ENGINEERING COMPUTATIONAL FLUID DYNAMICS (ECFD)

Dr Xiangdong Li

Module 1 – Introduction

Lecturer

Dr. Xiangdong Li

- I am an engineering researcher
  - PhD in Refrigeration and Cryogenics Engineering
  - Expertise in Computational Fluid Dynamics (CFD), Computational Multiphase Flow (CMF)
  - Active research in buildings/energy/manufacturing sectors – using CFD as a tool
  - 100+ research publications. See me on Scopus (ID: 57190485649)

- I am a tertiary educator
  - Graduate Certificate in Learning and Teaching Higher Education (Swinburne University of Technology, Australia)
  - Guest Professor at SWJTU-OSU Joint Program

Li, Xiangdong - Author details
scopus.com/authid/detail.uri?authorId=57190485650
Apps OneDrive Seek Life Roy Jonathan Xiangdong RMIT CFD DP23 Journals Reading list

Brought to you by RMIT University Library

Scopus

Search Sources Lists SciVal

This author profile is generated by Scopus Learn more

Li, Xiangdong

RMIT University, Melbourne, Australia Show all author info

57190485650 https://orcid.org/0000-0002-8669-6314

Edit profile Set alert Save to list Potential author matches Export to SciVal

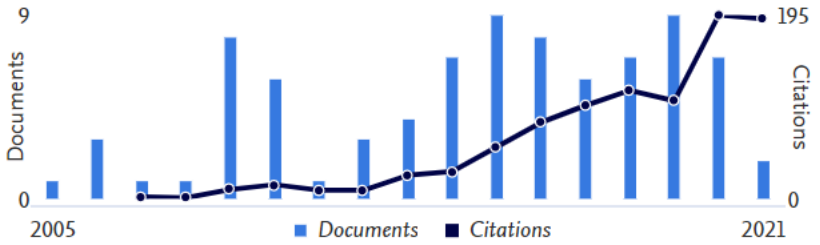
Metrics overview

83
Documents by author

940
Citations by **683 documents**

18
h-index: View *h*-graph

Document & citation trends



Analyze author output Citation overview

Most contributed Topics 2016–2020

Airborne Infection; Bioaerosols; Operating Rooms
12 documents

Geothermal Heat Pumps; Heat Exchangers; Boreholes
5 documents

Boiling; Nanofluids; Heat Flux
4 documents

View all Topics

Beta

10:58 PM
3/09/2021

You are here
I am here
Russia
Mongolia
Kazakhstan
Uzbekistan
Afghanistan
Iran
India
Myanmar (Burma)
Laos
Thailand
Vietnam
Cambodia
Philippines
Taiwan
South Korea
Japan
Yemen
Oman
United Arab Emirates
Sri Lanka
Malaysia
Singapore
Indonesia
Papua New Guinea
Australia
New Zealand
Madagascar
Somalia
Indian Ocean
Pacific Ocean
Bering Sea
Sea of Okhotsk
Sea of Japan
East China Sea
South China Sea
Philippine Sea
Bay of Bengal
Andaman Sea
Gulf of Thailand
Gulf of Aden
Laccadive Sea
Celebes Sea
Java Sea
Banda Sea
Bismarck Sea
Arafura Sea
Timor Sea
Solomon Sea
Coral Sea
Tasman Sea
Laptev Sea
East Siberian Sea
Novosibirsk
Nur-Sultan
Beijing
Sapporo
Shanghai
Osaka
Hong Kong
Luzon
Palawan
Negros
Mindanao
Basilan Island
Jakarta
Bengaluru
Honolulu
HAWAII
Papeete
NORTHERN TERRITORY
QUEENSLAND
WESTERN AUSTRALIA
SOUTH AUSTRALIA
Perth
Adelaide
Brisbane
Auckland
Tauranga
TASMANIA
Southern Ocean
Google

Melbourne, 2nd largest city in AUS
Most livable city （2011-2017）

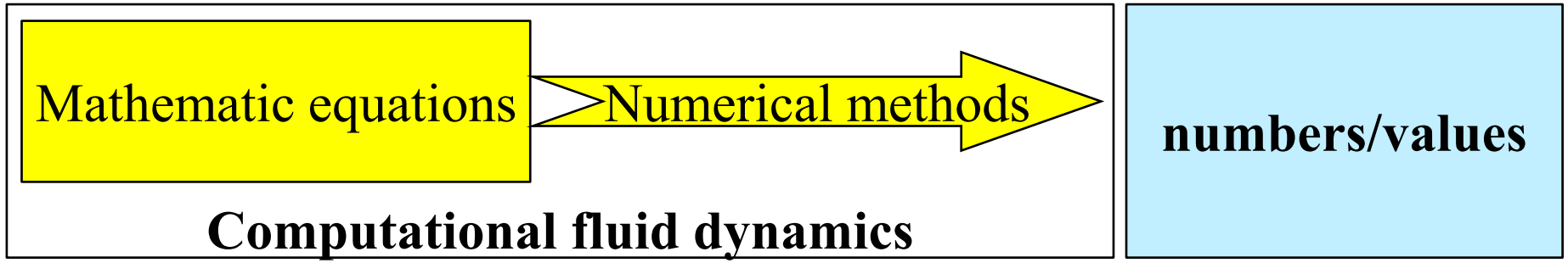
WHAT IS COMPUTATIONAL FLUID DYNAMICS (CFD)?

What's CFD?

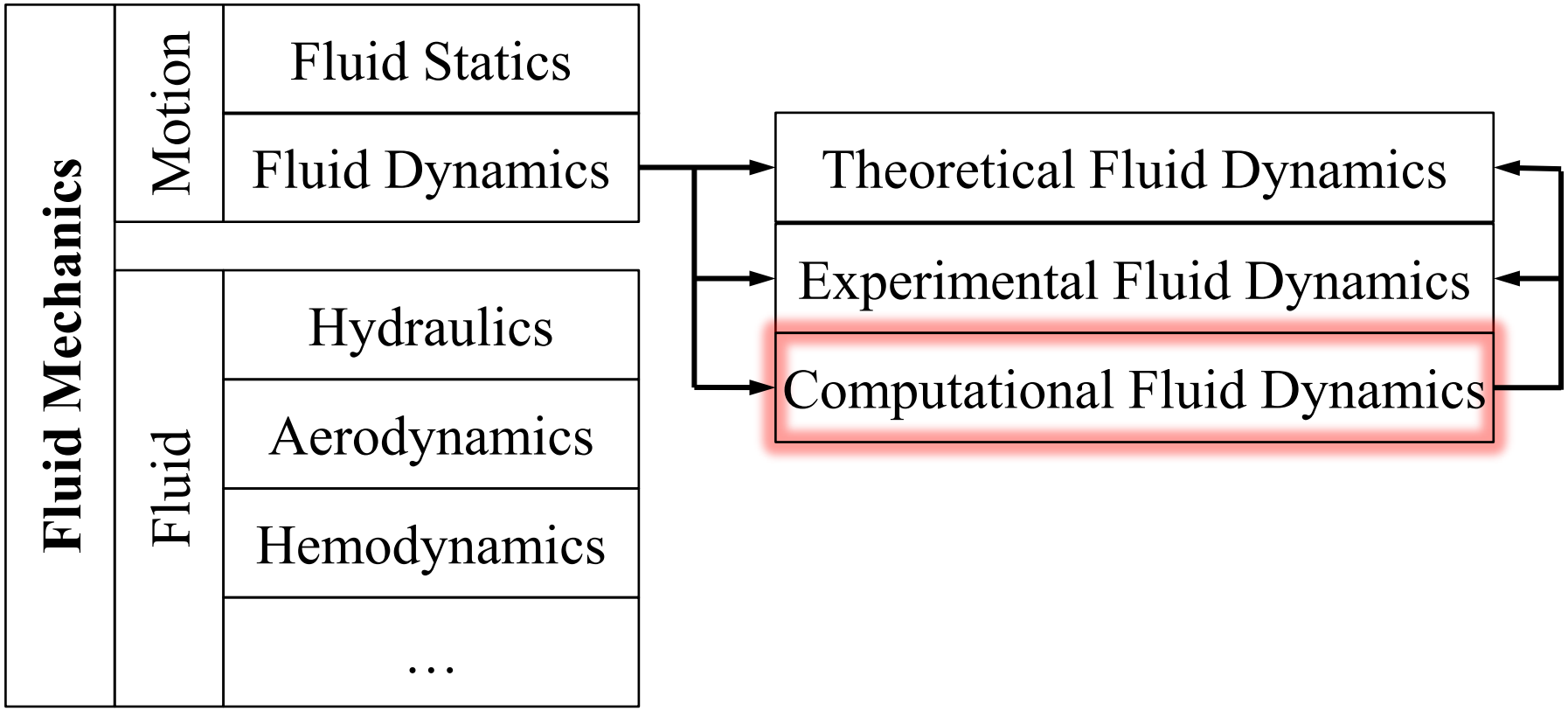
- J.D. Anderson: CFD is, in part, the art of replacing the governing partial differential equations of fluid flow with numbers, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow field of interest.
- Wikipedia: CFD is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.
- Simscale: CFD is the process of mathematically modeling a physical phenomenon involving fluid flow and solving it numerically using the computational prowess.

What's CFD?

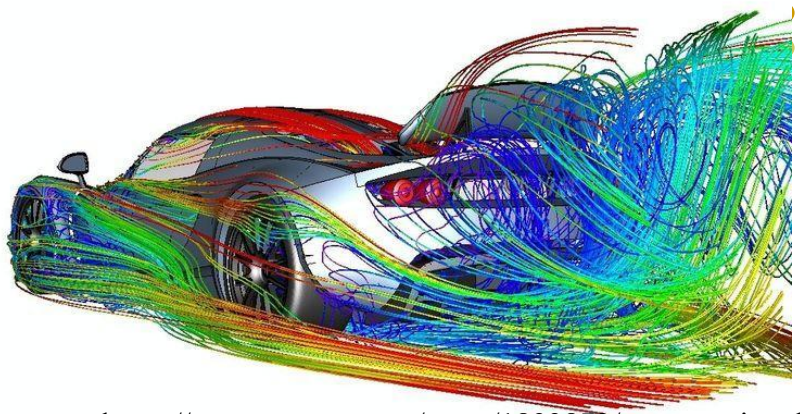
- No universally accepted definition
- Key components of CFD
  - Mathematics - fluid dynamics and multidisciplinary theories
  - Numerical methods - discrete numerical techniques and computing powers



CFD in the Fluid Mechanics family



CFD in the Fluid Dynamics family

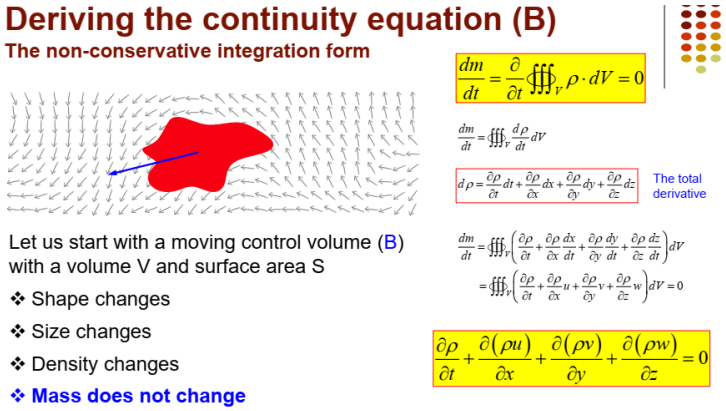


https://www.openpr.com/news/1893872/computational-fluid-dynamics-market-analysis-by-manufacturers

Computational Fluid Dynamics

Experimental Fluid Dynamics ↔ **Theoretical Fluid Dynamics**

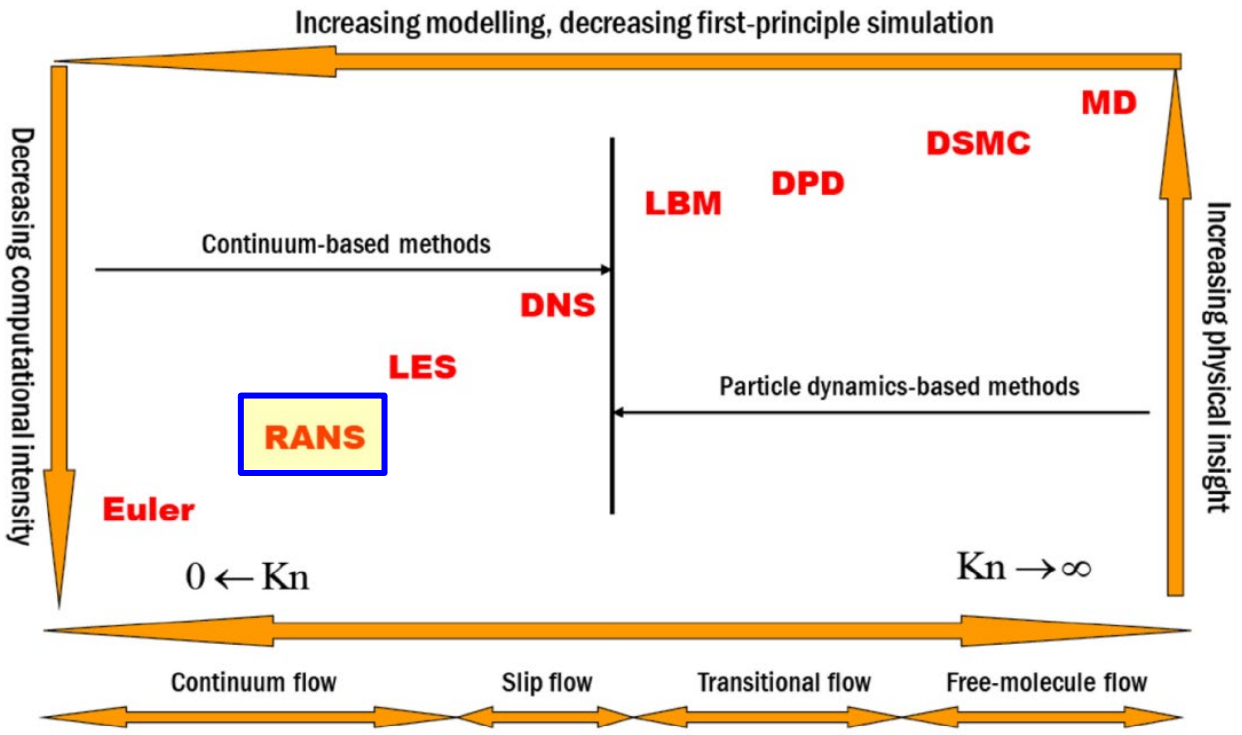
https://media.daimler.com/marsMediaSite/en/instance/picture.xhtml?oid=7603365



Various ways of flow simulation

CFD is built on mathematical equations describing fluid flows

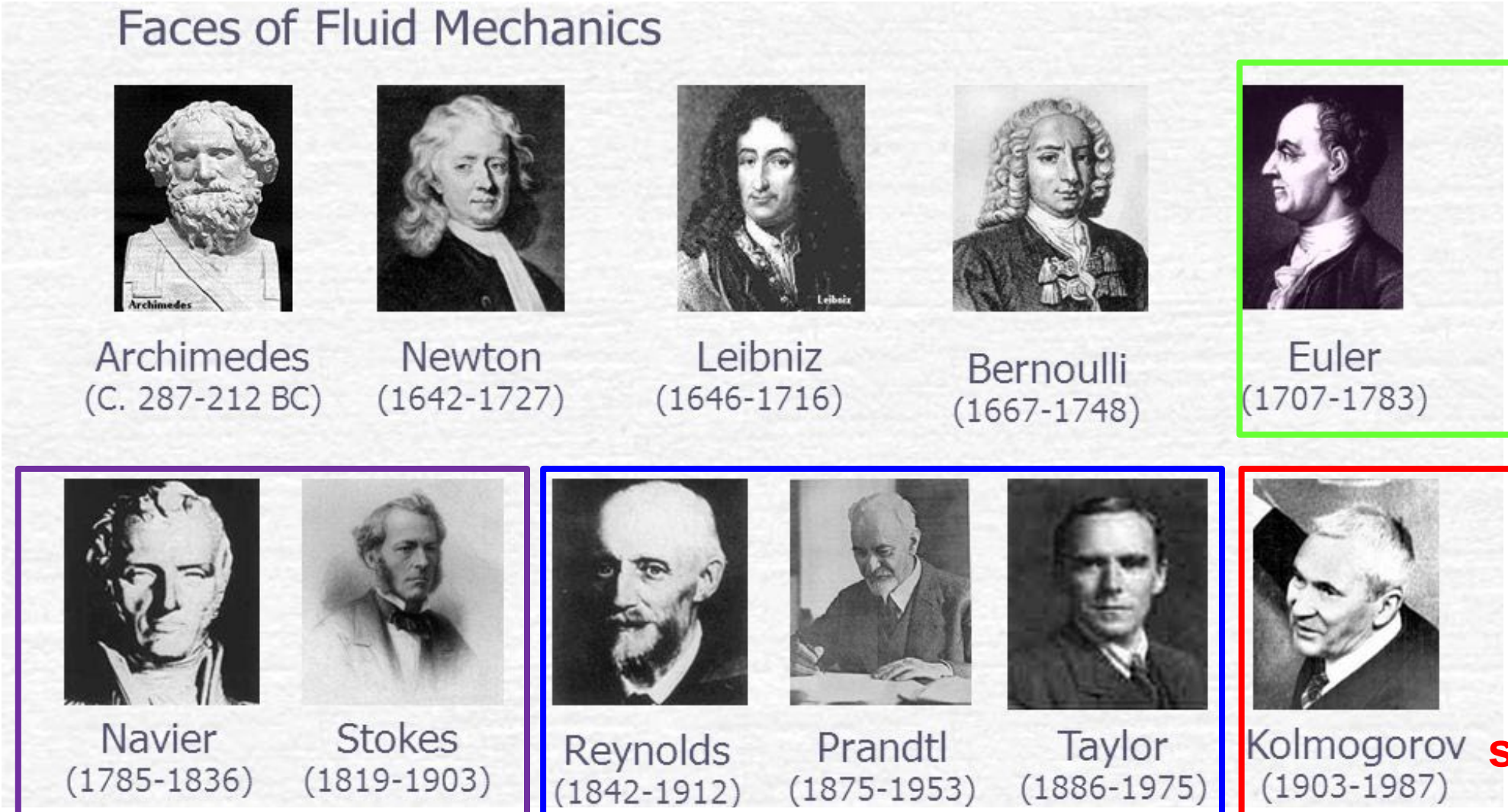
- ❖ Continuum-based methods
  - Euler
  - RANS
  - LES and DNS
- ❖ Particle-based methods
  - Molecules/atoms – MD
  - Fictious particles – LBM



https://www.sciencedirect.com/science/article/abs/pii/S0360128515300162

CFD
HISTORY, TODAY AND FUTURE

A bit of history - People



Euler equation

N-S equation

Boundary and Turbulence

Energy spectrum of turbulence

CFD in history

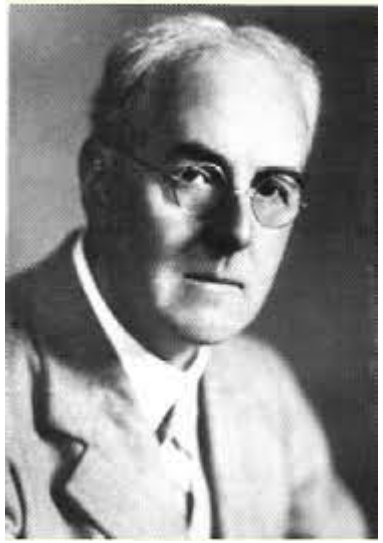
- Before 1700s: mass conservation, Newton's second law, energy conservation
- L. Euler (1707-1783), the Euler equation
- C.L. Navier (1785-1836) and G.G. Stokes (1819-1903), the Navier-Stokes (N-S) equations – $US 1 million prize offered by the Clay Mathematics Institute for an analytical solution
- Lewis F. Richardson (FDM, 1911), A. Thom (1933), …
- Los Alamos Lab, 1950-60's, started using digital computers
- Harlow and Fromm, 1965, "computer experiments"
- McDonald (1971) and MacCormack and Paullay (1972), finite volume method
- S.V. Patanka & D.B. Spalding, 1970 – 1980's, k-ε model, SIMPLE algorithm, up-wind differencing, …
- CHM Ltd. (UK), 1981, Phoenix – the world's first general-purpose commercial CFD package
- Exponential growth since 1980's: Fluent, CFX, OpenFoam, SimScale, …

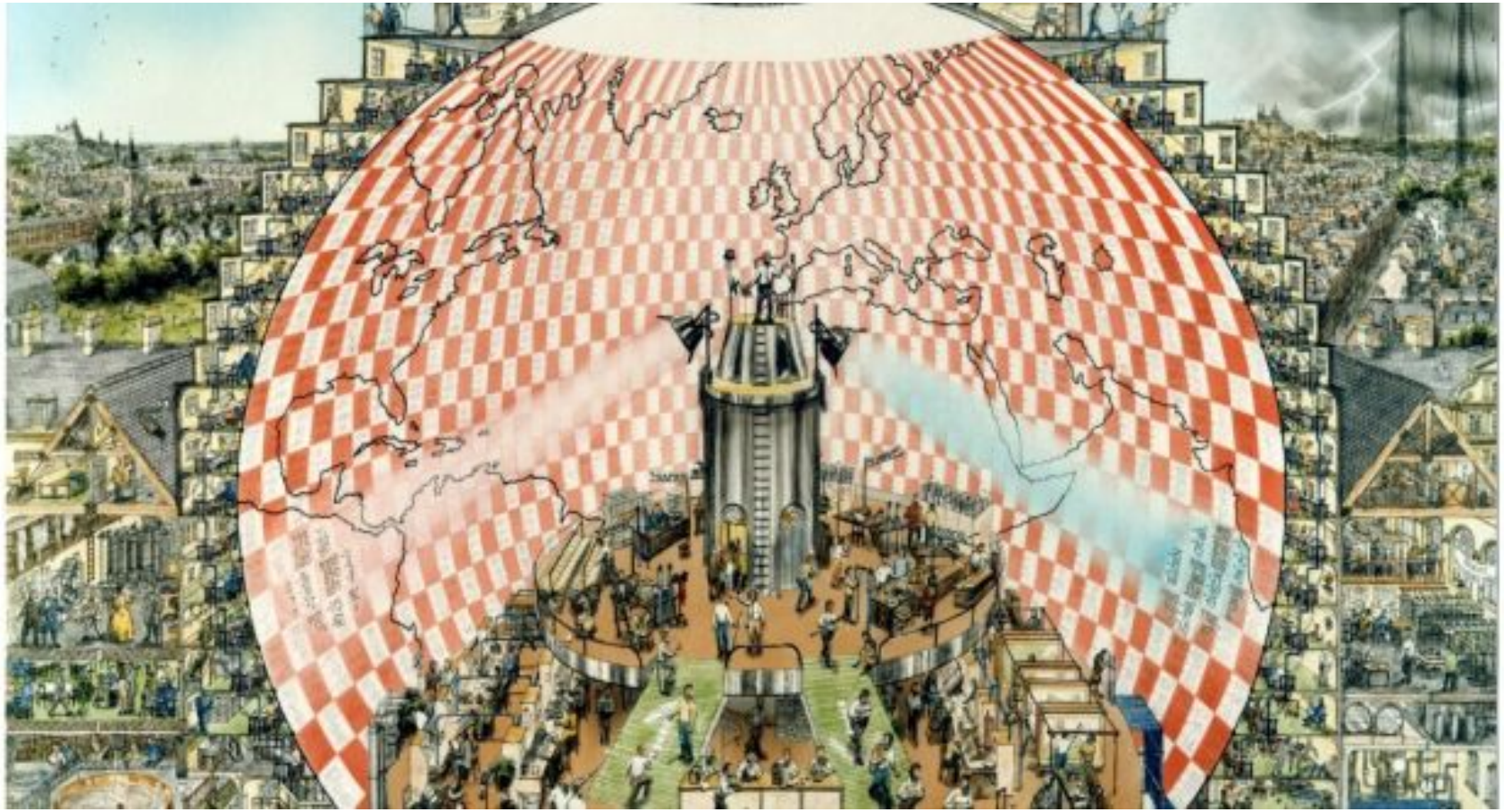
Manual Fluid Dynamics

Computational Fluid Dynamics

CFD in history – Event 1

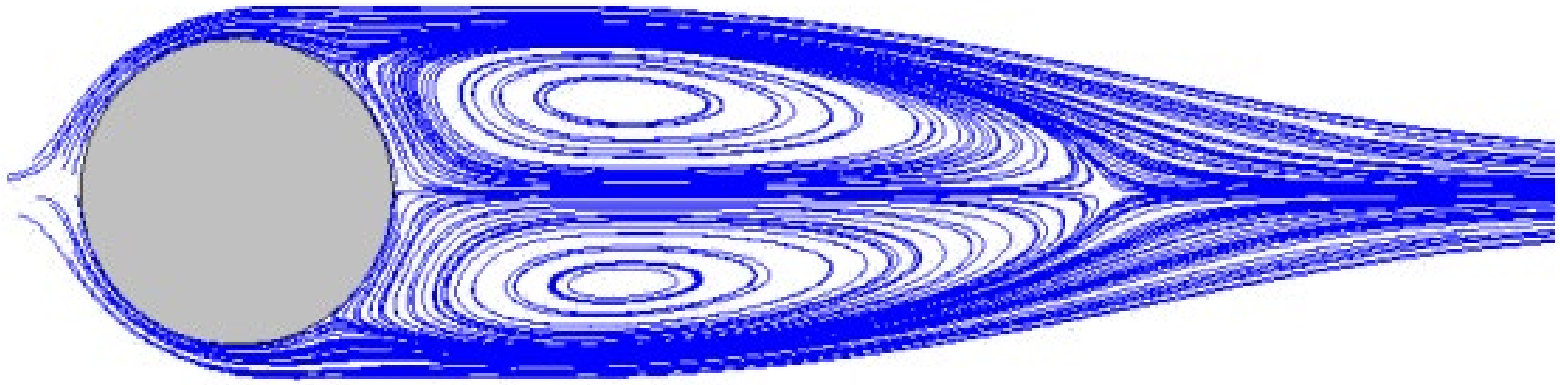
- Lewis Fry Richardson's (1881 - 1953) weather forecast factory (64,000 people)





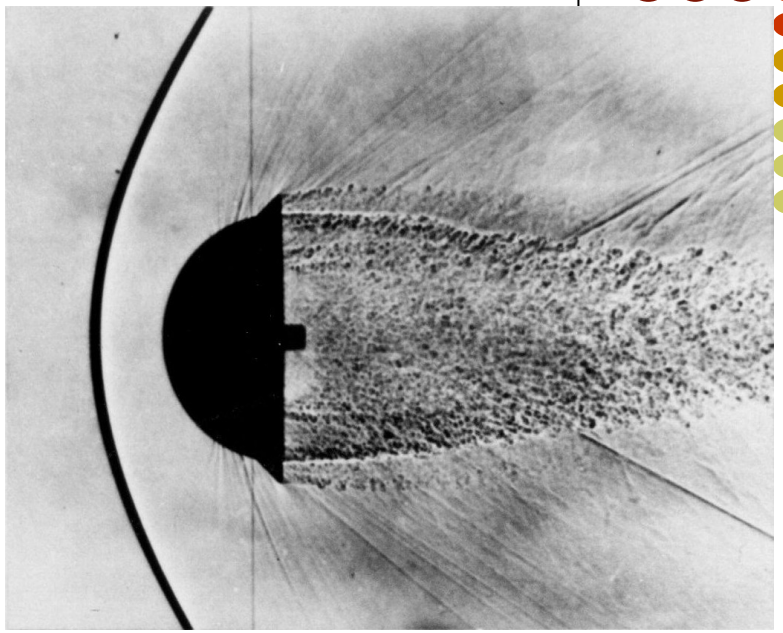
CFD in history – Event 2

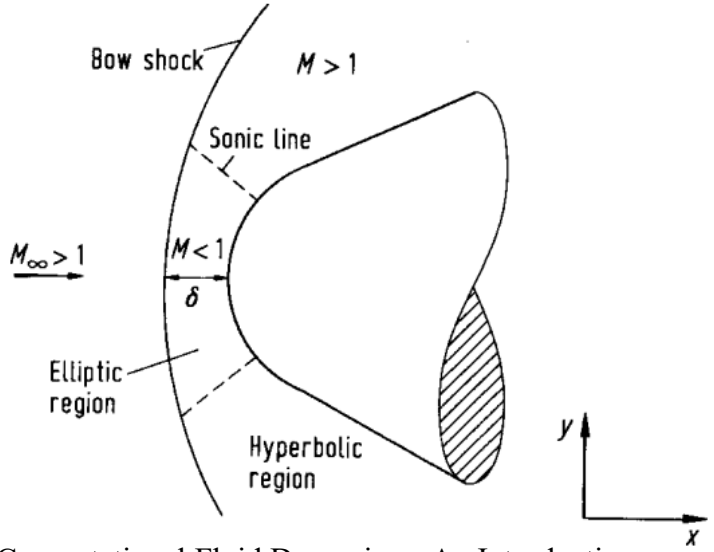
- A. Thom (1933). The flow past circular cylinder at low speeds, Proc Royal Society, A141, 651-666, London
- M. Kawaguti (1953). Numerical solution of the NS equations for the flow around a circular cylinder at Reynold number 40. J Phy Soc Japan, 8, 747-757. (Using a mechanical desk calculator, working 20 hours peer week for 18 months, citing “a considerable amount of labour and endurance”
- Today you only need a few minutes to finish this job.



CFD in history – Event 3

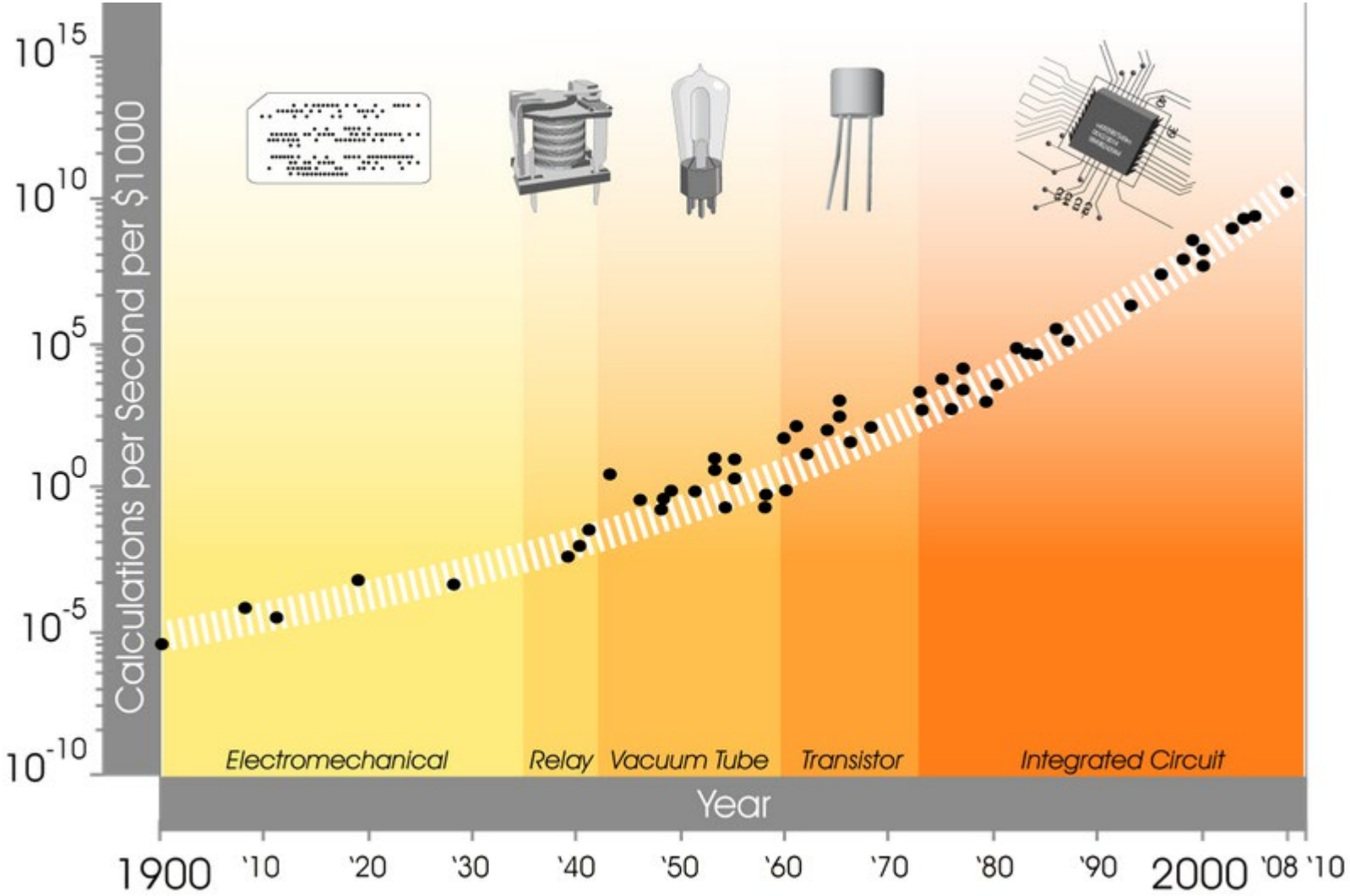
- Use of digital computers to solve the supersonic blunt body problem by the Institute of Aeronautical Sciences, NACA (later NASA) and US Air Force in the 1950-1960's.
- First implementation of 2D, transient, incompressible flow in the world: Moretti, G., Abbett, M., A time-dependent computational method for blunt body flows. AIAA J, 4(12), 1966: 2136-2141.
- Today this is just a homework for CFD students.



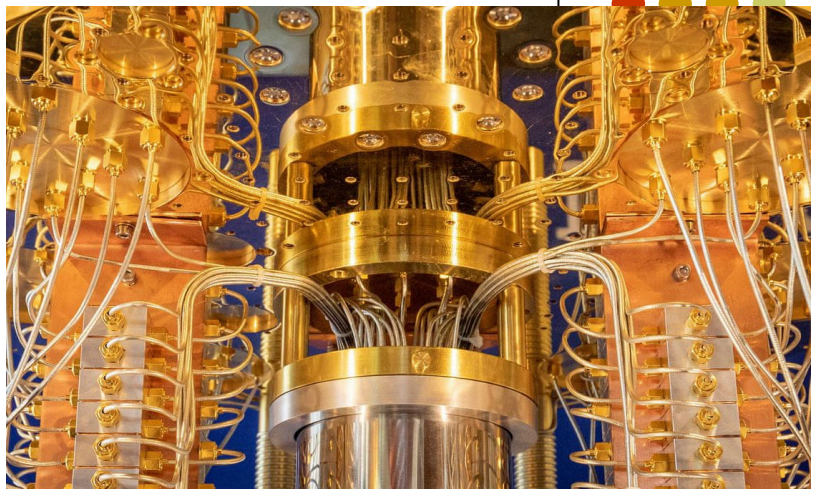


J.F. Wendt. Computational Fluid Dynamics – An Introduction

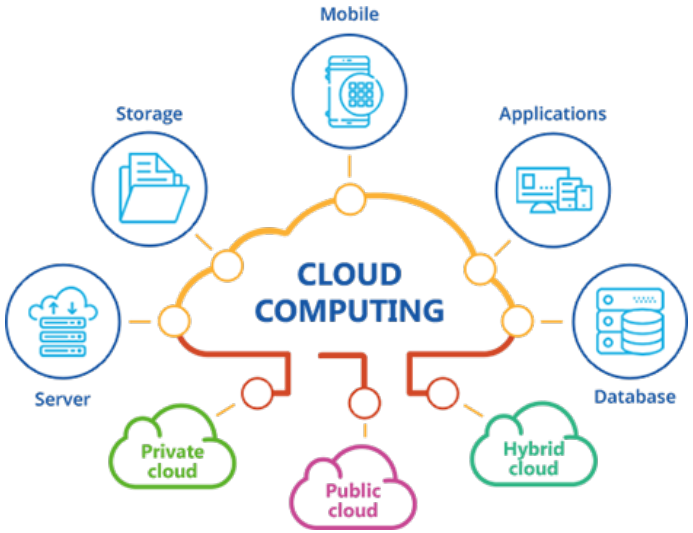
Driving force of CFD – Computing power



https://ourworldindata.org/technological-progress



Tomorrow



Today

Driving force of CFD – Computing power

Australia's Gadi (No. 24)

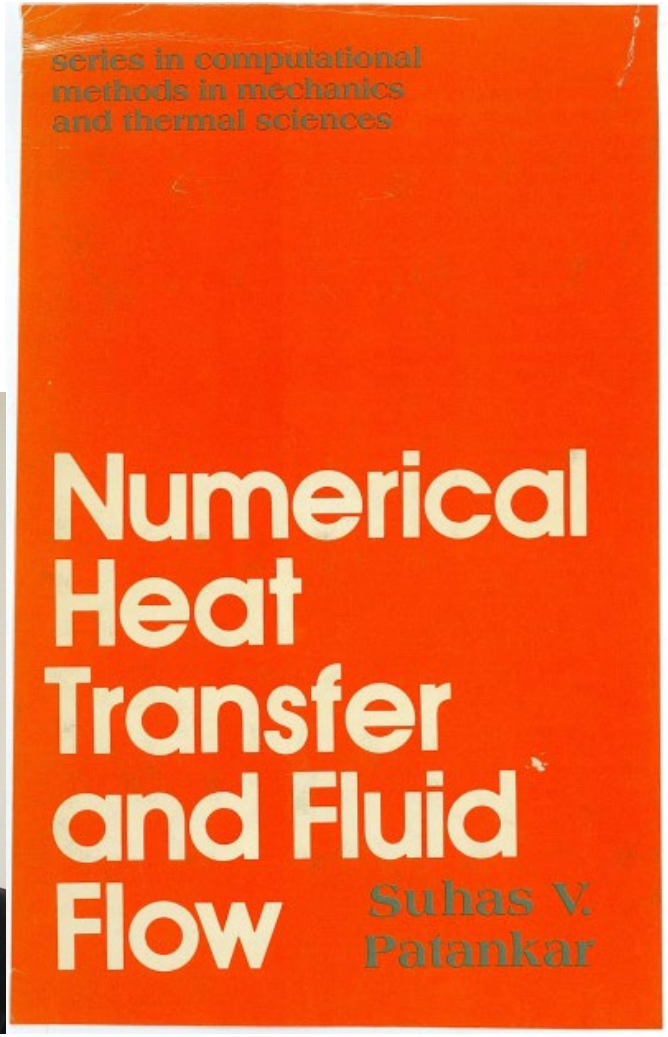
China's Sunway (No. 4)

https://www.top500.org/lists/top500/2021/06/

Driving force of CFD – Discrete numerical techniques

- Boundary element method
- Spectral methods
- Finite difference method
- Finite volume method
- Finite element method
- Vorticity based methods
- And more

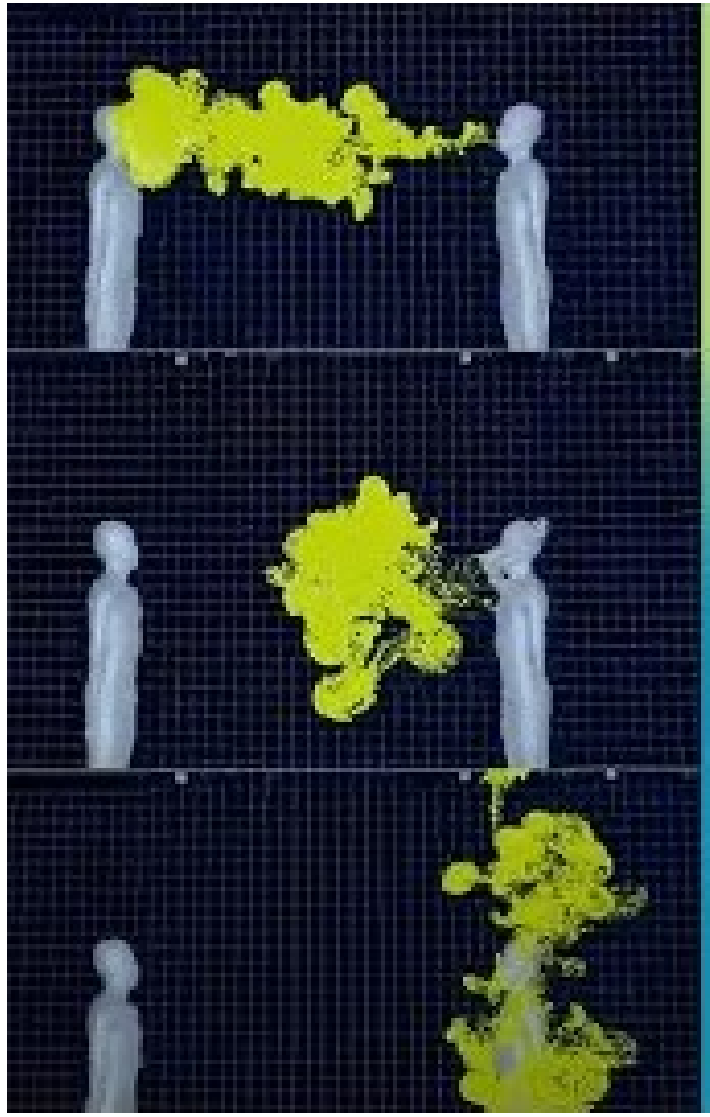
Published in 1980



CFD today

- CFD literally can deal with anything involving fluid flows
  - CFD is a research tool
  - CFD is a part of CAE/CAM

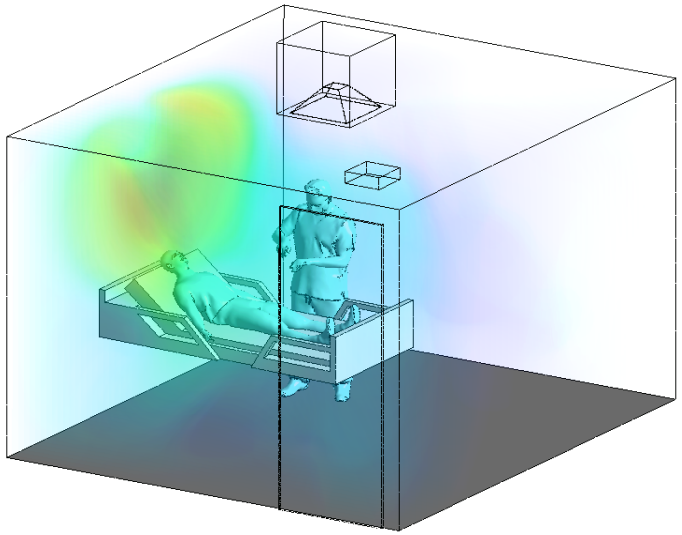
CFD today – COVID-19, mask or no mask?

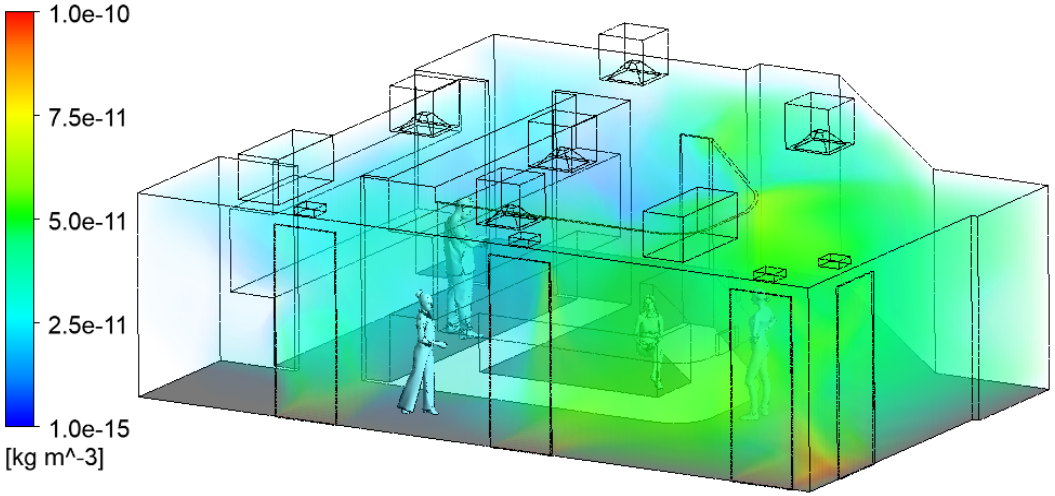


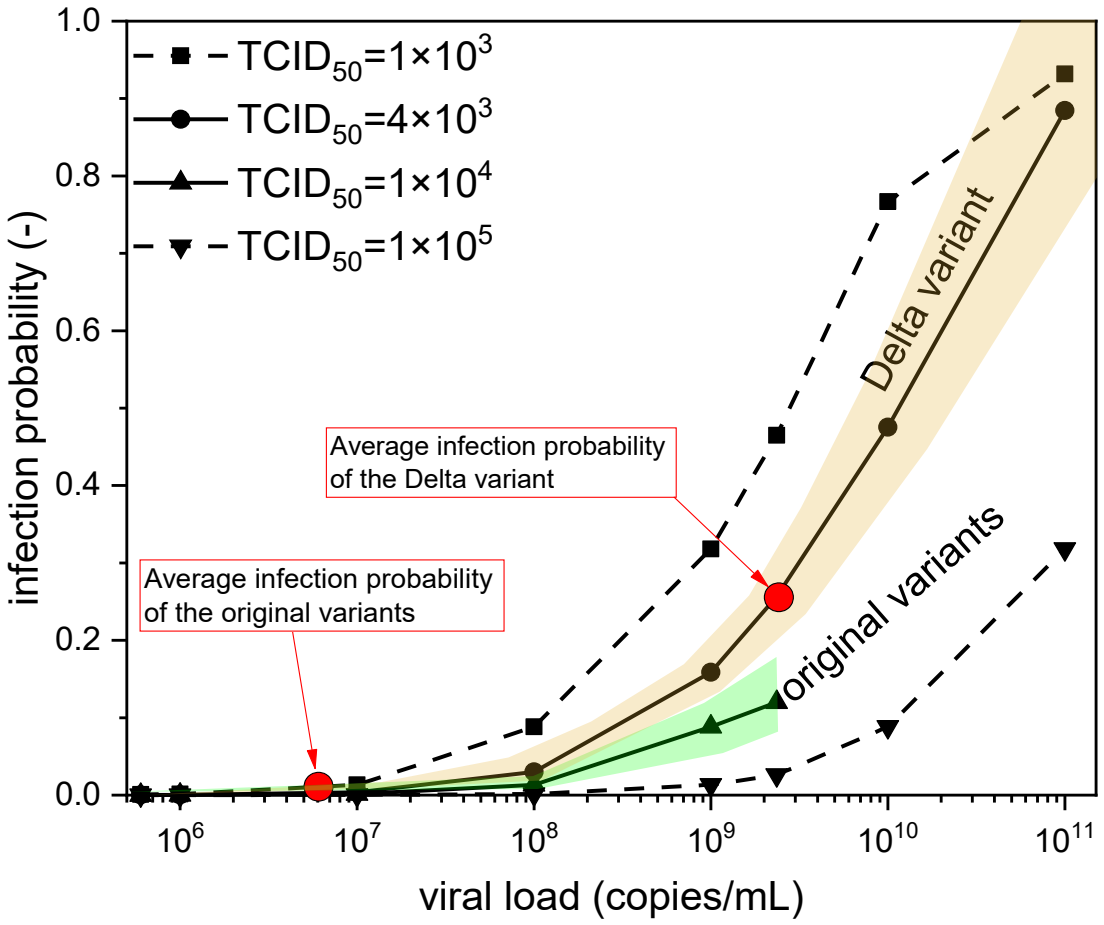
Coronavirus (Covid19):
See the impact of a sneeze
with no protection, with an elbow, and with a mask and how germs could potentially spread without the proper face protection.

HEXAGON | MSC Software

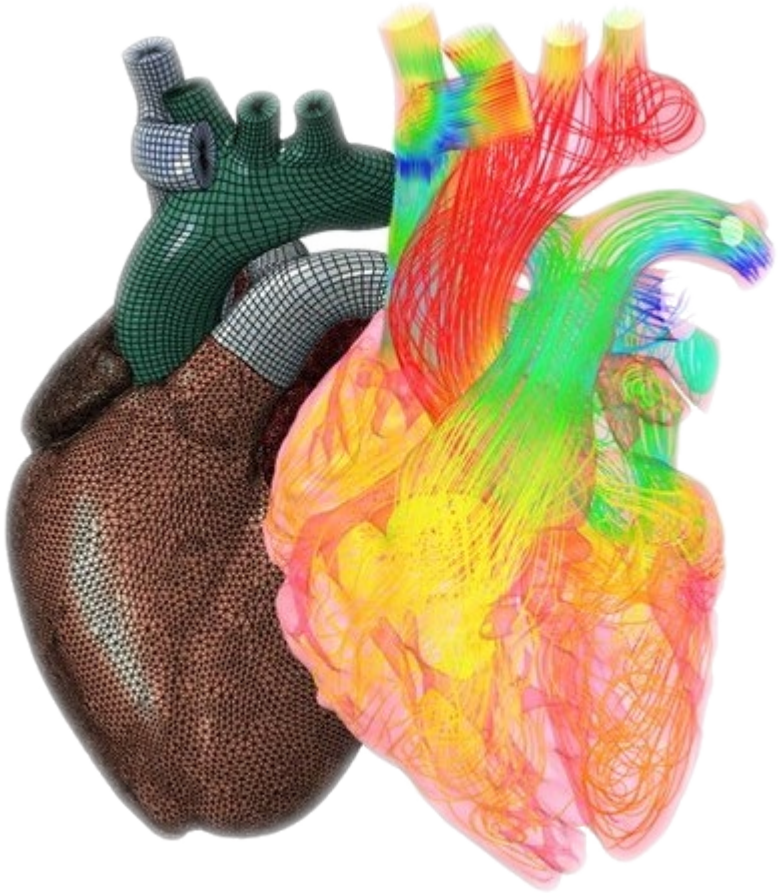
CFD today – COVID-19, indoor infections



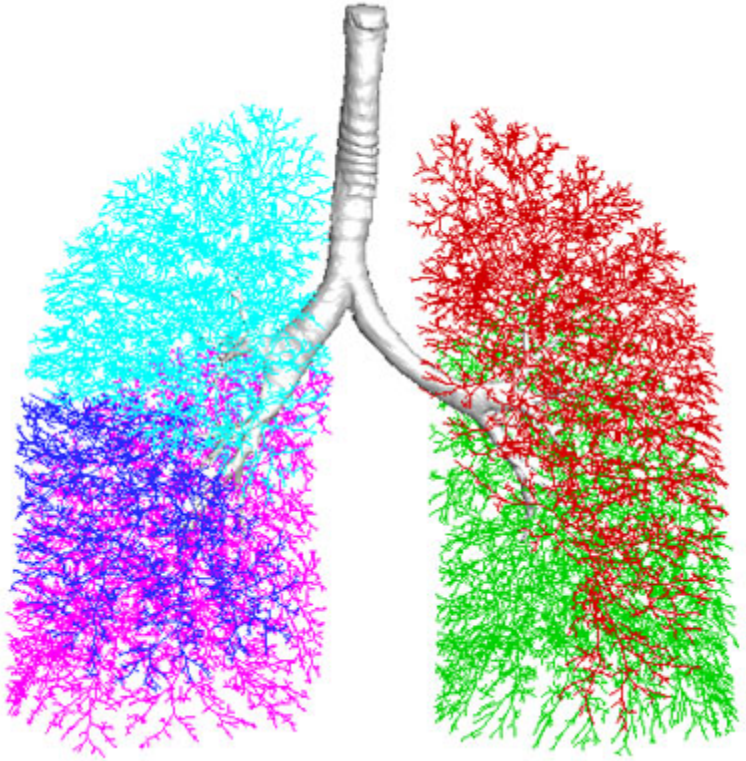


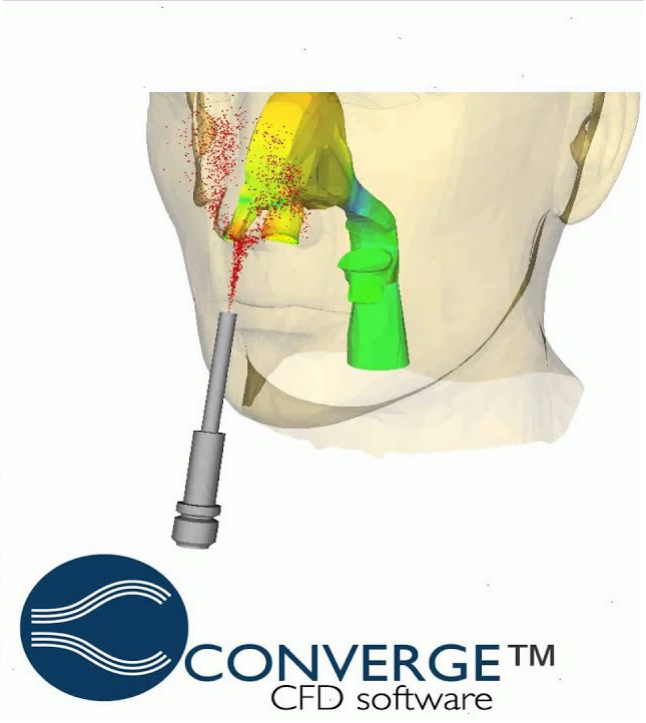


CFD today – in your body



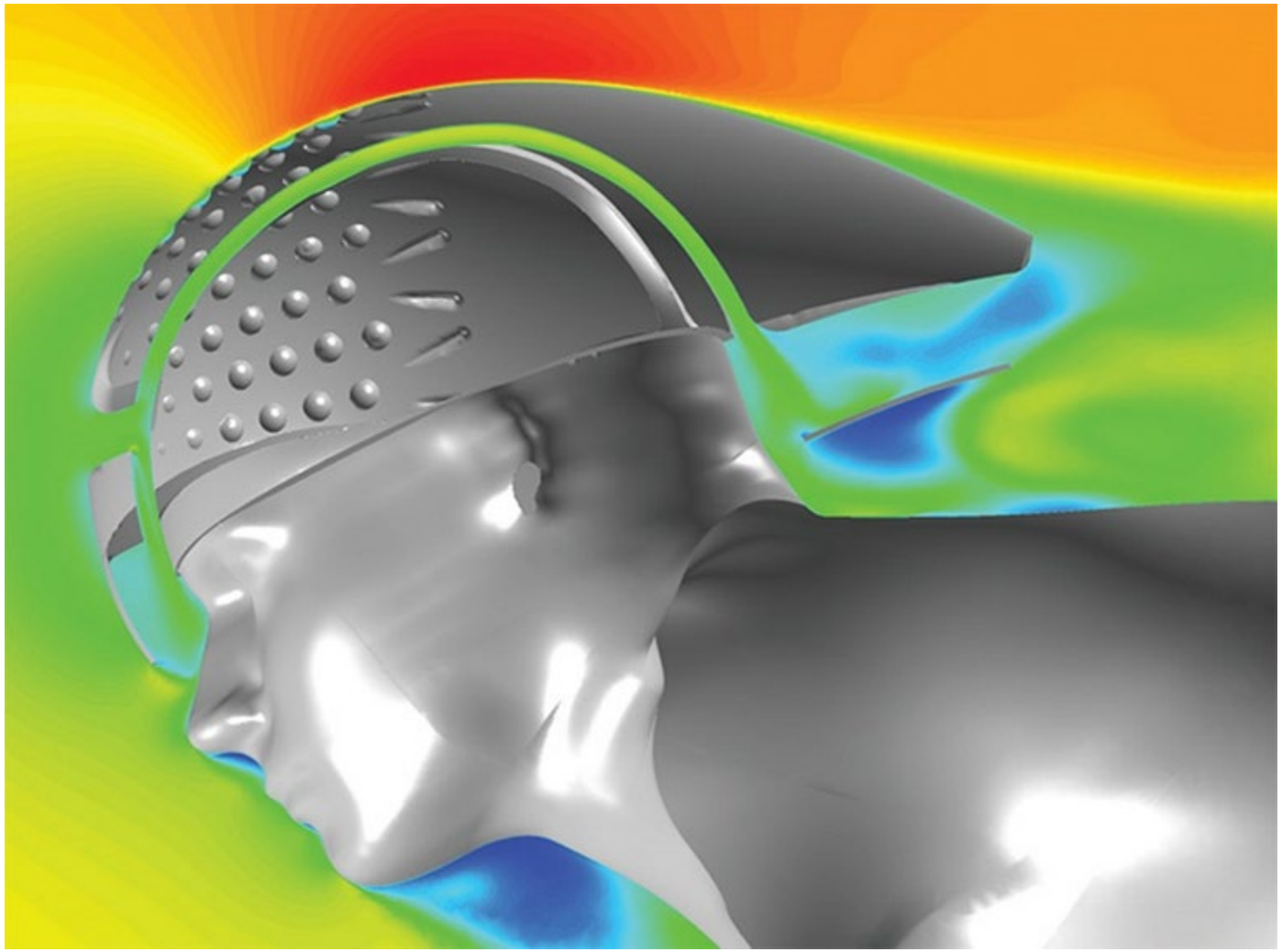
https://flowvisioncfd.com/en/flowvision-applications/cardiac-simulations



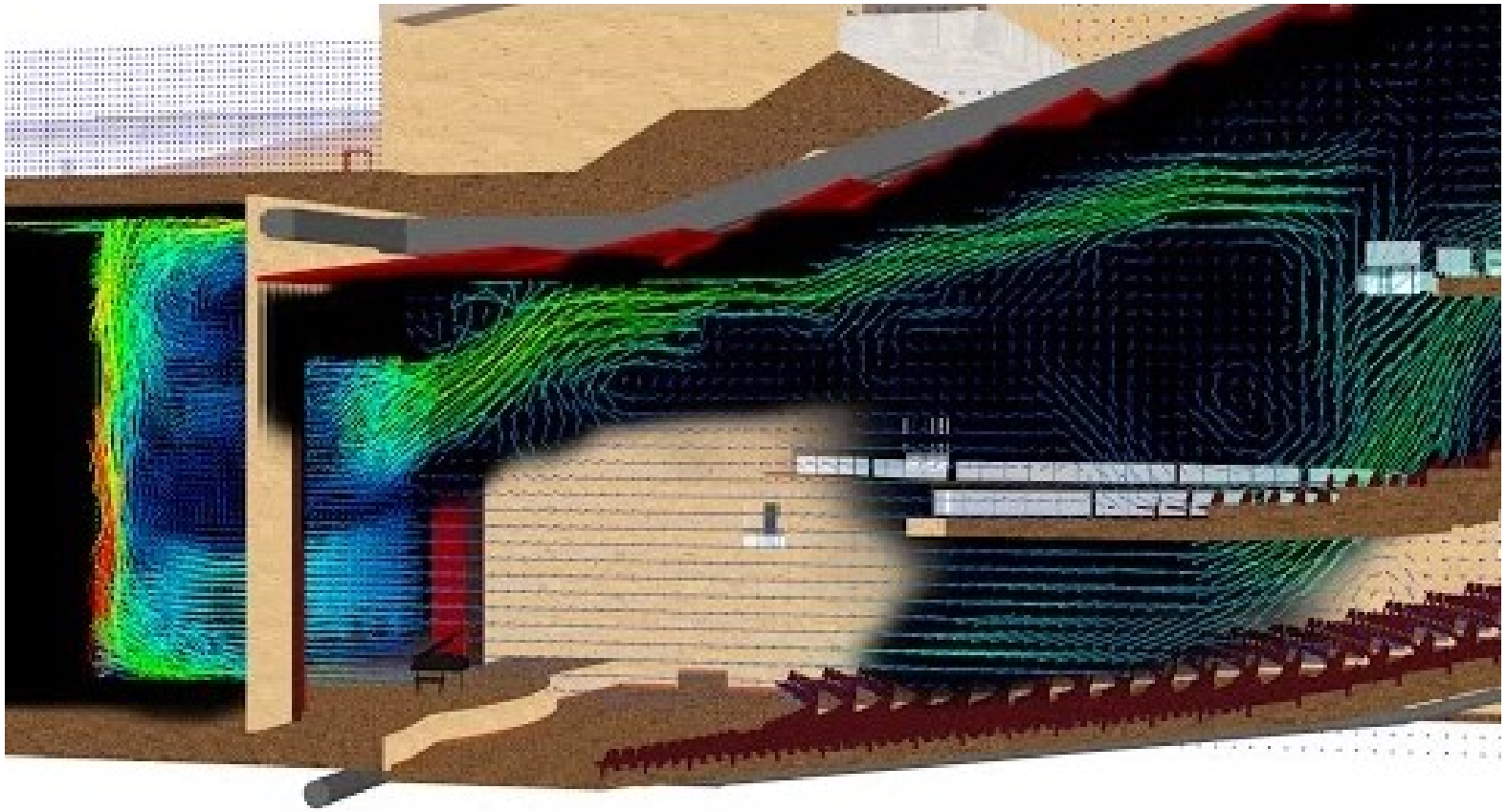


https://www.engineeringexchange.com/profiles/blogs/cfd-simulations-to-save-lives

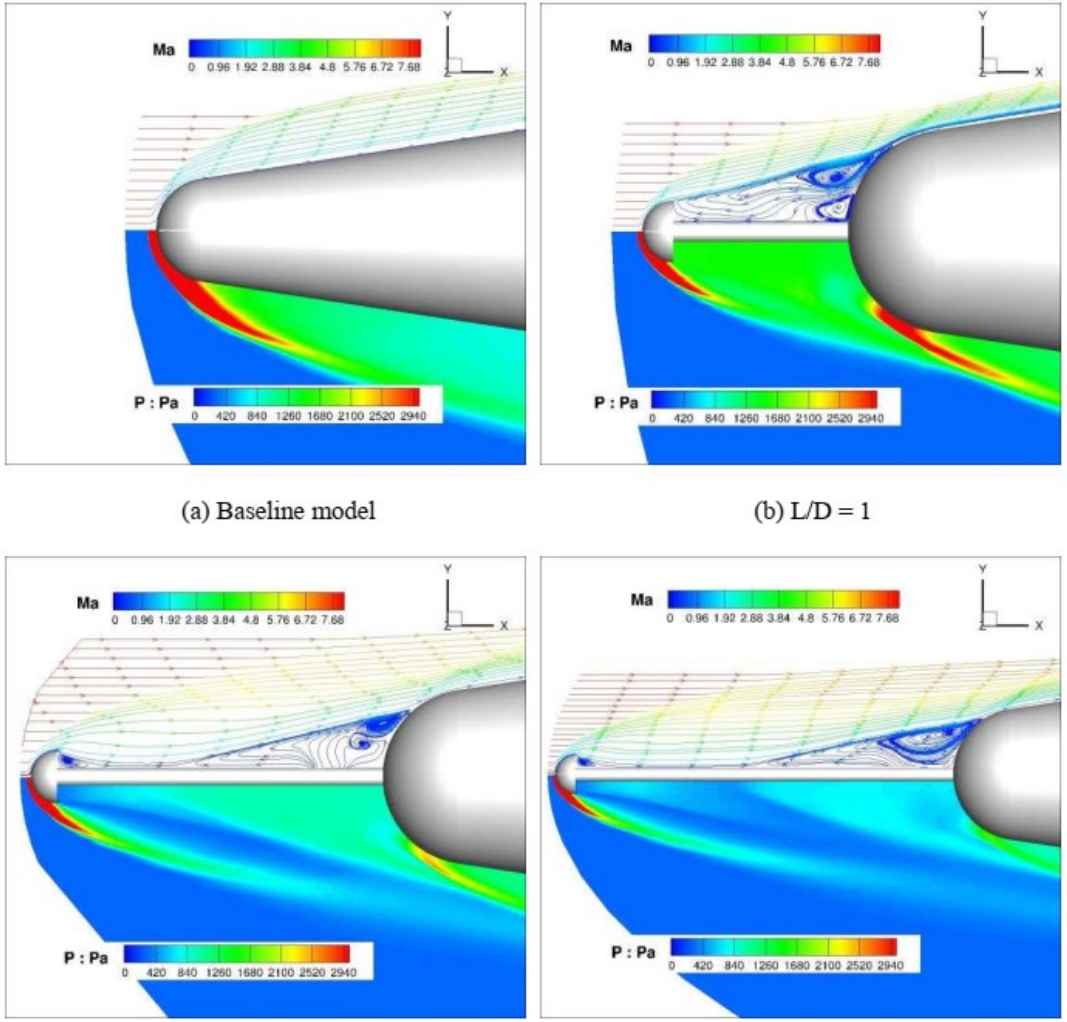
CFD today – Clothing and protection gears



CFD today – Fire and safety



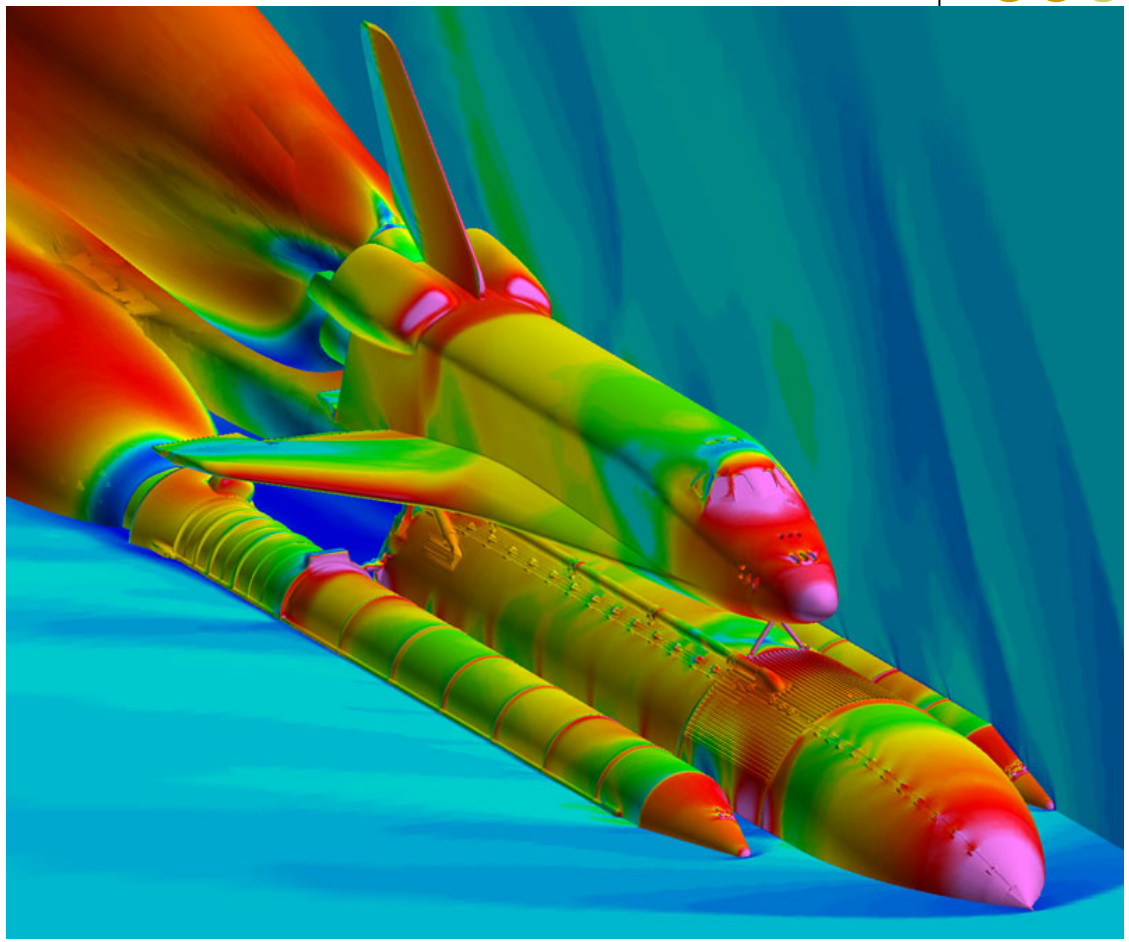
CFD today – Aerospace



(a) Baseline model (b) L/D = 1

(c) L/D =2 (d) L/D = 3

http://eprints.whiterose.ac.uk/130141/1/JSR-Revised%20manuscript.R1%28%28ID%202017-01-A33865%29.pdf

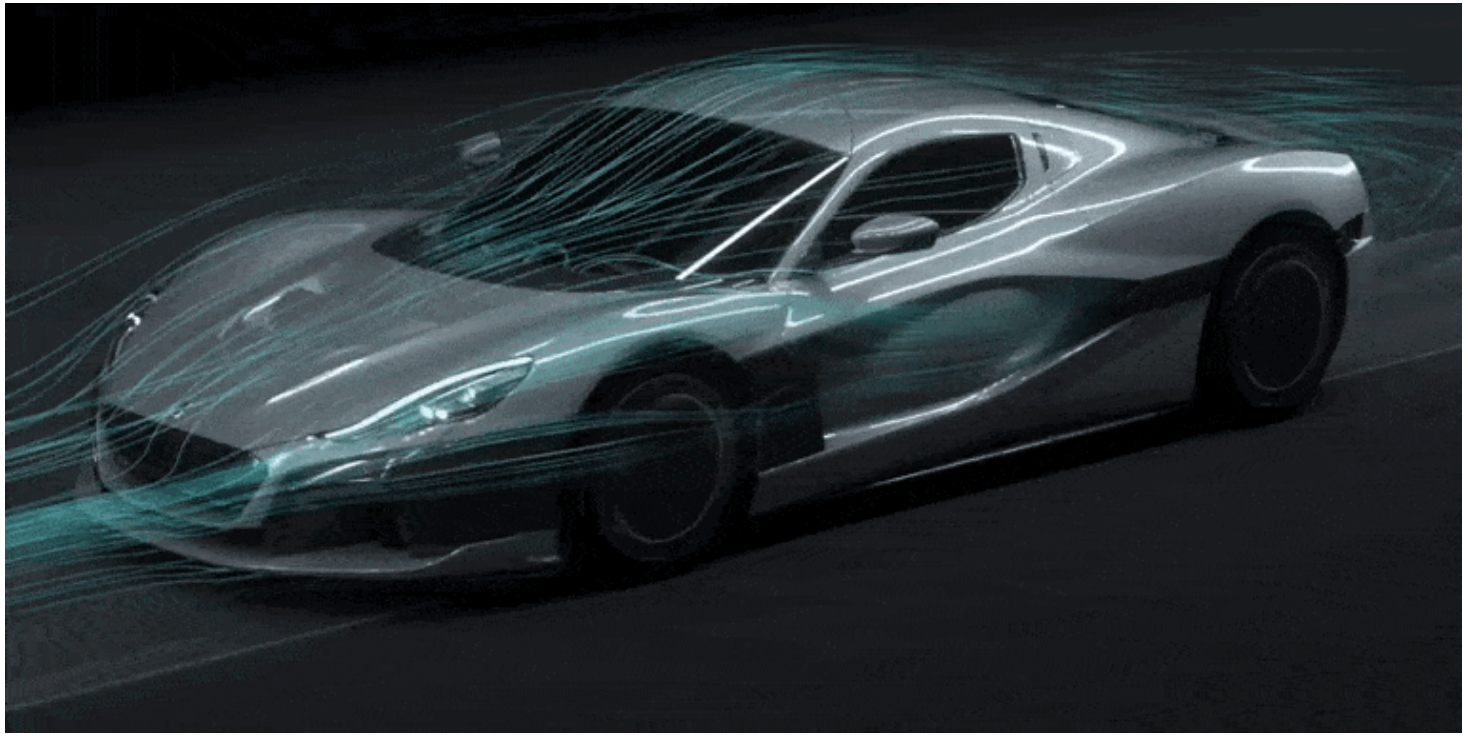


https://www.nas.nasa.gov/SC11/demos/demo20.html

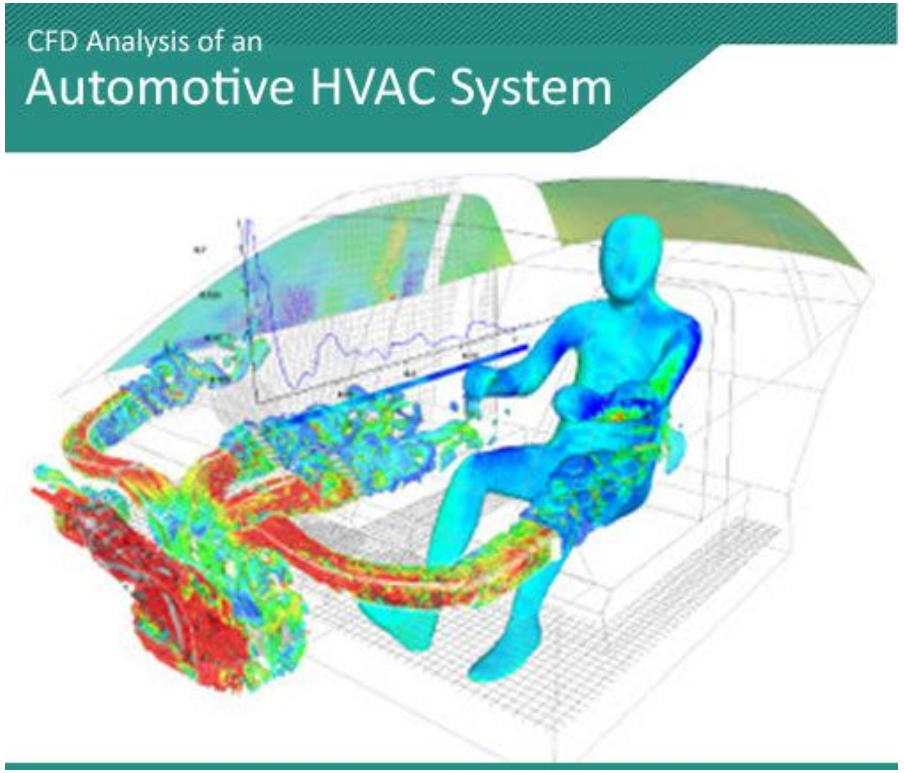
CFD today – Aerospace

Prof Antony Jameson, Stanford

CFD today – Automotive

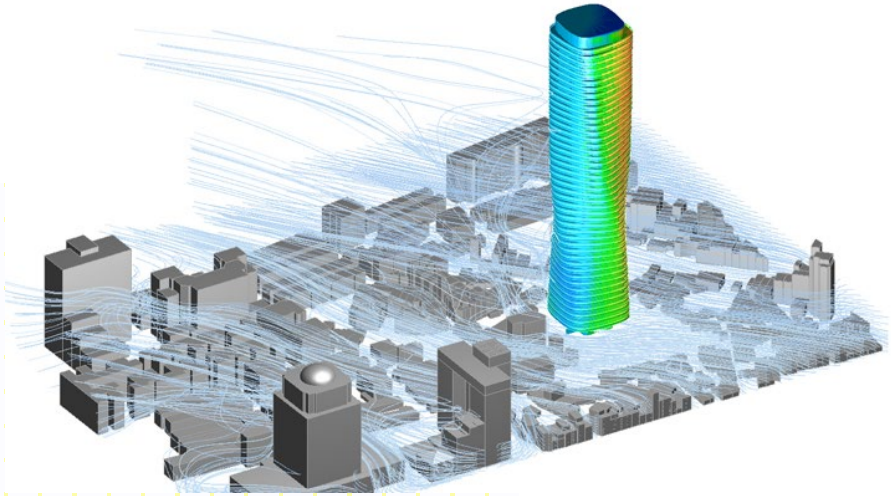


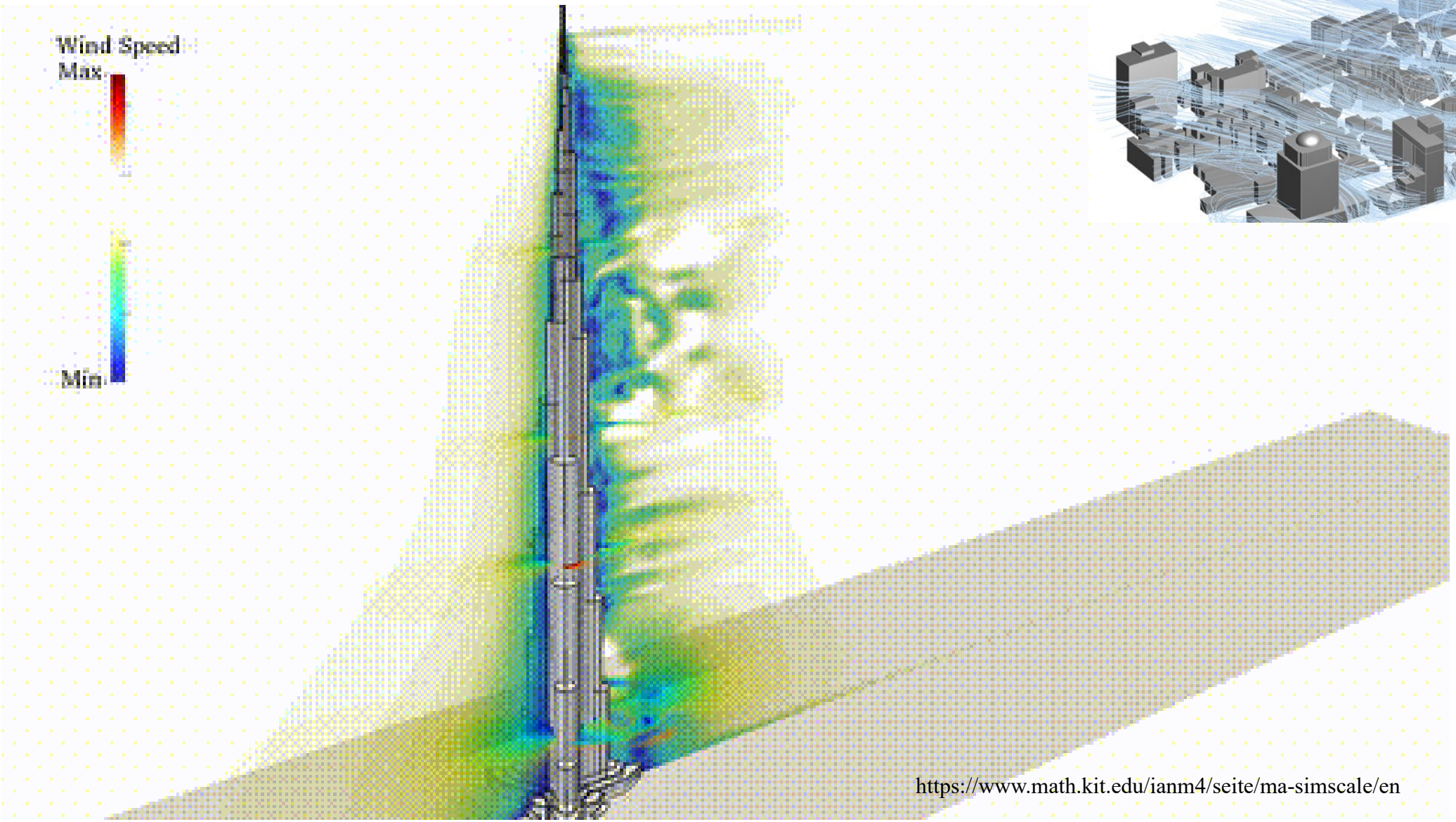
https://www.roadandtrack.com/new-cars/future-cars/a25304695/rimac-c-two-convertible-rendering/



https://www.hitechaes.com/news/cfd-analysis-of-an-automotive-hvac-system/

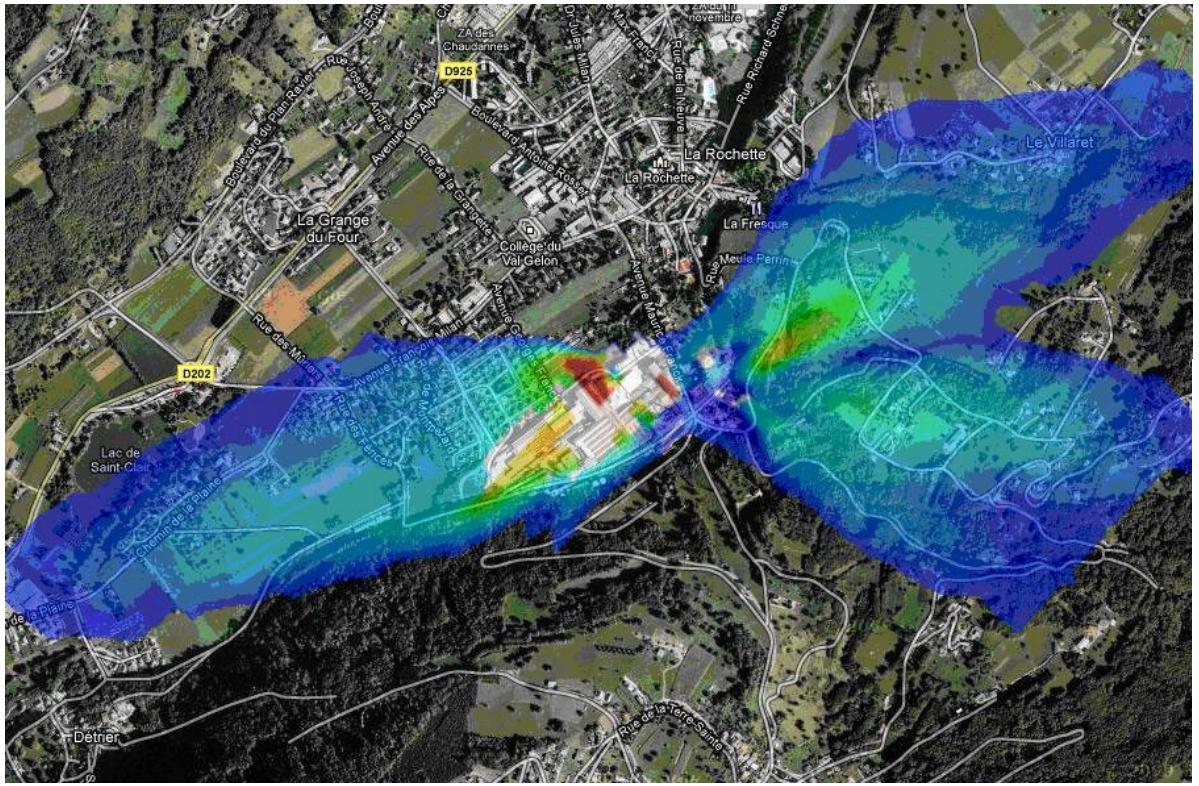
CFD today – Buildings





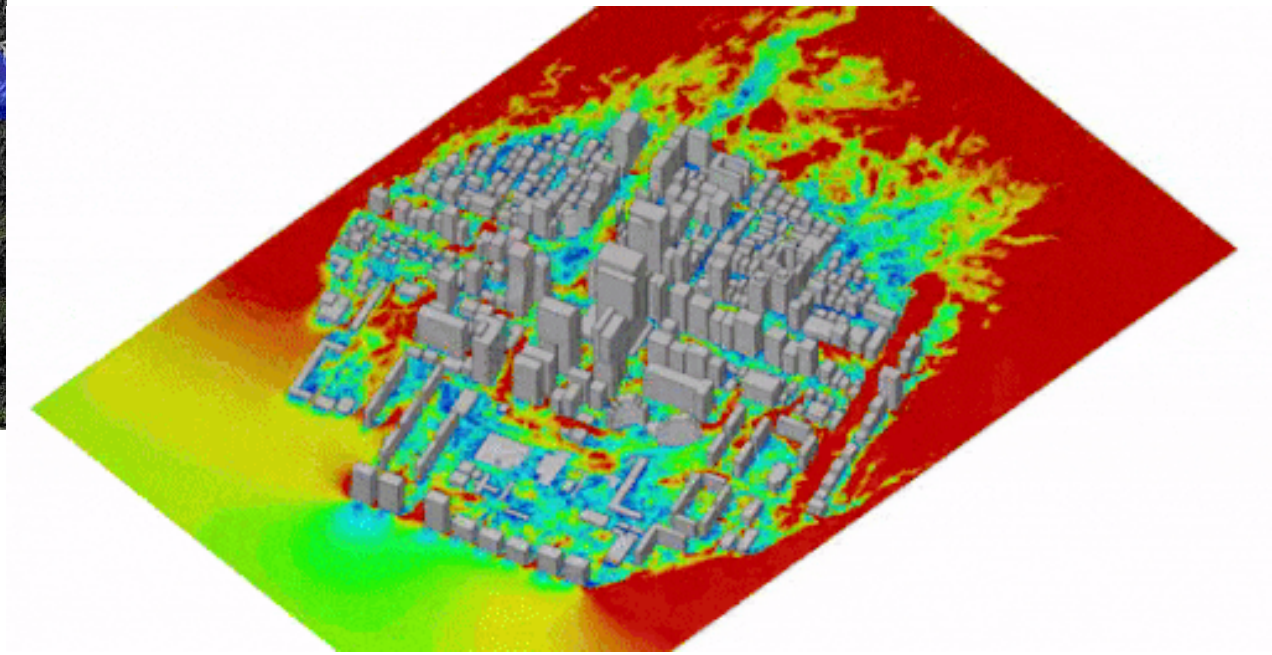
https://www.math.kit.edu/ianm4/seite/ma-simscale/en

CFD today – Environment



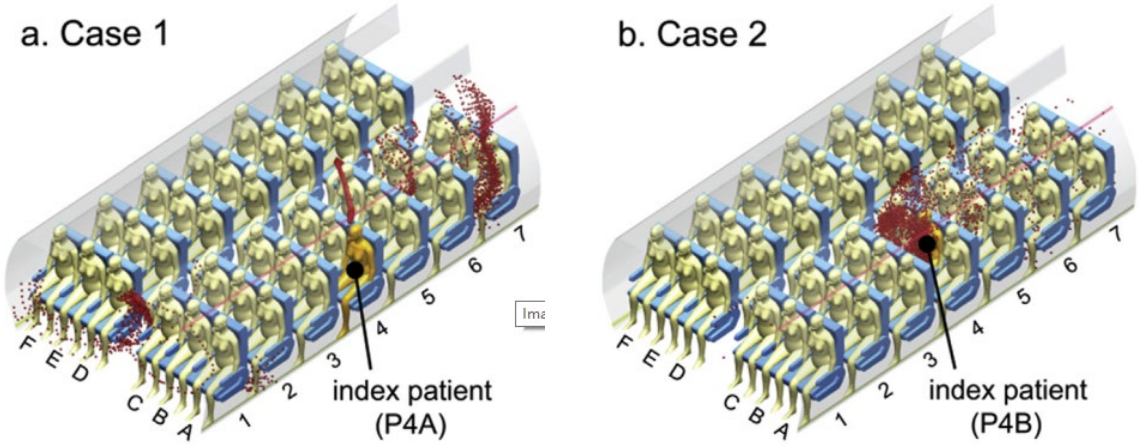
Pollutant dispersion

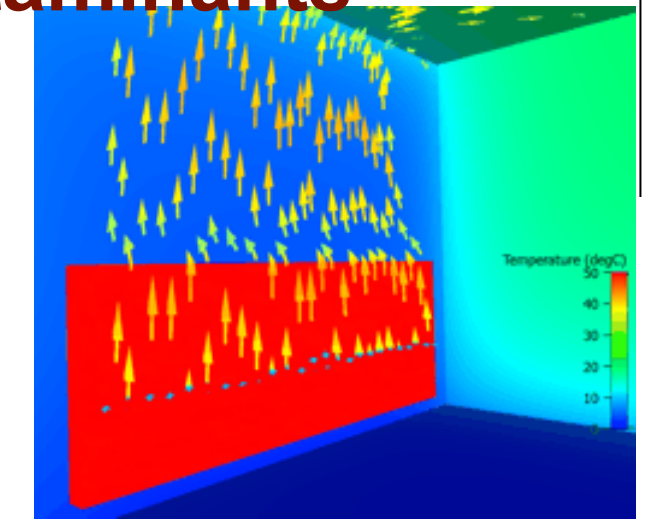
Urban design

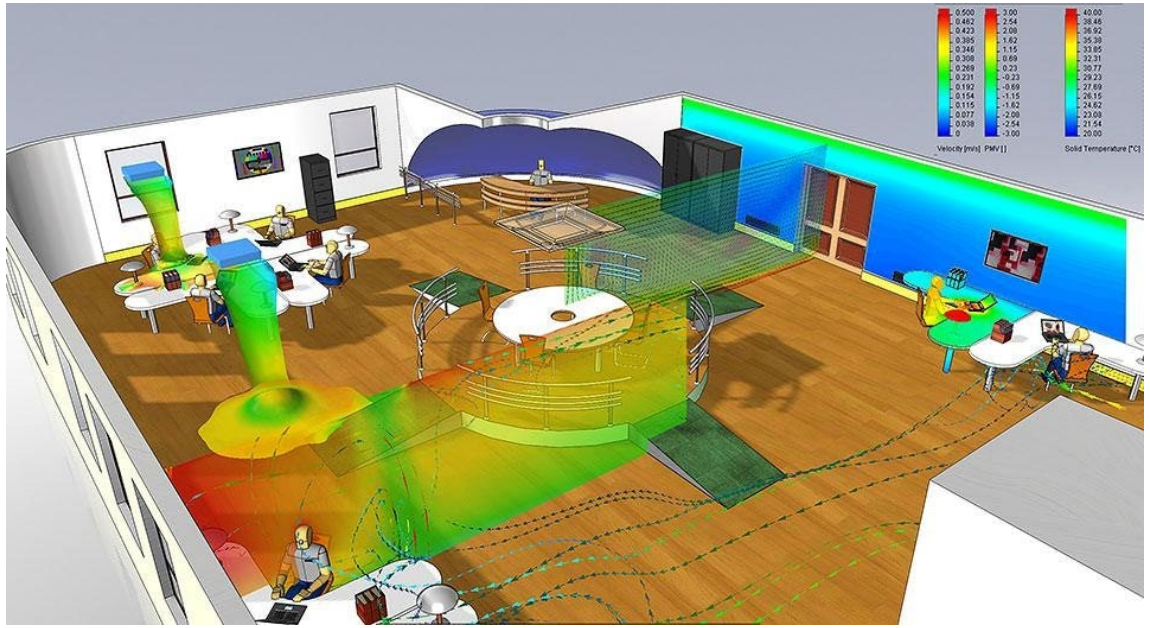


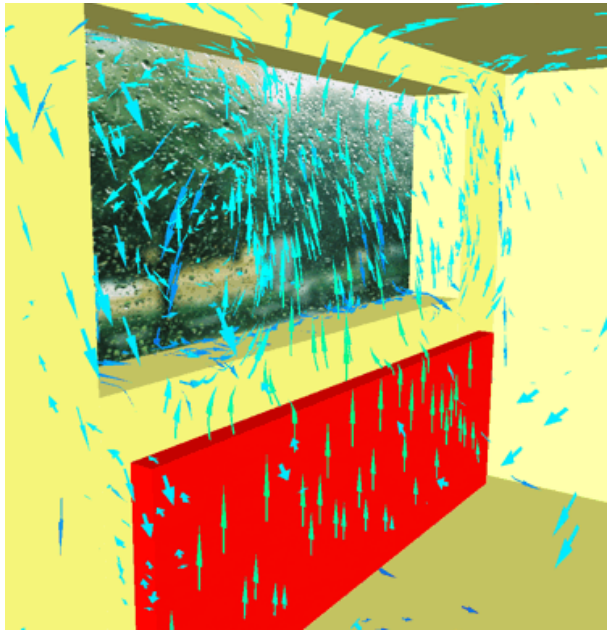
https://www.simscale.com/wind-engineering/pedestrian-wind-comfort-wind-modelling/

CFD today – HVAC and indoor contaminants







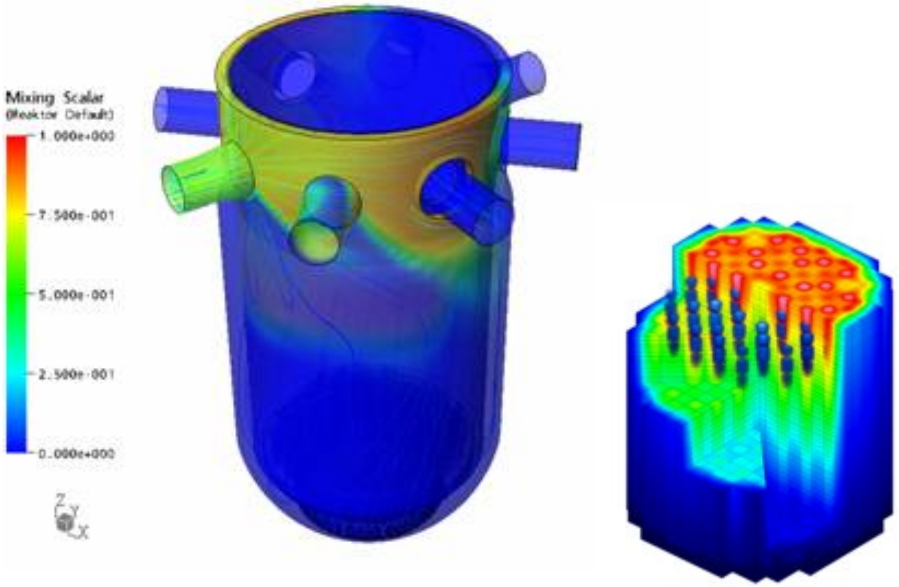


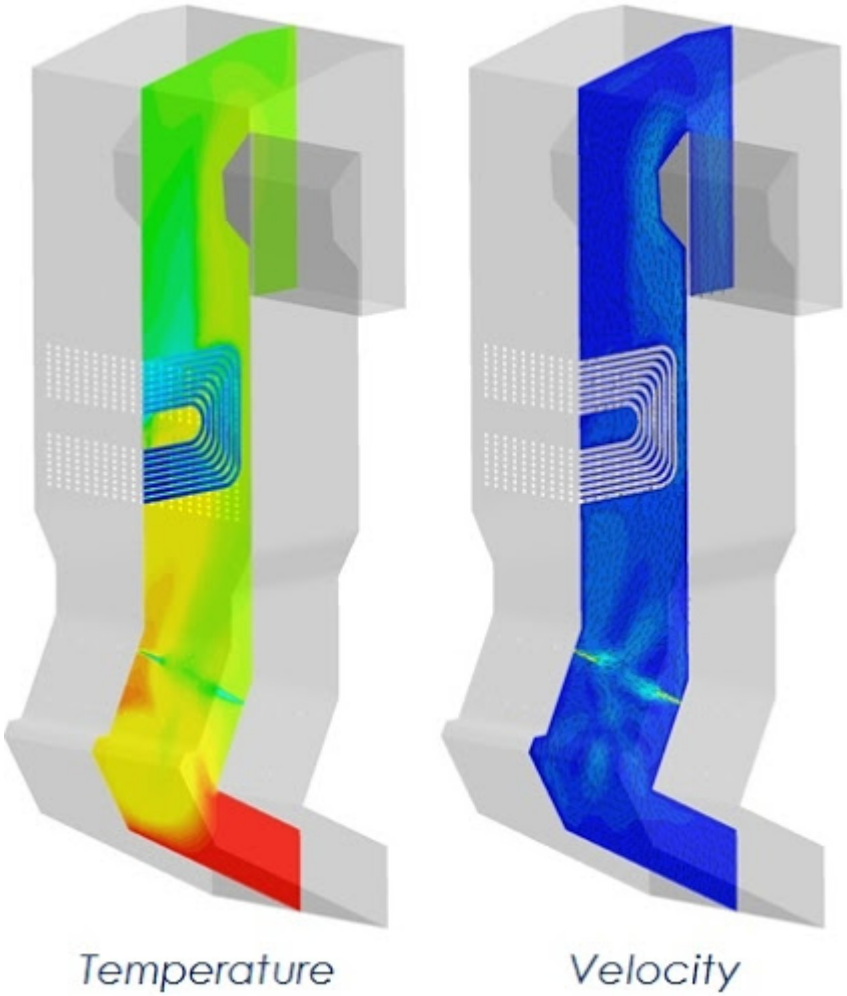
https://blogs.sw.siemens.com/simulating-the-real-world/2011/01/31/floefd-hvac-module-taking-built-environment-cfd-simulation-to-the-next-level/

https://blogs.sw.siemens.com/simulating-the-real-world/2012/07/12/wheres-the-best-place-to-put-a-radiator-in-a-room-part-1-such-things-are-important/

CFD today – Energy

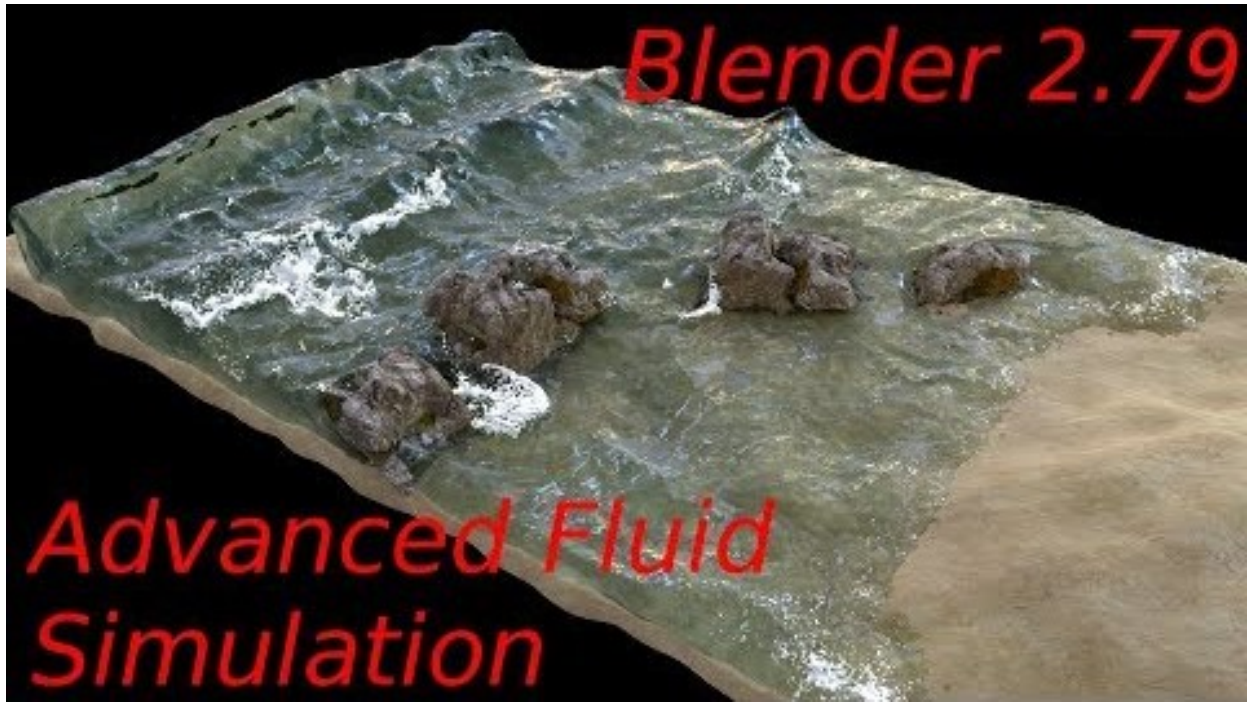
http://www.pretechnologies.com/sectors/renewable-energies/wind-power



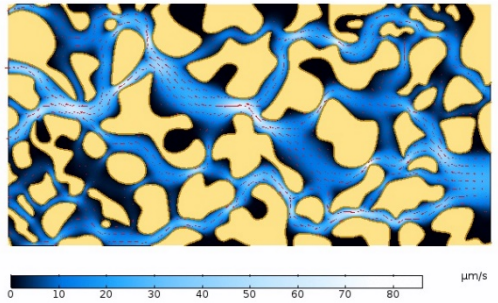


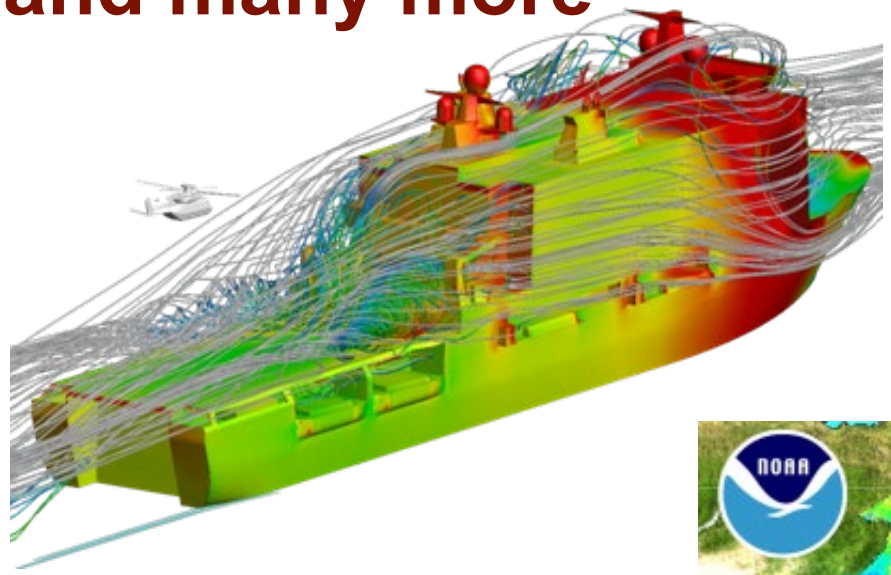
http://www.pretechnologies.com/sectors/waste-to-energy

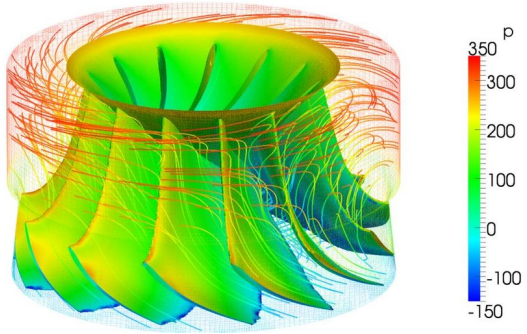
CFD today – Movies and games

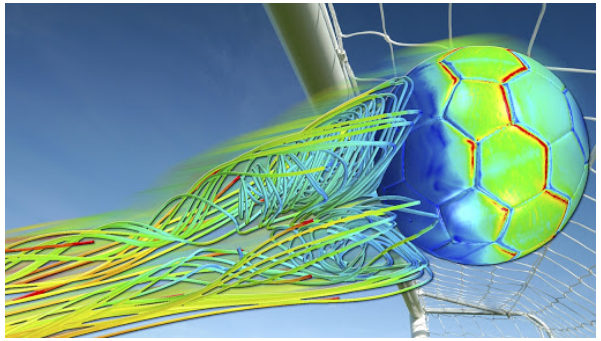


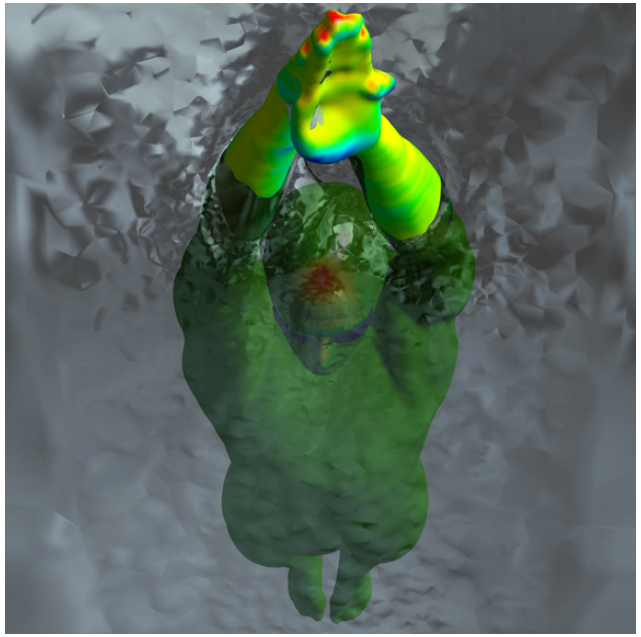
CFD today – and many more

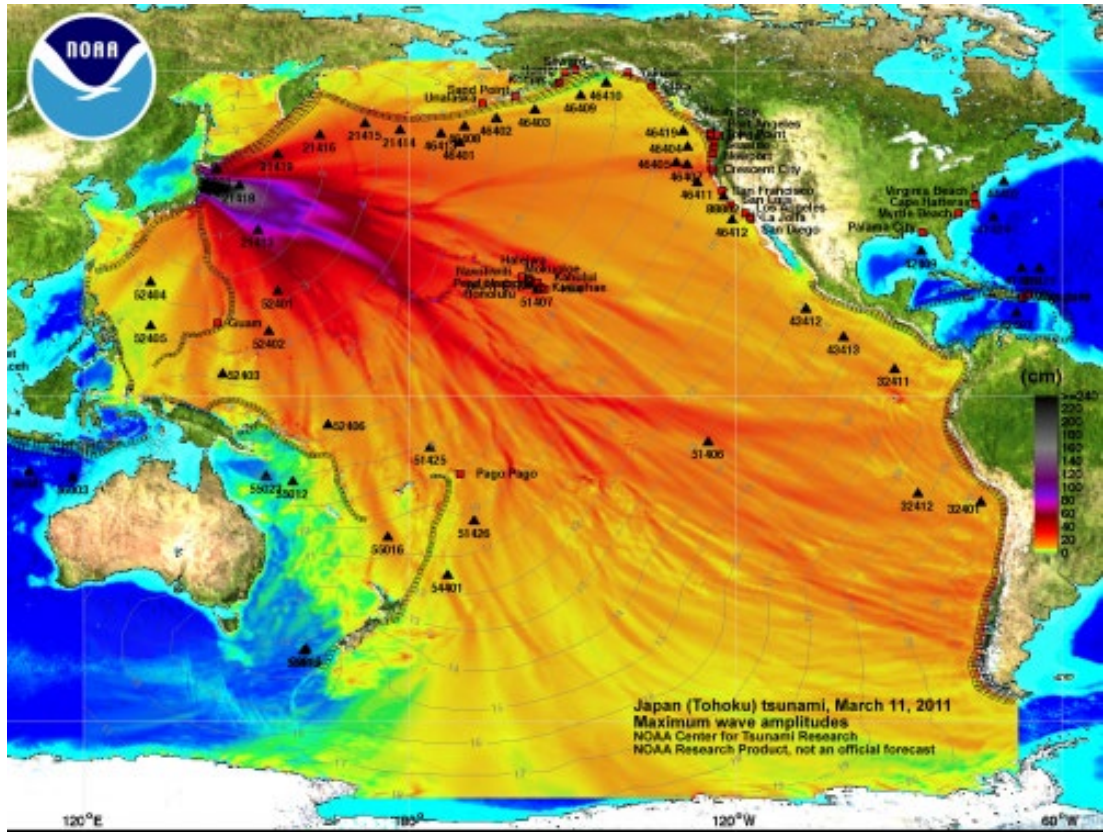




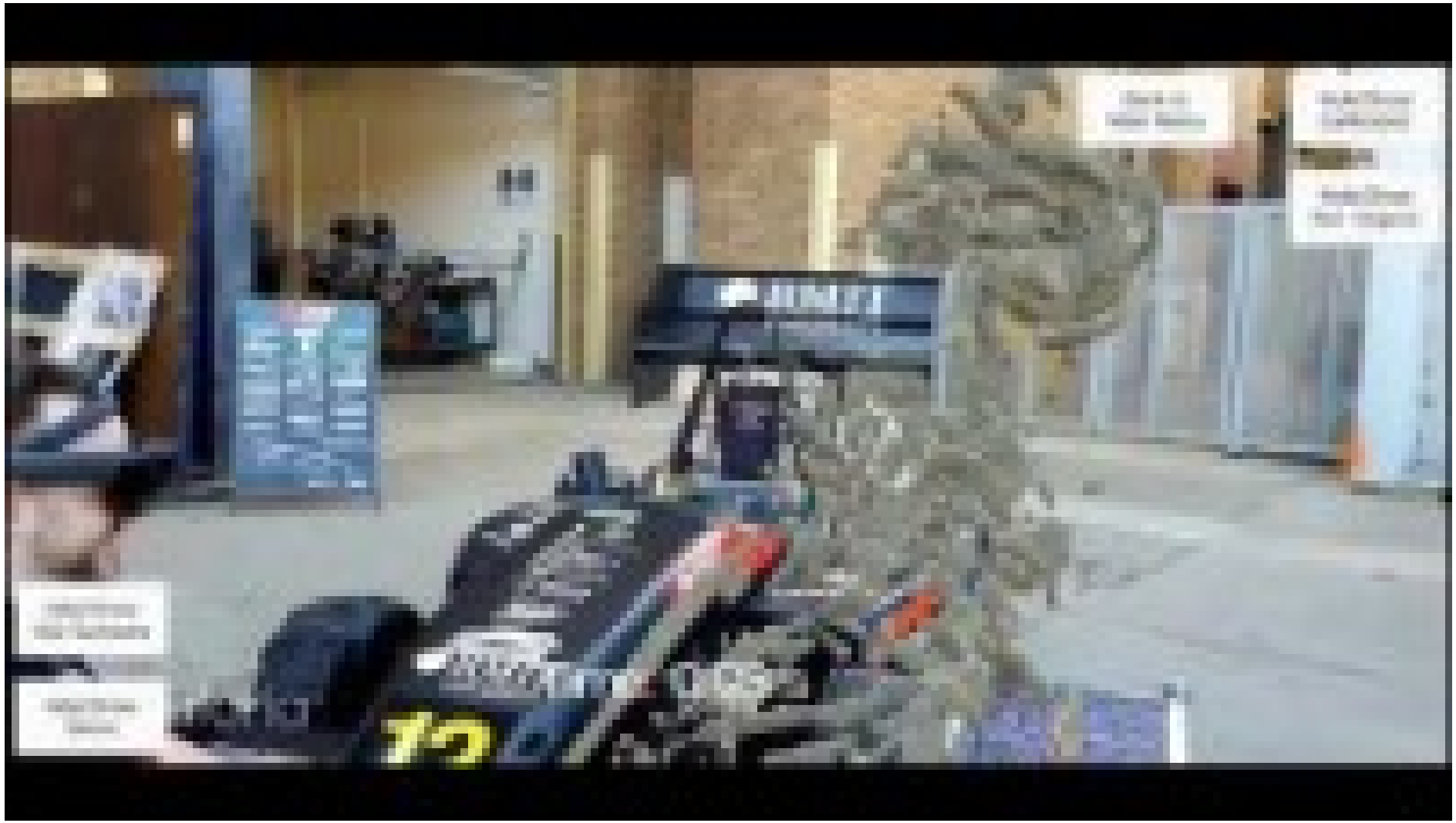




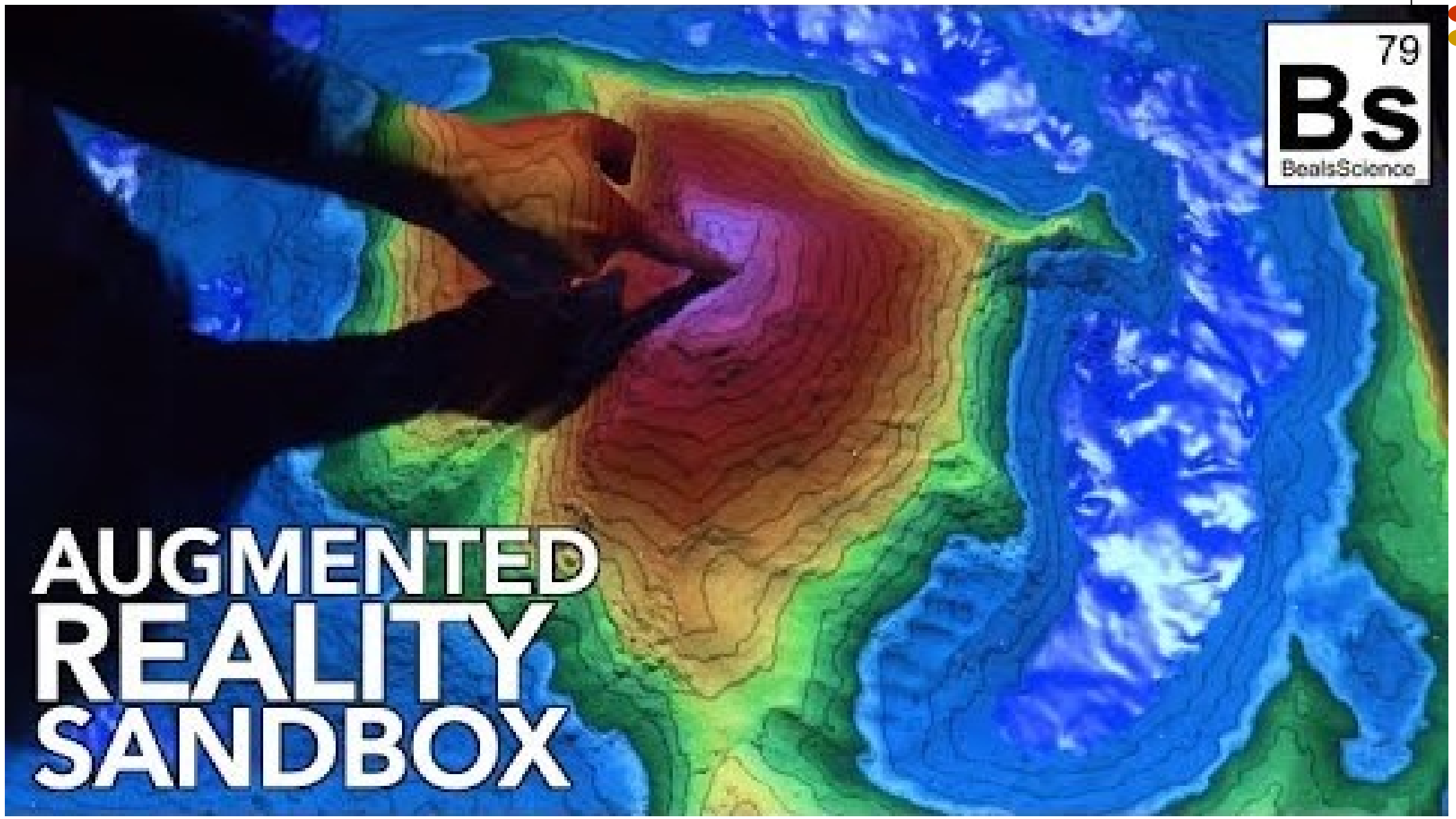




CFD in future – CFD-based AI/AR/VR



CFD in future – CFD-based AI/AR/VR



CFD – THE GOOD AND BAD

The good

- Cost-efficient
- Information-rich
- Best for hazardous and dangerous circumstances
- Virtual design/optimisation

The bad

- Errors and uncertainty
- Lack of standardization
- Many people don’t trust CFD
  - CFD is art, but not technology
  - Color For Directors / Colorful Fanny Decorations / Colorful Faulty Dynamics
- Remains experimental analysis
- Time consuming, sometimes

CFD modellers

"We complete this upbeat section by wondering whether the next constraint to the further spread of CFD amongst the industrial community could be a **scarcity of suitably trained personnel** instead of availability and/or cost of hardware and software" – H.K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics – the Finite Volume Method*, 2nd Ed, 2007.

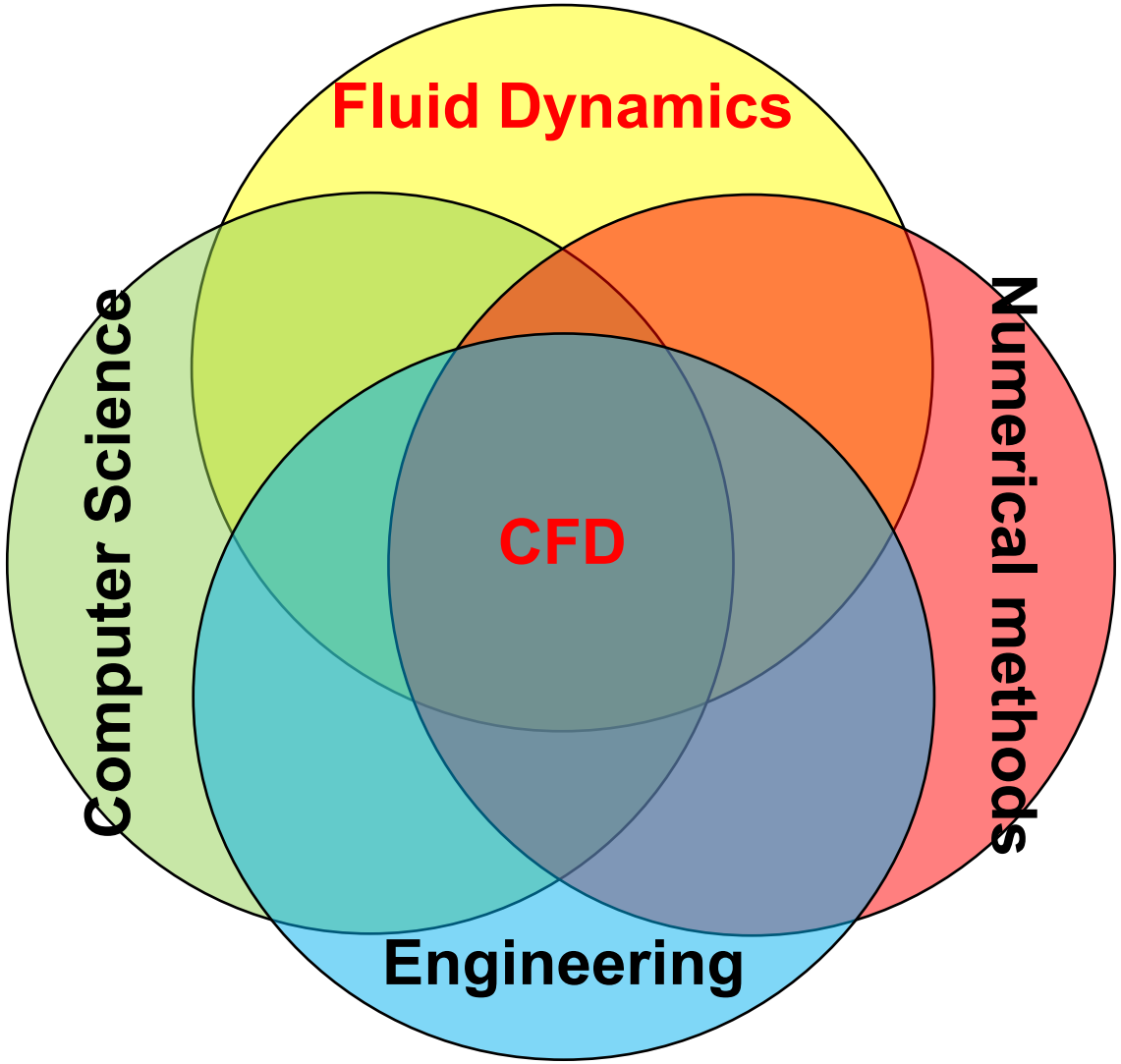
THE ECFD COURSE

- to train future CFD modellers

Pre-requisites

- **Basic physics** – Mass conservation, Newton's 2nd low, energy conservation, etc
- **Partial differential equations** – gradient, divergence, curl, etc
- **Heat transfer** – thermal conduction, convection, radiation
- **Engineering Fluid Mechanics** – viscosity, shear/normal stress, non-slip condition, Bernoulli equation, etc
- **3D CAD**
- **Computer skills**

The ECFD course



In this course, we will explore Fluid Dynamics fundamentals and basic CFD principles with commercial packages, giving you the basis to apply CFD analysis to real engineering problems.

Intended Learning Outcomes (ILOs)

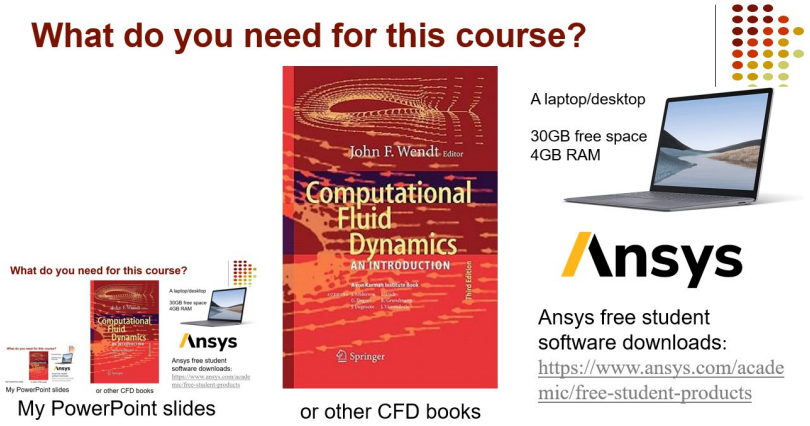
Upon successfully completing the course, you should be able to

- Recite and explain the governing equations
- Be familiar with popular solution techniques
- Develop a simple CFD model
- Have established a working vocabulary in the discipline
- Have some insight into the philosophy/power/constraints of CFD and be prepared for more sophisticated study and research

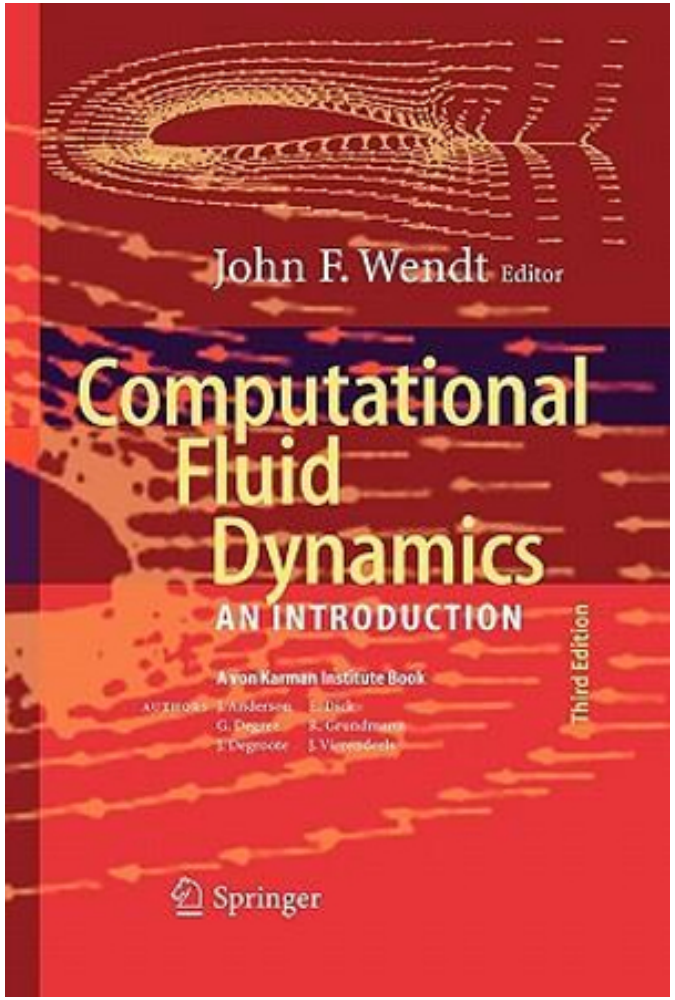
Course outline – a project-based learning journey

Lectures	Computer Labs
Module 1 – CFD intro (W1)	
	Module 2 – CFD workflow (W2)
Module 3 – Conservation equations (W3, 4)	
Module 4 – Turbulence and boundary layer (W5, 6)	
	Module 5 – Geometry and mesh (W7)
Module 6 – Solving the equations (W8)	
Module 7 – Boundary conditions (W9)	
	Module 8 – Model setup and solver (W10, 11)
	Module 9 – Post-processing (W12, 13)
Module 10 – Best practice guide (W14)	
Module 11 – Advanced topics (W15)	

What do you need for this course?



My PowerPoint slides



or other books your choice

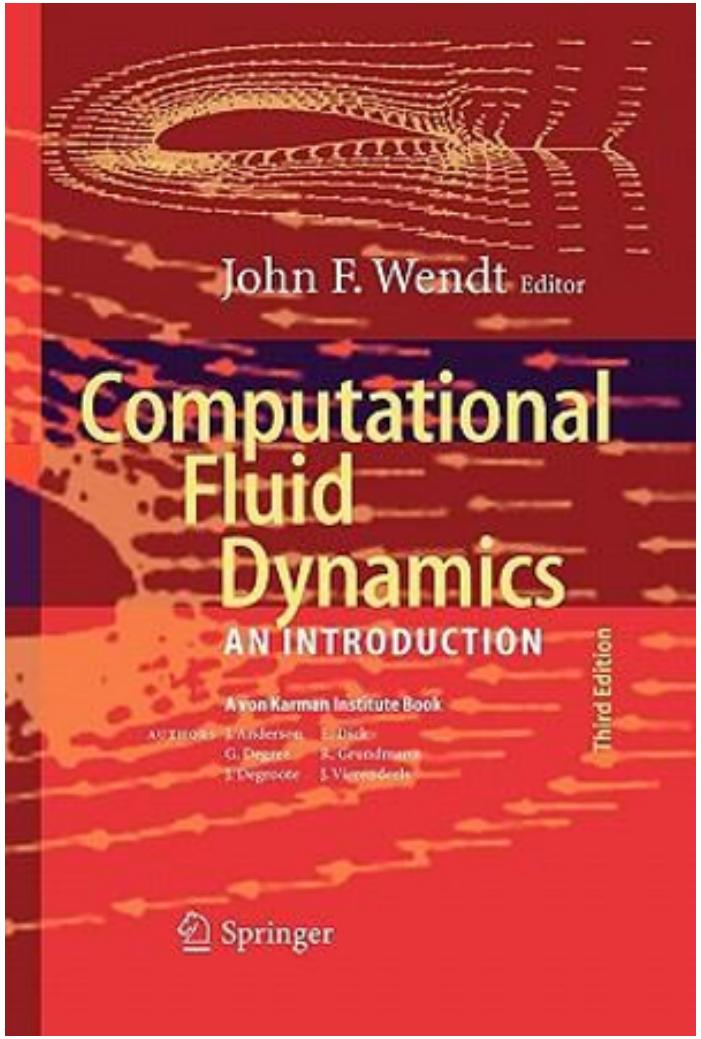
A laptop/desktop

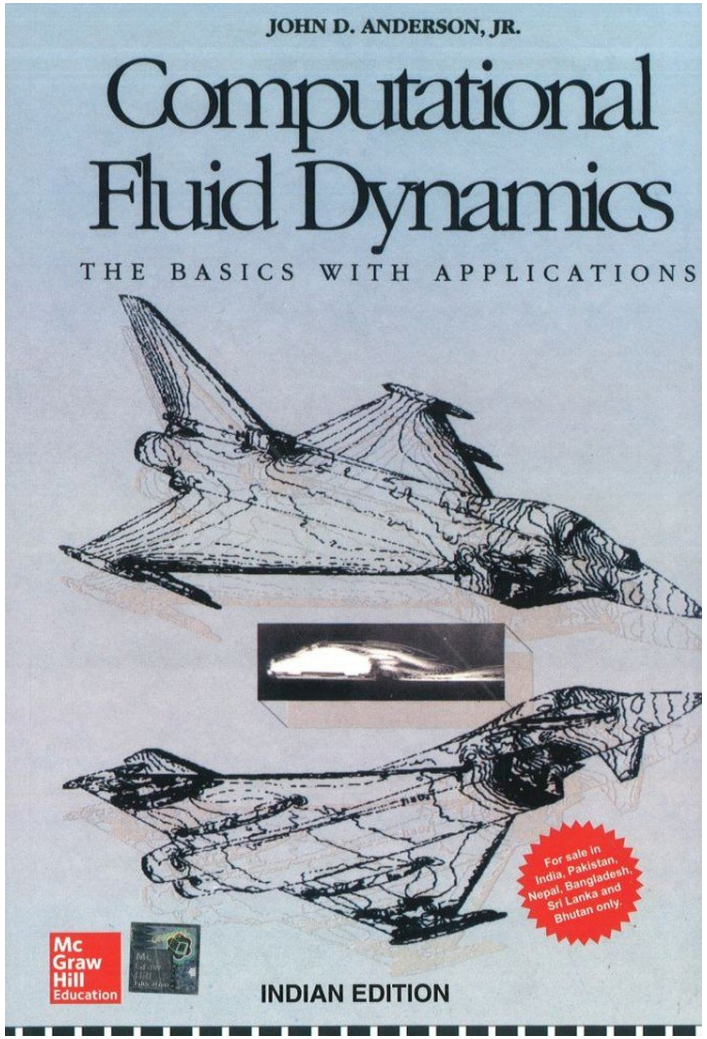
30GB free space
4GB RAM

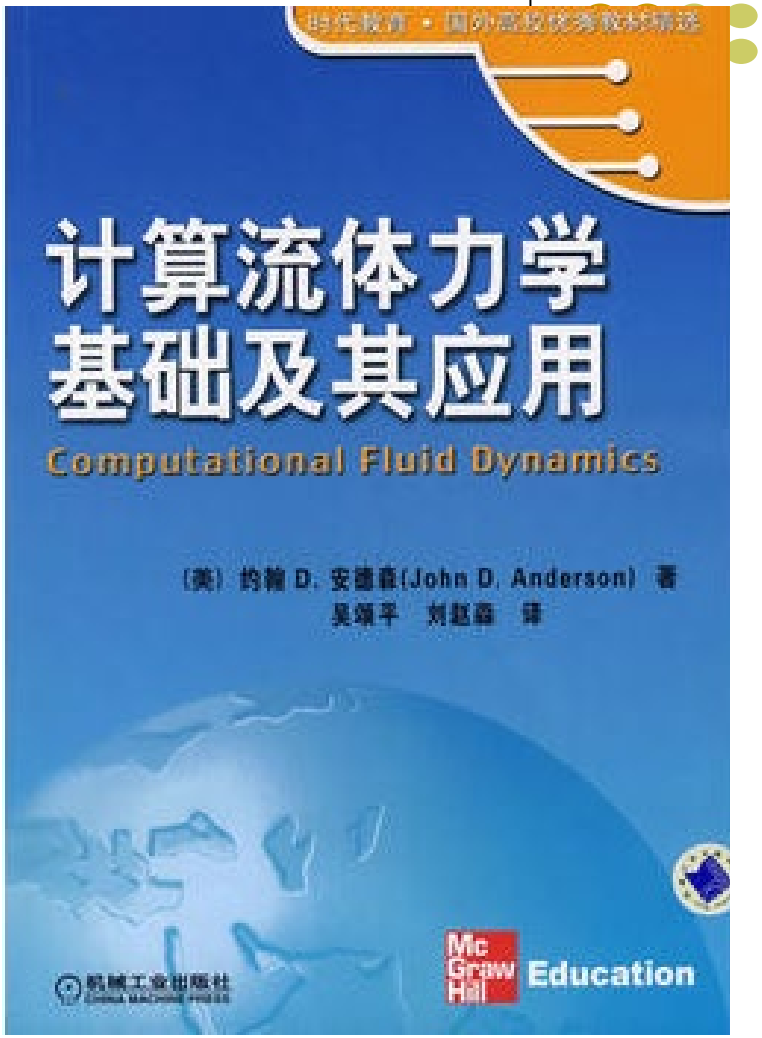
Ansys

Ansys free student software downloads:
https://www.ansys.com/academic/free-student-products

More about the textbook







How will I assess your learning?
-- to discuss

Exam 40%

- Fluid dynamics theory 25%
- Numerical procedures 15%

Project 30%

- A CFD analysis
- A short report

Assignment and Engagement 30%

- Attendance 10%
- Assignment 20%

The standard project

This is your project – TBA

Your customised project

You are encouraged to propose your own project – it can be an individual or group project (max 3 people)

- Choose your topic of interest
- Come to talk with me
- For group project – the work will be assessed as a single work, each team member is required to submit a self-reflection on your personal contribution

In this course

- **Project-based learning** – Assignments given on the first day of each module
- **You are the knowledge constructor, I am your learning facilitator** – To facilitate, I mean I make your learning possible, but I don't do it for you.
- **Your learning, your ownership & your responsibility** – You have options and flexibility about what and how to learn. But if you are not applying yourself to your learning, you need to take responsibility for the consequences.
- **Teamwork and social learning** – I don't discourage independent work but am a strong advocate for teamwork and social learning. Never expect you can get all the answers for your assignment from the lectures.

How can you succeed?

- Self-motivated learning
- Making use of multiple resources
- Seriously completing the assignments and projects
- Using instructions and rubrics for assignments and project
- Engaging in discussions and learning activities

Other resources

- Elseviser
- Google/Baidu
- Wiki
- Forums
  - http://www.cfd-online.com/
  - http://www.e-fluids.com/
  - http://www.nas.nasa.gov/SC08/images.html
- Media
  - CFD之道 https://mp.weixin.qq.com/s/843e6ZL7qS_xtiVEljShXw
  - Bilibili
  - Youtube
- Simscale
  - https://www.simscale.com/docs/validation-cases/

Reminders of today

Download and install Ansys Fluid products

https://www.ansys.com/academic/free-student-products

Visit your LMS and access your learning resources

https://xiangdong.moodlecloud.com

Q&A TIME