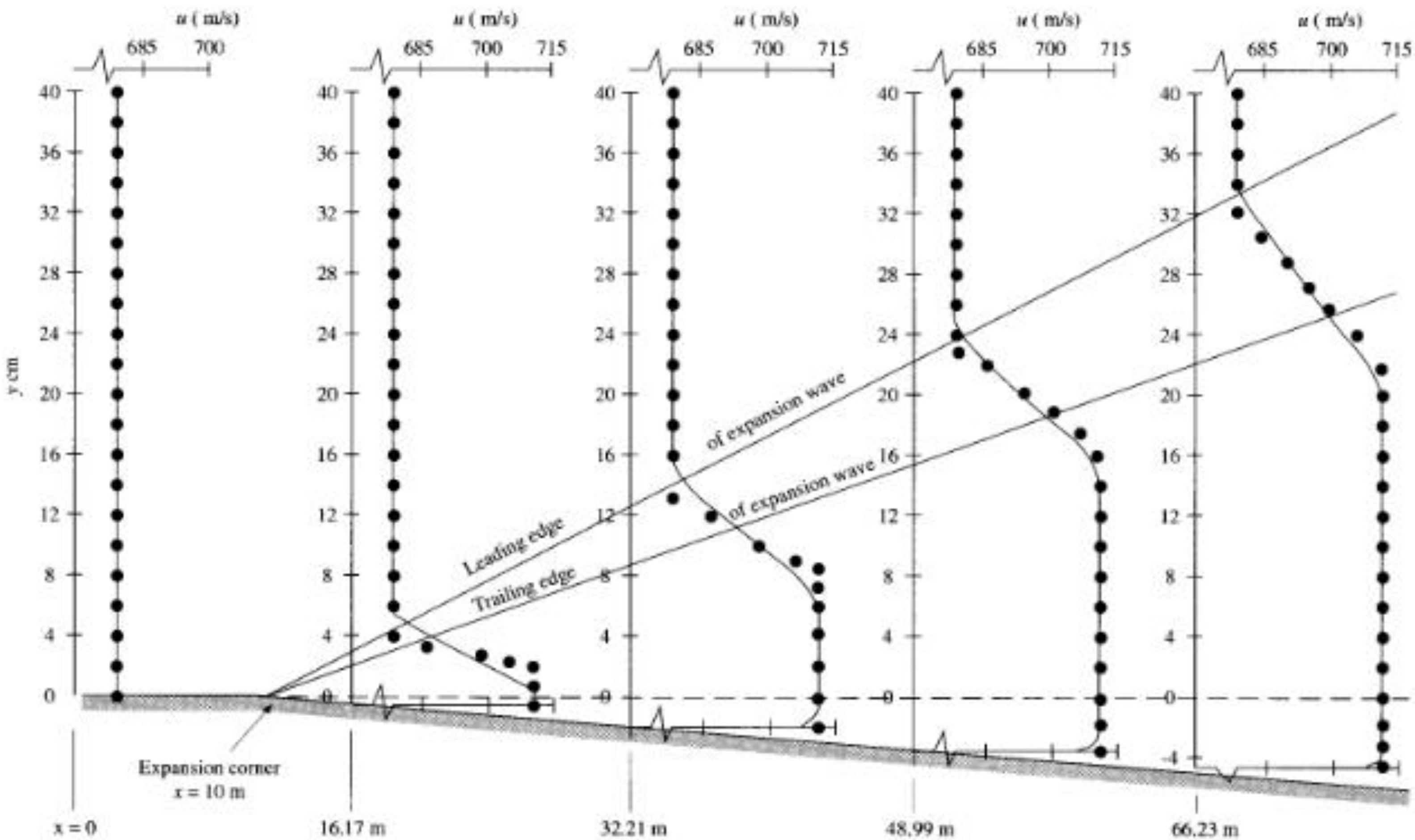


FIG. 8.8
Comparison between the CFD results and the exact, analytical solution (dark circles) for supersonic flow through a centered expansion wave. Solid lines are numerical results.

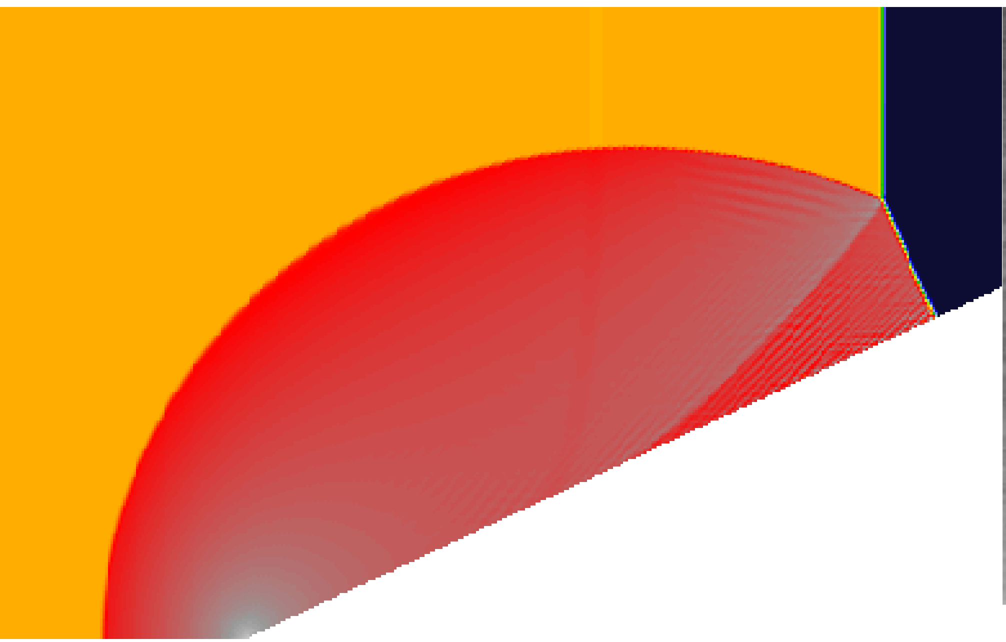
(Courtesy of Anderson, Computational Fluid Dynamics)

Shock Wave Reflection from a Wedge
(Compressible Flows)

Ma = 1.7, wedge 25 degree

Mach Reflection (self-similar)

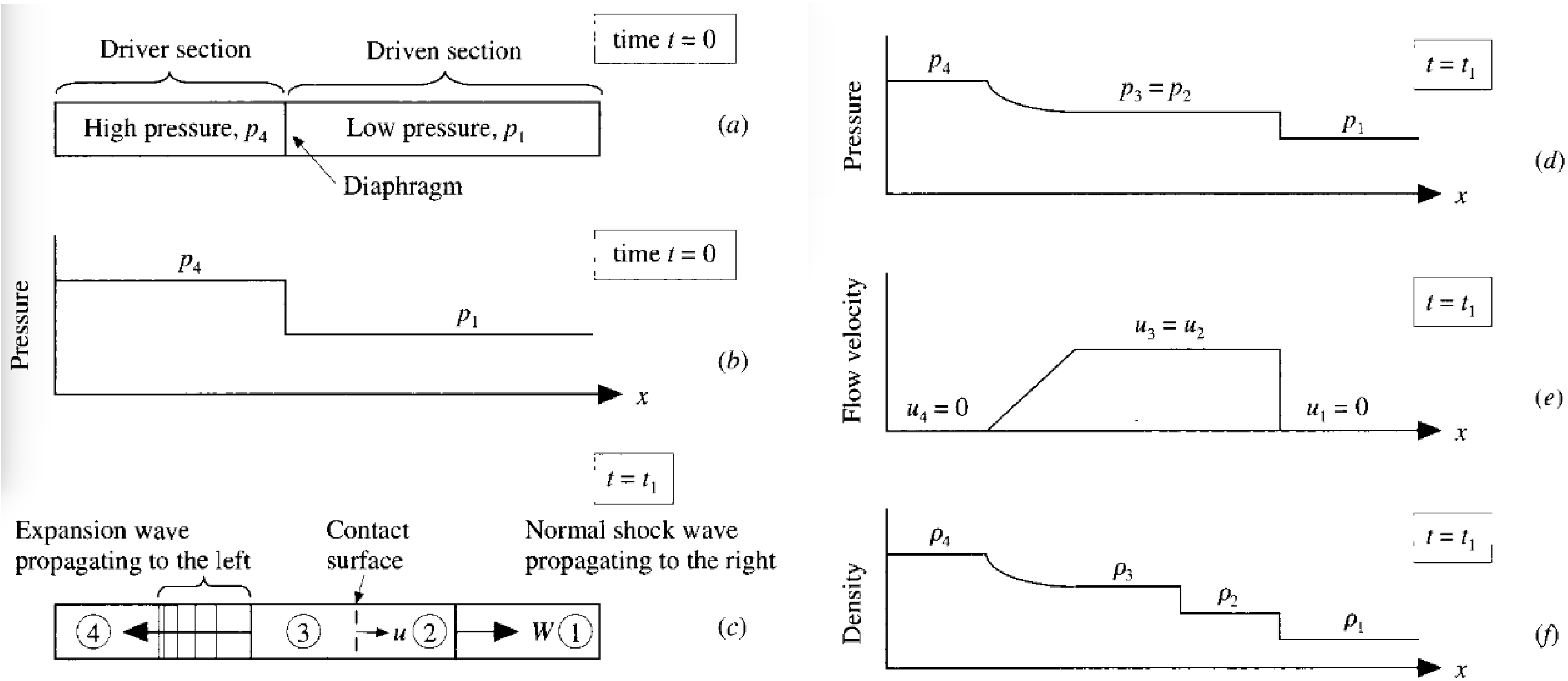
high order method simulation



experiment (Prof. Takayama)

Shock Tube Problem (3A3?)

Anderson, Modern Compressible Flow



(Courtesy of Anderson, Computational Fluid Dynamics)

Shock Tube Problem

Similarity solution

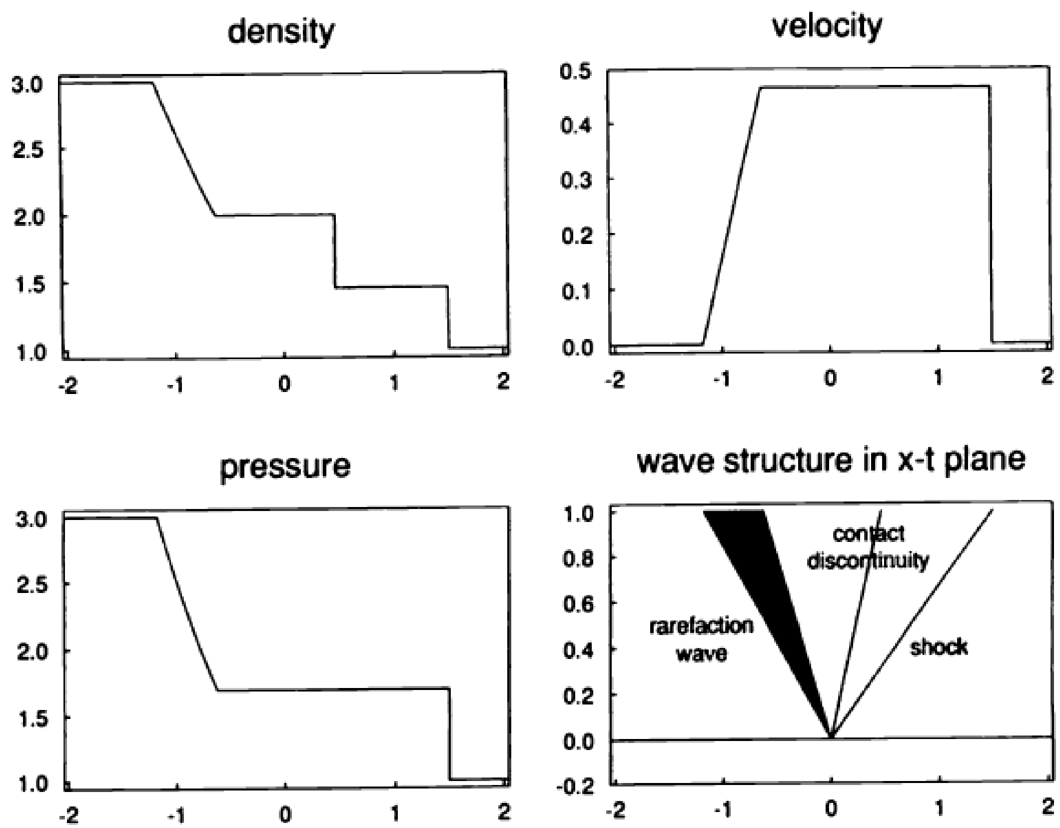
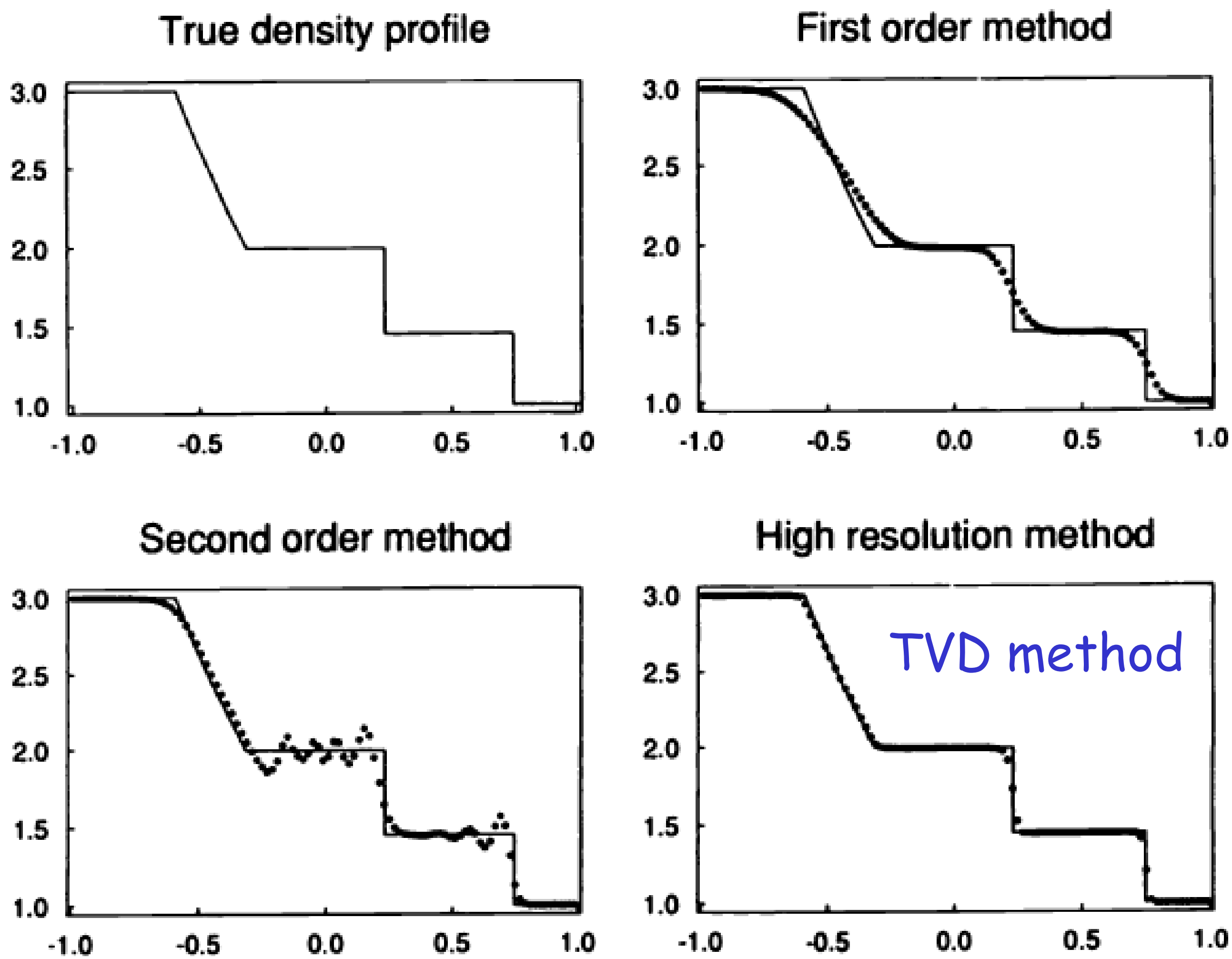


Figure 1.1. Solution to a shock tube problem for the one-dimensional Euler equations.

(Courtesy of LeVeque, CUP)

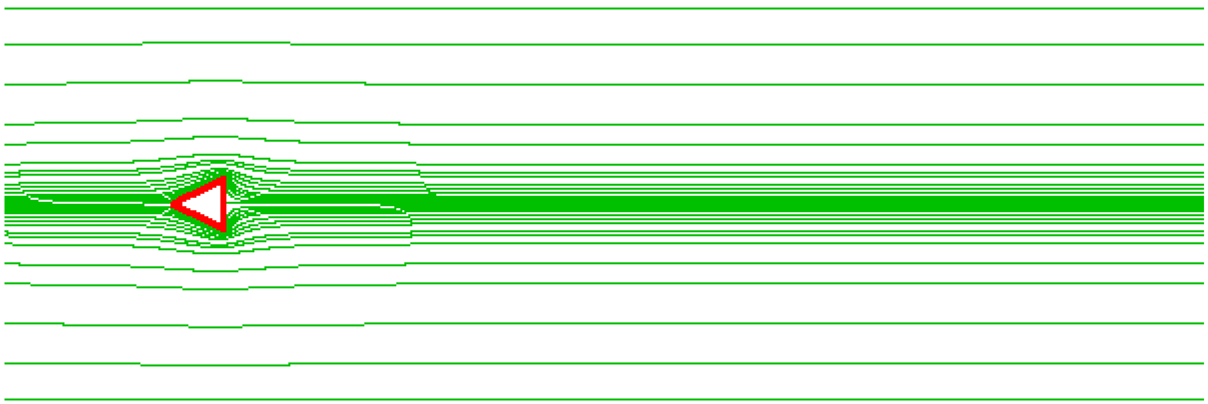
Diffusion, Dissipation and TVD Method



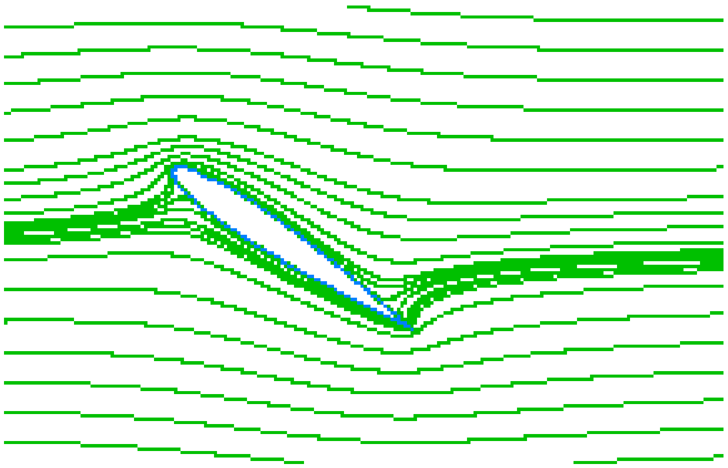
(Courtesy of LeVeque, CUP)

Incompressible Flows
(Pressure based Solver)

• Von Karman
Vortex Street



• Flow around NACA0012
Re = 1000



experiment (Shen et al.) *numerical* simulation

Compressible Flows vs Incompressible Flows

Mach Number: U/a, a =: sound speed

F/A-18, Sound barrier
Courtesy of wiki

Courant–Friedrichs–Lewy (*CFL*) condition (explicit methods):

$$\Delta t < \frac{\Delta x}{U + a}$$

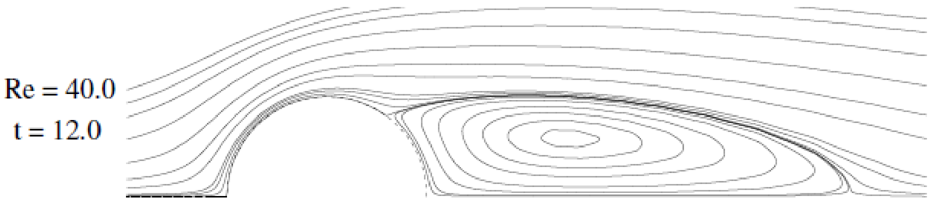
Incompressible flow: $a = \infty$

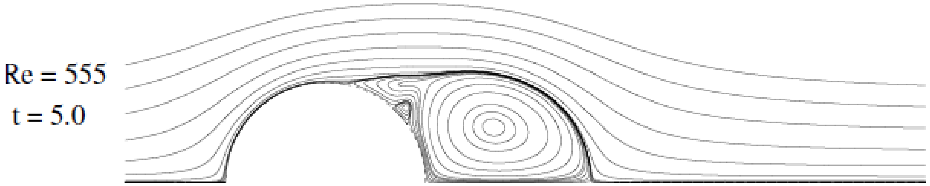
This course focuses on compressible flows

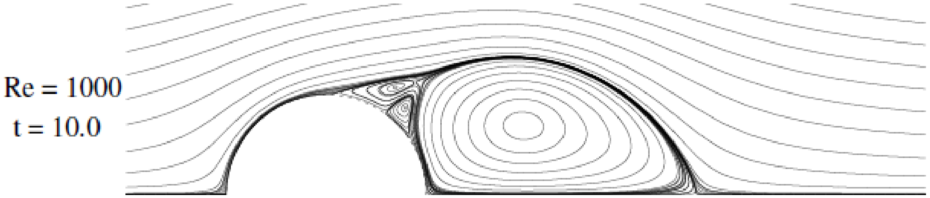
Not suitable for incompressible flows

Conforming Boundary and Non-Conforming methods (no-slip condition on wall)

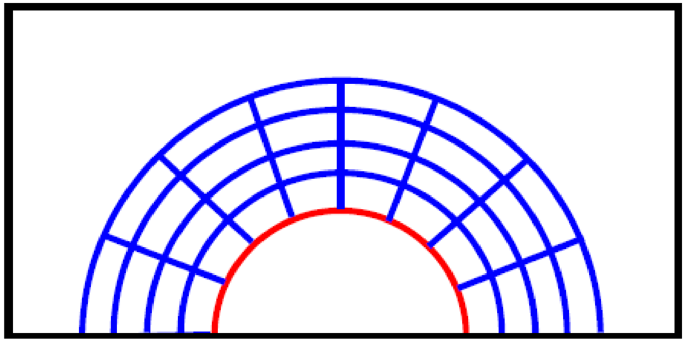
- Flow passing a cylinder



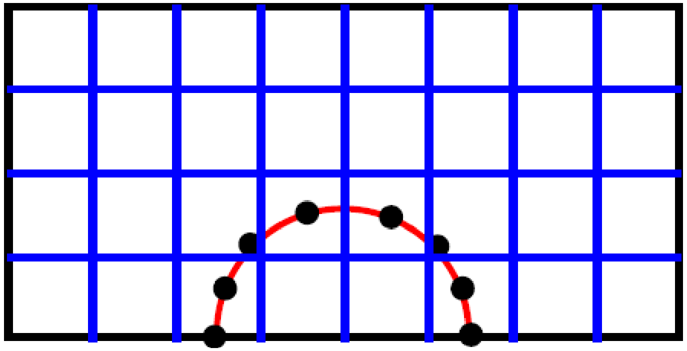




- Body fitting methods

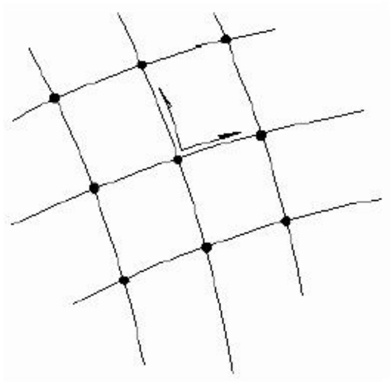


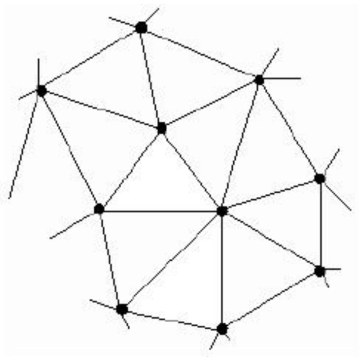
- Cut-cell methods
- Immersed boundary methods

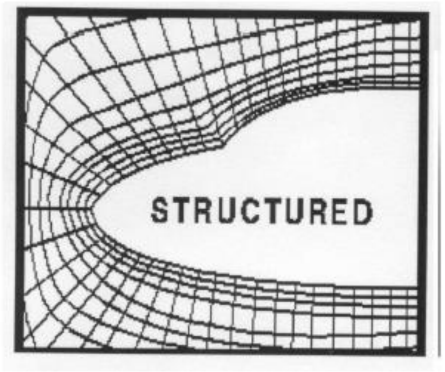


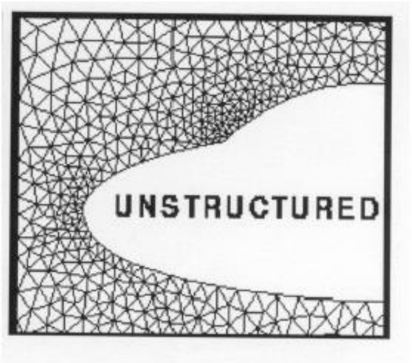
Structured and Unstructured Meshes

- Structured : fixed neighbour relations, efficient
- Unstructured: variable neighbour relations, flexible
- Mixed type meshes



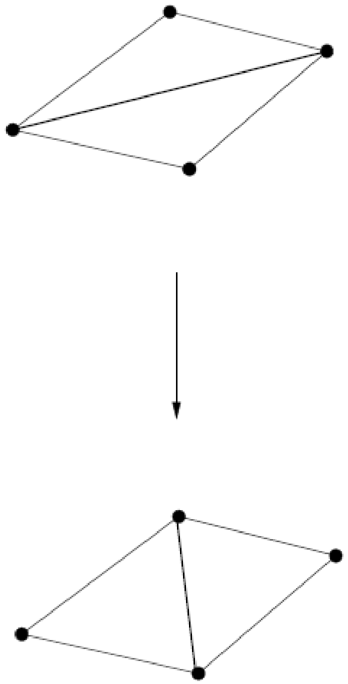


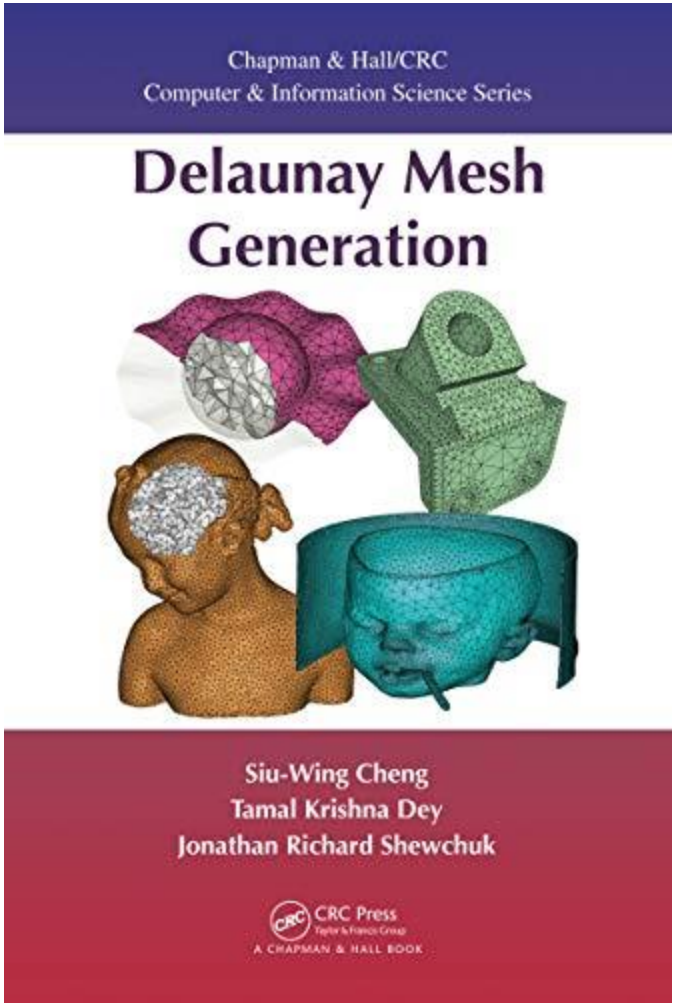




Delaunay Mesh Generation

Edge Flip:

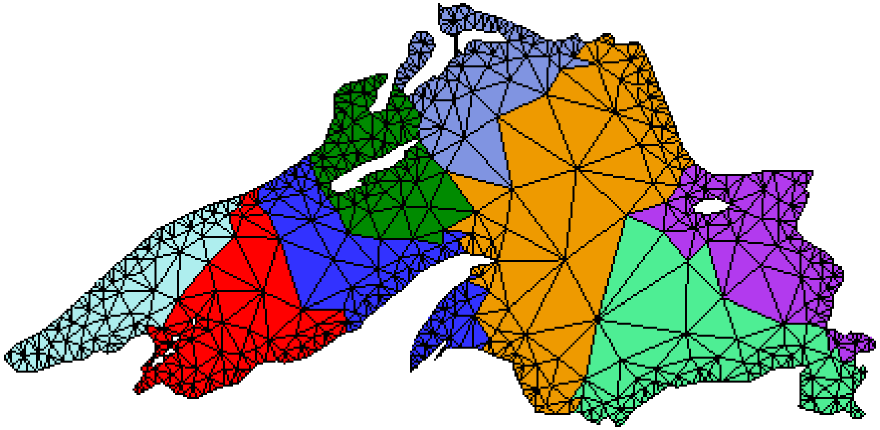


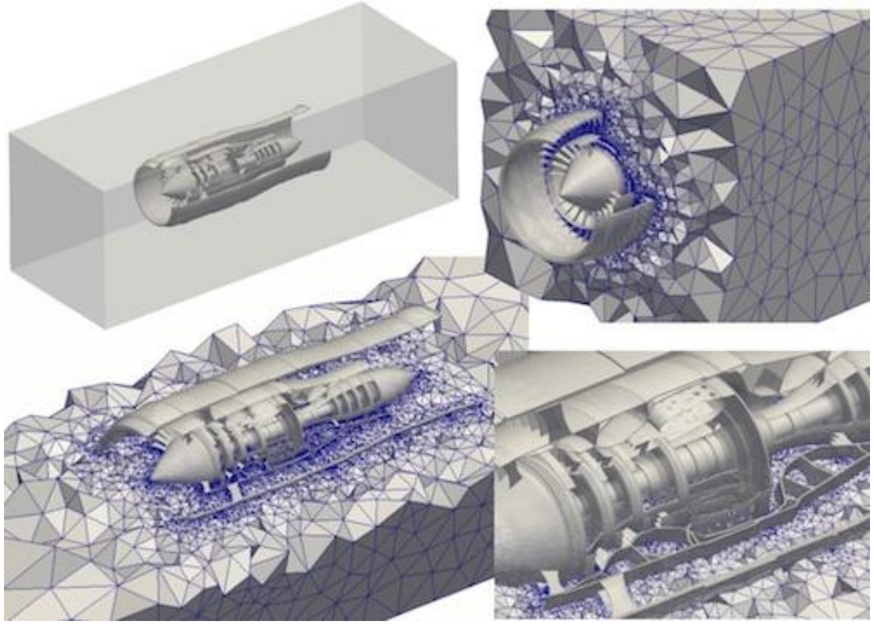


Mesh Generators in Public Domain

• [Triangle (Prof. J. Shewchuk)]

Berkeley University, California





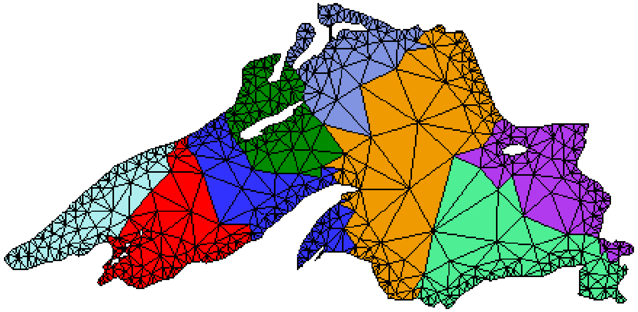
• [TetGen (Dr. Hang Si)]

Weierstrass Institute, Berlin

Mesh Generators in Public Domain

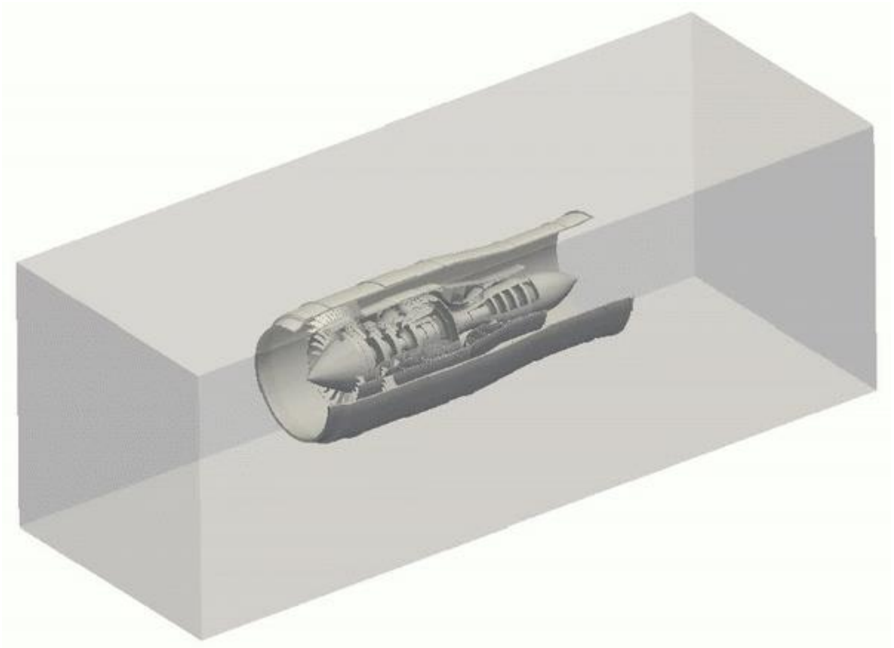
• Triangle (Prof. Jonathan Shewchuk)

Berkeley University, California



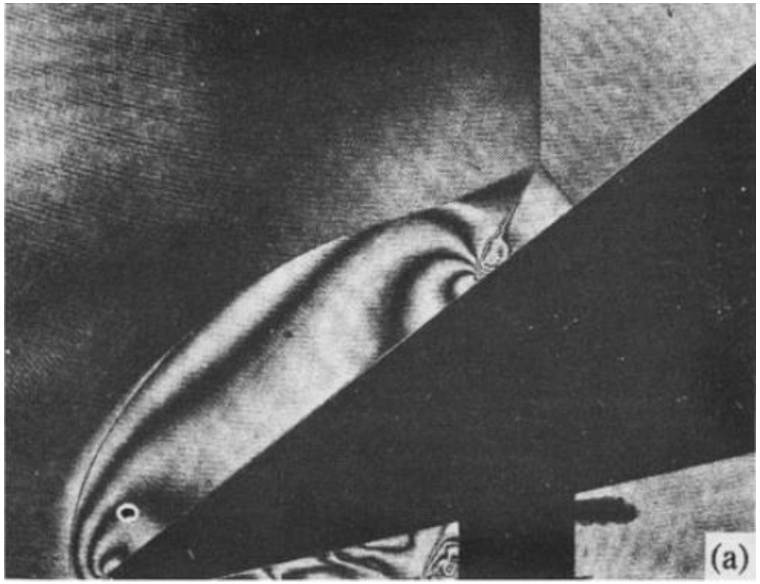
• TetGen (Dr. Hang Si)

Weierstrass Institute, Berlin

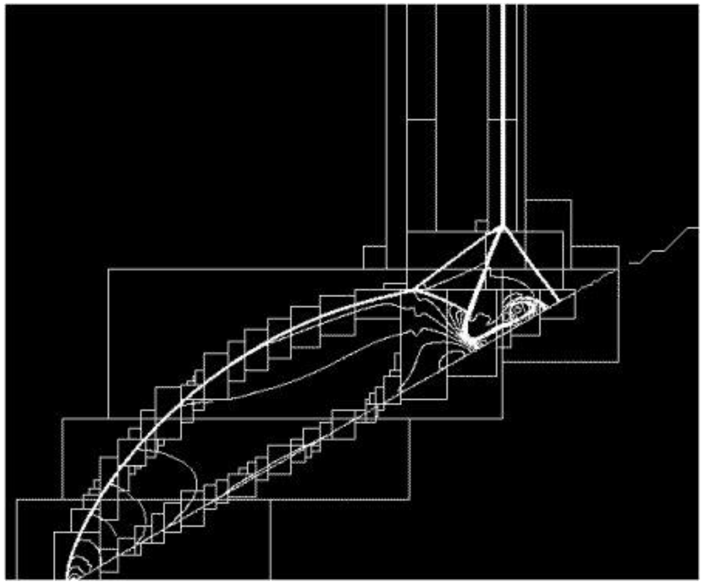


Adaptive Mesh Refinement (AMR)

Double Mach Reflection



experiment

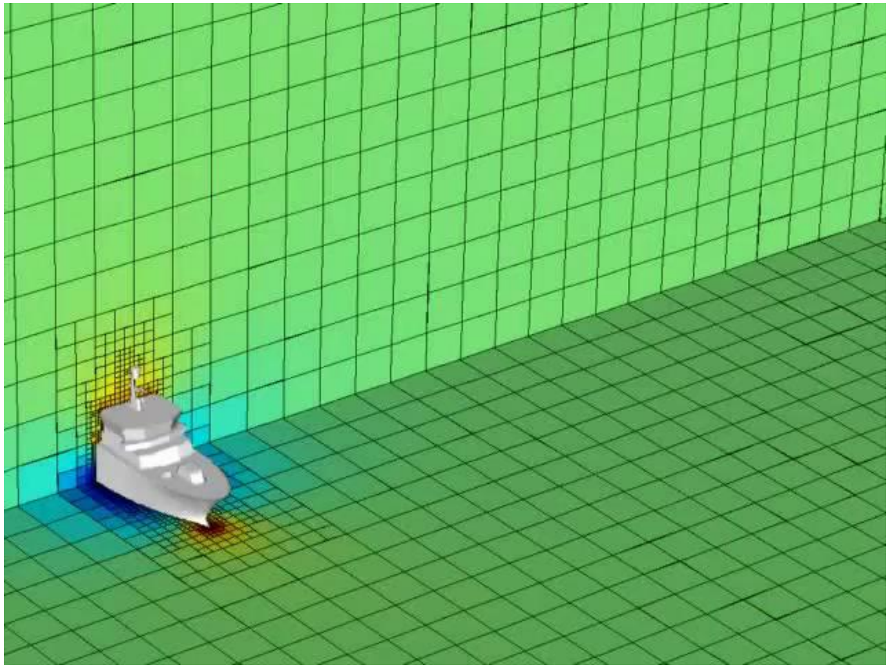


AMR

Berger & Colella (1989). "Local adaptive mesh refinement for shock hydrodynamics". J. Comput. Phys. 82: 64–84

Software Packages in Public Domain

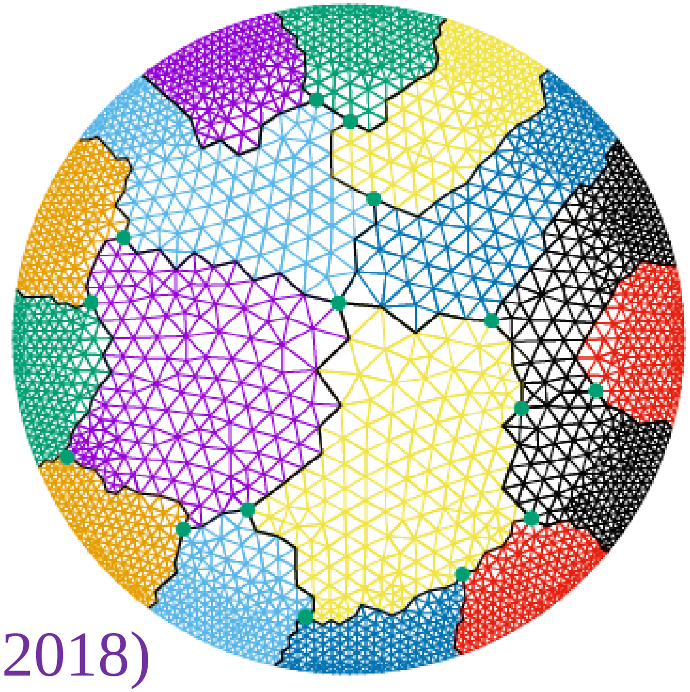
- FENICS
- OpenFoam
- Gerris (Oct-Tree)
- Fludity
- FreeFEM



Courtesy of Stephane Popinet, Gerris

Parallel Computing

- Memory Limitation & Computation Time
- [Message Passing Interface (MPI)]
- [ParMetis: Mesh Partition]
- [Linear Solver: PETSc]

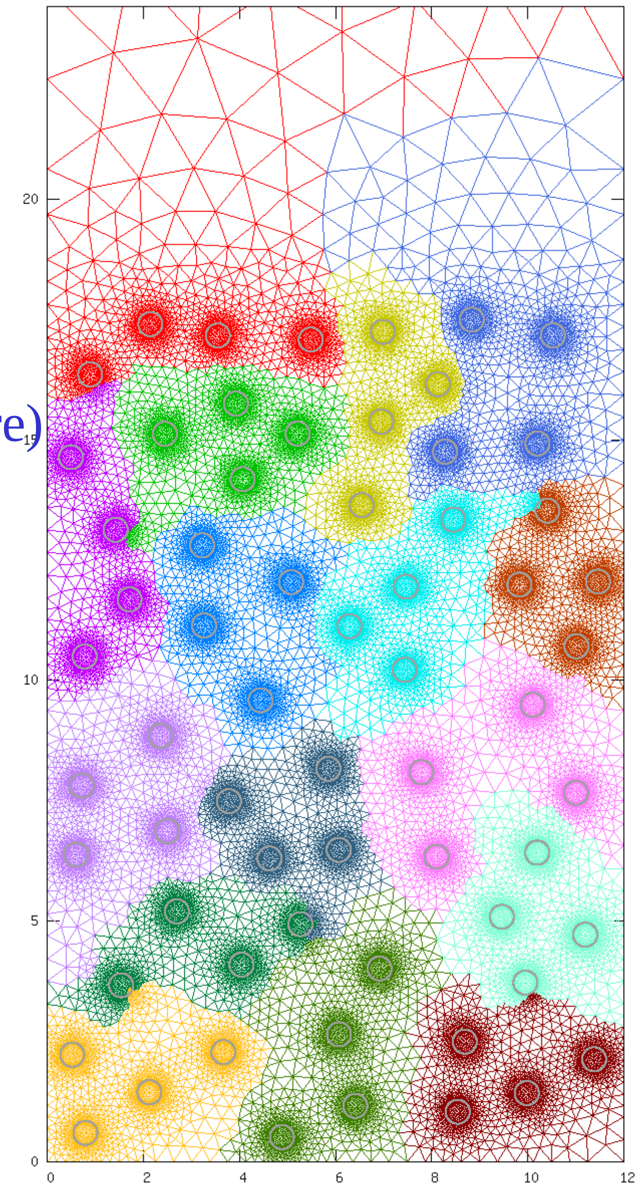


- Summit (Oak Ridge National Lab, 2018)
- 4608 nodes
- 9,216 CPUs & 27,648 GPUs
- speed: 200 petaflops

Partitioned into 16 Parts using ParMetis

Parallel Computation of 64 Rising Bubbles

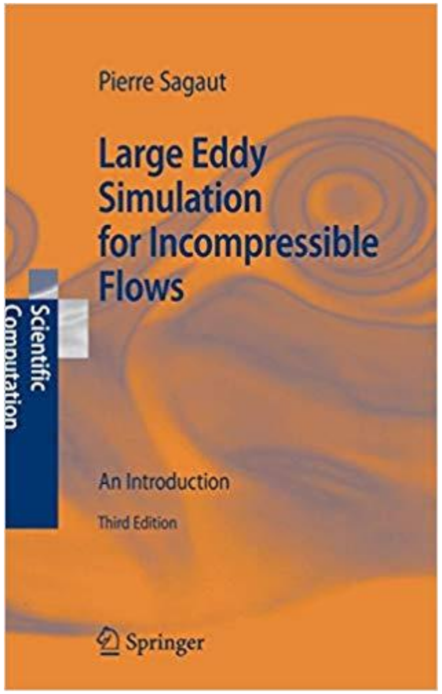
- Limit of Memory of a Single Machine
- Duration of computing time
- Partition of Mesh by ParMetis
- Incompressible Flows
- Pressure based method (implicit on pressure)
- Linear Solver PETSc

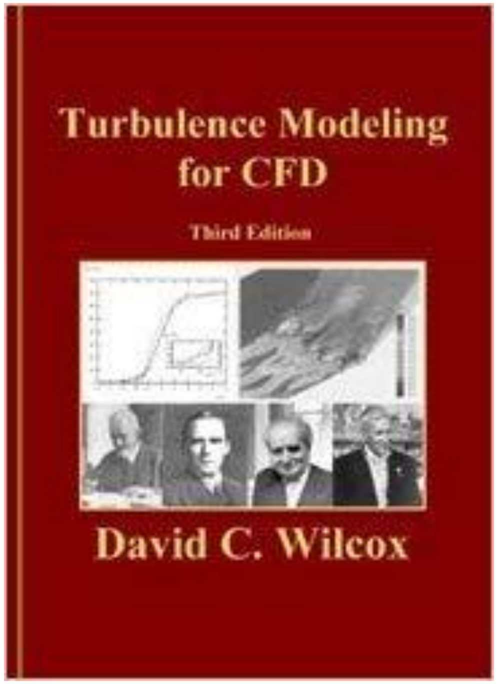


Rising bubbles computed
on 16 cores by Shidi Yan

DNS, LES and RANS

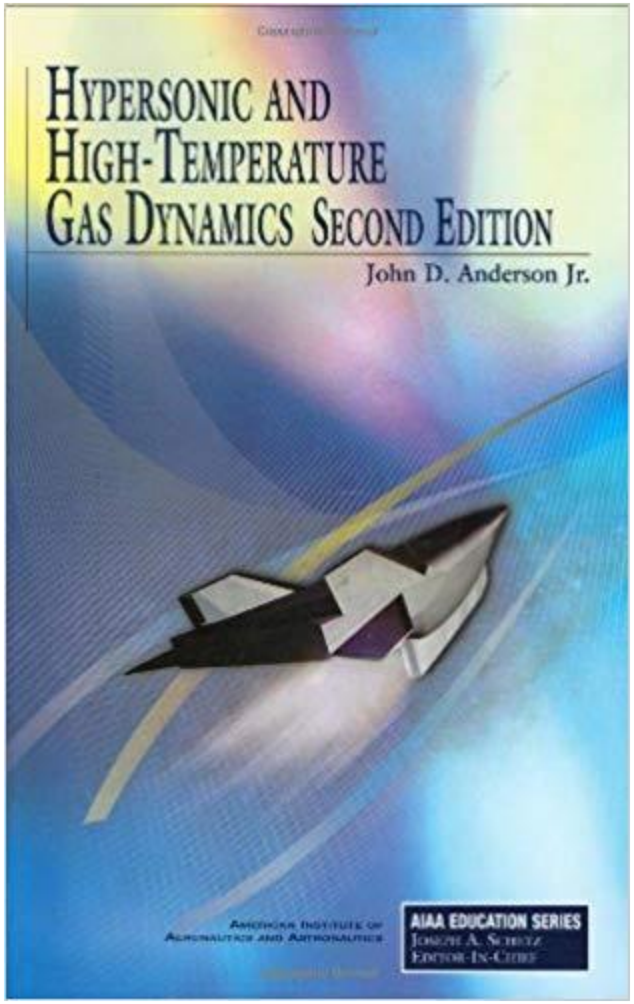
- **Direct Numerical Simulations (DNS)**
- *Kolmogorov* scale: $\eta = (\nu^3/\epsilon)^{1/4}$
- 3D DNS requires mesh points : $N^3 = Re^{9/4} = Re^{2.25}$
- **Large Eddy Simulations (LES)**
- **Reynolds Average Navier-Stokes Eqns (RANS)**



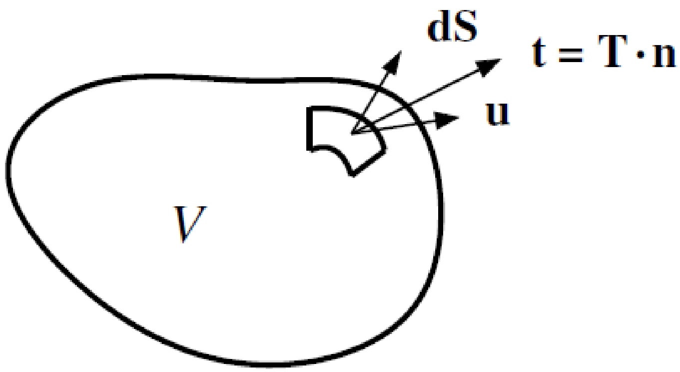


Hypersonic and High-Temperature Flows

- Ma > 5.0?
- Thin shock layer
- High temperature
- Dissoication and ironizationof gas



Conservation Equations of Compressible Flows



Cauchy Stress Tensor: $\mathbf{T} = -p\mathbf{I} + \boldsymbol{\tau}$

Stokes: $\boldsymbol{\tau} = -\frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} + \mu\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right)$

Fourier heat flux: $\mathbf{q} = -\kappa\nabla T$

Mass: $$\frac{\mathrm{d}}{\mathrm{dt}}\int_V \rho \mathrm{dV} + \int_S \rho\mathbf{u}\cdot \mathrm{d}\mathbf{S} = 0$$

Momentum (**f** body forces) :

$$\frac{\mathrm{d}}{\mathrm{dt}}\int_V \rho\mathbf{u}\mathrm{dV} + \int_S \rho\mathbf{u}(\mathbf{u}\cdot\mathrm{d}\mathbf{S}) = \int_V \rho\mathbf{f}\mathrm{dV} - \int_S \mathbf{T}\cdot\mathrm{d}\mathbf{S}$$

Energy (**Q** heat sources):

$$\frac{\mathrm{d}}{\mathrm{dt}}\int_V \rho E\mathrm{dV} + \int_S \rho E\mathbf{u}\cdot\mathrm{d}\mathbf{S} = \int_S \mathbf{u}\mathbf{T}\cdot\mathrm{d}\mathbf{S} + \int_V \rho\mathbf{f}\cdot\mathbf{u}\mathrm{dV} - \int_S \mathbf{q}\cdot\mathrm{d}\mathbf{S} + \int_V \rho Q\mathrm{dV}$$

Navier-Stokes Eqns for Compressible Flows

Mass: $$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

Momentum: $$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot (\boldsymbol{\tau}) + \mathbf{f}$$

Energy: $$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{u}) = -\nabla \cdot (p \mathbf{u}) + \nabla \cdot (\mathbf{u} \boldsymbol{\tau} - \mathbf{q}) + \rho (\mathbf{f} \cdot \mathbf{u}) + \rho Q$$

Boundary Conditions: *no-slip* on wall, ….

Constitutive Laws:

Stokes Viscous stress: $$\boldsymbol{\tau} = -\frac{2}{3}\mu (\nabla \cdot \mathbf{u}) \mathbf{I} + \mu \left(\nabla \mathbf{u} + \nabla \mathbf{u}^T \right)$$

Total Energy: $$E = e + \frac{\mathbf{u}^2}{2}$$

Ideal gas: $$p = R \rho T$$

Polytropic gas: $$e = c_v T$$

Fourier heat flux: $$\mathbf{q} = -\kappa \nabla T$$

4A2: Euler Equations in 2D: $\mu = 0$ and $\kappa = 0$

Mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0 \ ,$$

Momentum:

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} = -\frac{\partial p}{\partial x} \ ,$$

$$\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho u v)}{\partial x} + \frac{\partial (\rho v^2)}{\partial y} = -\frac{\partial p}{\partial y} \ ,$$

Energy:

$$\frac{\partial (\rho E)}{\partial t} + \frac{\partial \left((\rho E + p) u\right)}{\partial x} + \frac{\partial \left((\rho E + p) v\right)}{\partial y} = 0 \ .$$

Ideal gas: $p = R \rho T$

Polytropic gas: $e = c_v T$

Speed of Sound: $a^2 = \frac{\gamma p}{\rho} = \gamma R T$

- **Finite Difference** Methods (solving differential equations by approximating them with difference equations, in which finite differences approximate the derivatives). For example,

$$\frac{df}{dt} \approx \frac{f(t+\Delta t) - f(t)}{\Delta t}$$

- **Finite Volume** Methods (based on conservation laws. **Volume integrals** in a partial differential equation that contain a **divergence term** are converted to **surface integrals**, popular in Fluid Mechanics, especially with **hyperbolic** equations).

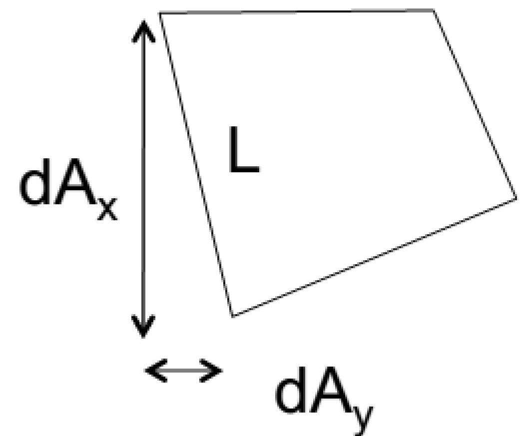
$$\int_{\Omega} \nabla \cdot \boldsymbol{q} dV = \int_{\partial\Omega} \boldsymbol{q} d\boldsymbol{S}$$

  The method has the important physical property that certain **conservation laws** are maintained.

- **Finite Element methods are based on weak form of PDE, require a lot of maths (*module 3D7*)**

Conservation Laws and Finite Volume method

$$Vol\frac{\Delta\rho[?]}{\Delta t} = \sum_{cvs} Flux$$



- Where for edge L:

$Flux_{mass} = m = \rho\ \mathbf{V.dA} = \rho\ V_x dA_x + \rho\ V_y dA_y$

$Flux_{mom} = mu = (\rho\ \mathbf{V.dA})\mathbf{V}$

or

$Flux_{x,mom} = (\rho\ V_x dA_x + \rho\ V_y dA_y)V_x + P\ dA_x$

$Flux_{y,mom} = (\rho\ V_x dA_x + \rho\ V_y dA_y)V_y + P\ dA_y$

Flux + Pressure Source
-> Strong Conservation Eqn.