Chapter 1 Introduction

OUTLINE

1.1 General Remarks

1.2 Methods for Solving Fluid Dynamics Problems

1.3 The Impact of Computational Fluid Dynamics---Some Other Examples

1.4 CFD Process

1.1 General Remarks (1)
Preface

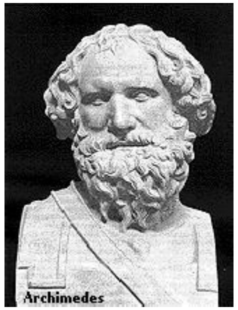
- Practice of engineering and science has been dramatically altered by the development of
  - Scientific computing
  - Mathematics of numerical analysis
  - The Internet

- Computational Fluid Dynamics is based upon the logic of applied mathematics
  - provides tools to unlock previously unsolved problems
  - is used in nearly all fields of science and engineering
    - Aerodynamics, acoustics, bio-systems, cosmology, geology, heat transfer, hydrodynamics, river hydraulics, etc…

1.1 General Remarks (2)

Fluid Mechanics-1

Faces of Fluid Mechanics : some of the greatest minds of history have tried to solve the mysteries of fluid mechanics

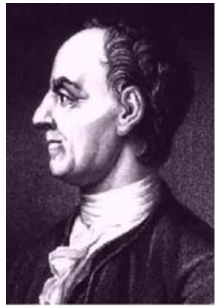


Archimedes

Da Vinci

Newton

Leibniz



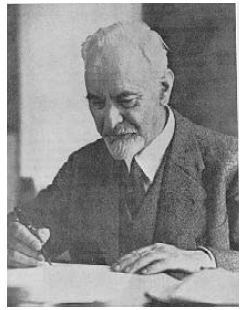
Euler

Bernoulli

Navier

Stokes

Reynolds

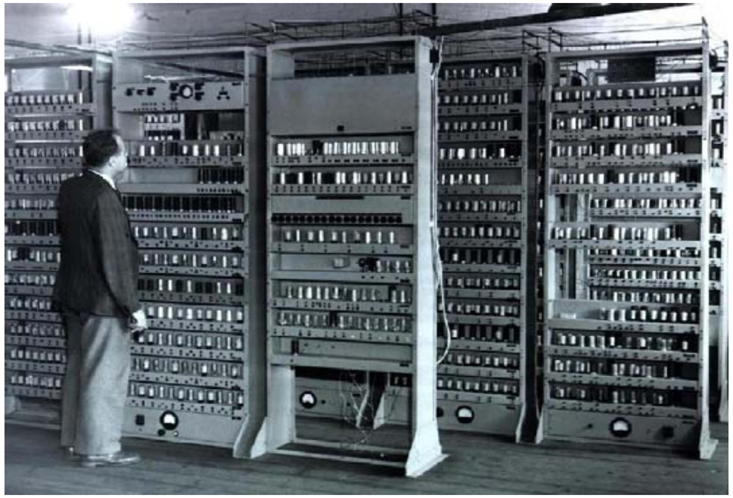


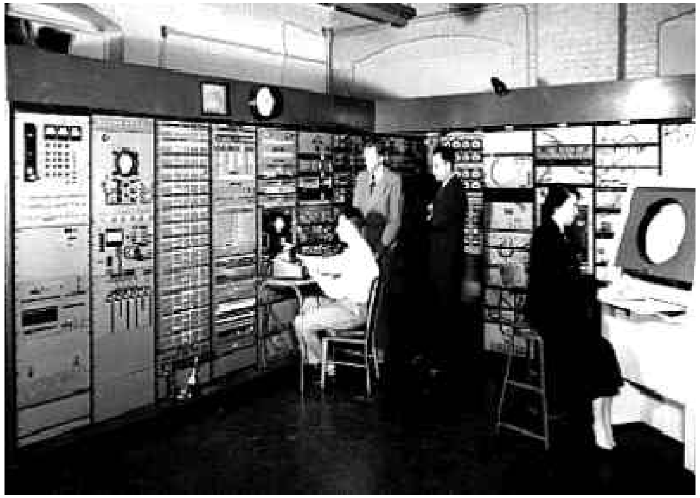
Prandtl

1.1 General Remarks (3)
Fluid Mechanics-2

- From mid-1800’s to 1960’s, research in fluid mechanics focused upon
  - Analytical methods
    - Exact solution to Navier-Stokes equations (~80 known for simple problems, e.g., laminar pipe flow)
    - Approximate methods, e.g., Ideal flow, Boundary layer theory
  - Experimental methods
    - Scale models: wind tunnels, water tunnels, towing-tanks, flumes,...
    - Measurement techniques: pitot probes; hot-wire probes; anemometers; laser-doppler velocimetry; particle-image velocimetry
    - Most man-made systems (e.g., airplane) engineered using build-and-test iteration.
- 1950’s – present : rise of computational fluid dynamics (CFD)

1.1 General Remarks (4)
History of computing

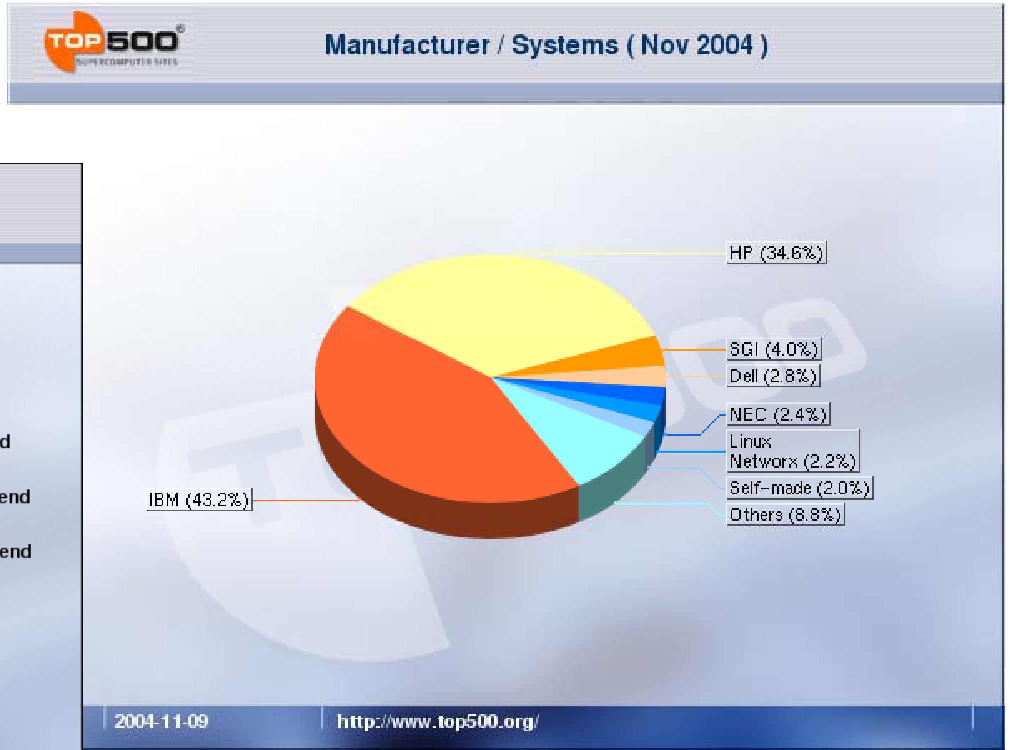


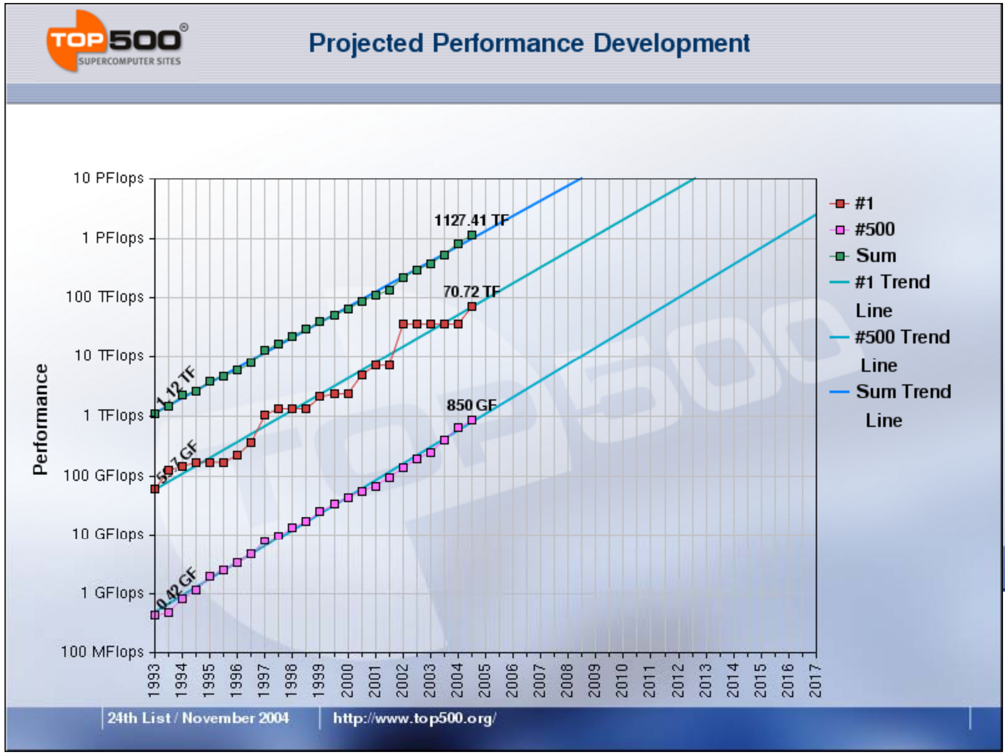


- Mastodons of computing, 1945-1960
  - Early computer engineers thought that only a few dozen computers required worldwide
  - Applications: cryptography (code breaking), ***fluid dynamics***, artillery firing tables, atomic weapons
  - ENIAC, or Electronic Numerical Integrator Analyzor and Computer, was developed by the Ballistics Research Laboratory in Maryland and was built at the University of Pennsylvania's Moore School of Electrical Engineering and completed in November 1945

1.1 General Remarks (5)
High-performance computing

- Top 500 computers in the world compiled: www.top500.org
- Computers located at major centers connected to researchers via Internet





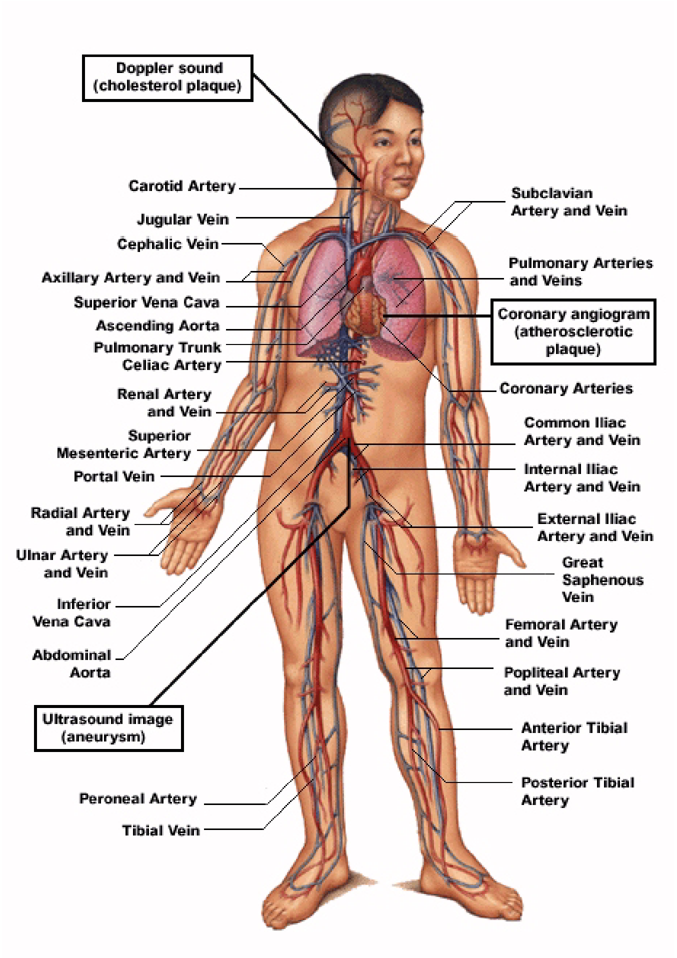
1.1 General Remarks (6)
Motivation for Studying Fluid Mechanics

- Fluid Mechanics is omnipresent
  - Aerodynamics
  - Bioengineering and biological systems
  - Energy generation
  - Geology
  - Hydraulics and Hydrology
  - Hydrodynamics
  - Meteorology
  - Ocean and Coastal Engineering
  - Water Resources
  - …numerous other examples…

1.1 General Remarks (7)
Aerodynamics

1.1 General Remarks (8)

Bioengineering



1.1 General Remarks (9)

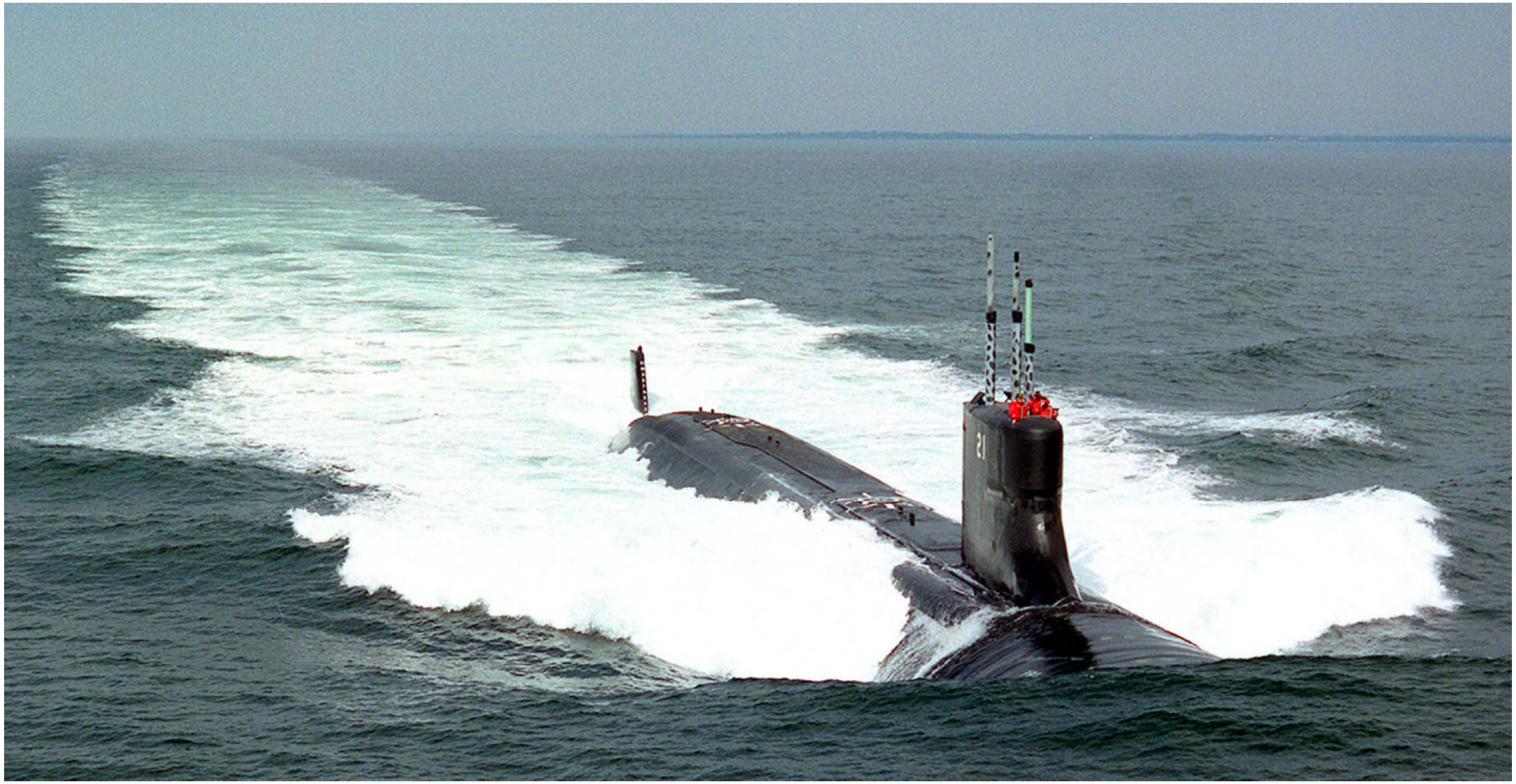
Energy generation

1.1 General Remarks (10)
Geology

1.1 General Remarks (11)
River Hydraulics

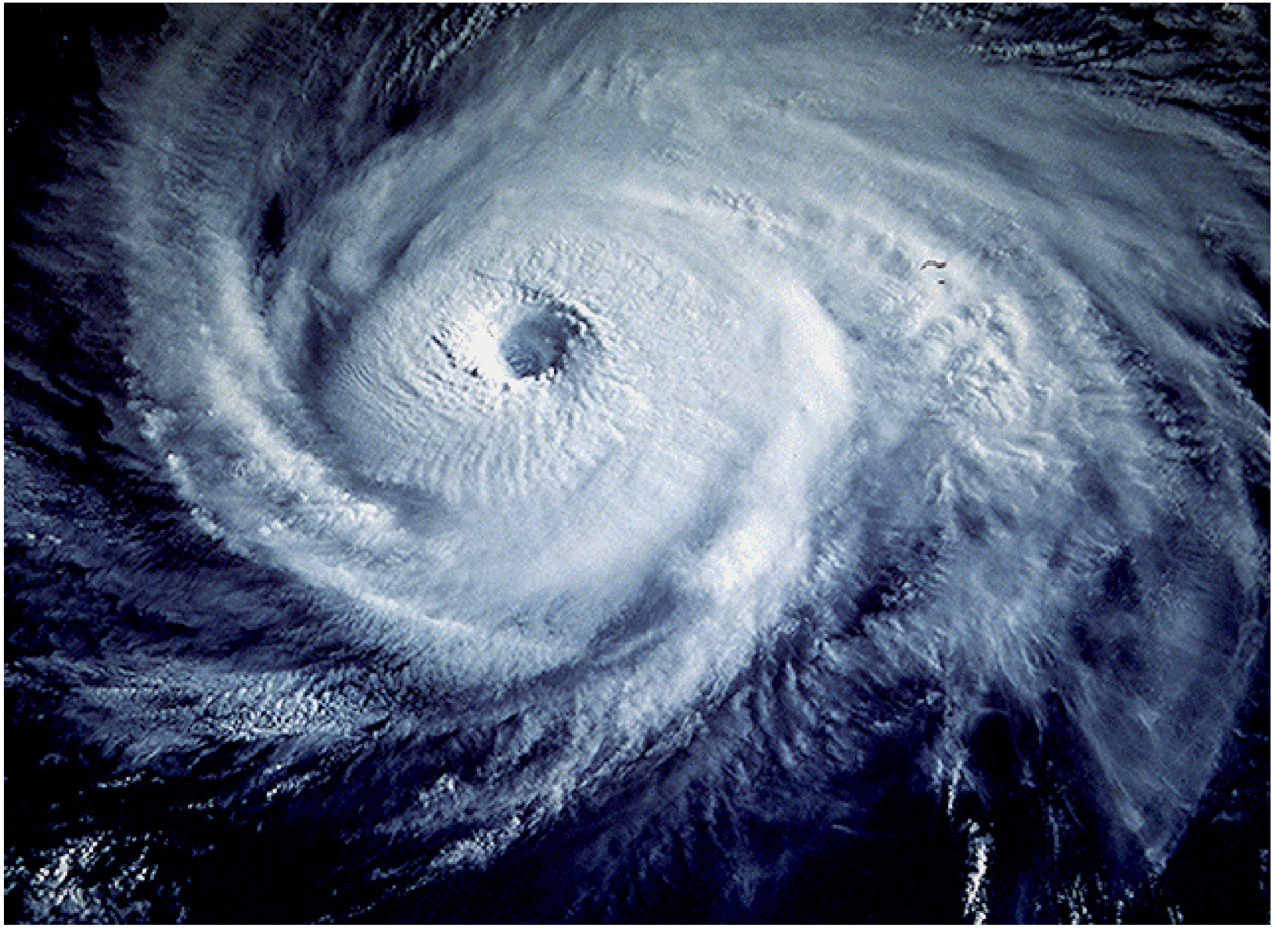
1.1 General Remarks (12)
Hydraulic Structures

1.1 General Remarks (13)
Hydrodynamics



1.1 General Remarks (14)

Meteorology



1.1 General Remarks (15)

Water Resources

1.2 Methods for Solving Fluid Dynamics Problems (1)

- Over the past half century, we have witnessed the rise to importance of a new methodology for attacking the complex problems in fluid mechanics and heat transfer. The new methodology has become known as **Computational Fluid Dynamics (CFD)**.
- In this approach, the equations that govern a process of interest are solved **numerically**. The evolution of numerical methods, especially **finite-difference methods** for solving **ordinary** and **partial differential equations**, started approximately with **the beginning of the twentieth century**.
- The explosion in computational activity did not begin until general availability of **high-speed digital computers, occurred in 1960s**.

1.2 Methods for Solving Fluid Dynamics Problems (2)

- Traditionally, both **experimental** and **theoretical** methods have been used to develop designs for equipment and vehicles involving fluid flow and heat transfer. With the advent of the digital computer, a third method, the **numerical approach**, has become available.
- Over the years, computer speed has increased much more rapidly than computer costs. **The net effect has been a phenomenal decrease in the cost of performing a given calculation**.
- The suggestion here is not that computational methods will soon completely replace experimental testing as a means to gather information for design purpose. Rather, **it is believed that computer methods will be used even more extensively in the future**.

1.2 Methods for Solving Fluid Dynamics Problems (3)

- The need for experiments will probably remain for quite some time in applications involving **turbulent flow, where it is presently not economically feasible to utilize computational models that are free of empiricism for most practical configurations**. This situation is destined to change eventually, since it has become clear that turbulent flows can be solved by **direct numerical simulation (DNS)** as computer hardware and algorithms improve in the future. The prospects are also bright for the increased use of **large-eddy simulations (LES)**, where modeling is required for only the smallest scales.
- In applications involving **multiphase flows, boiling, or condensation, especially in complex geometries, the experimental method remains the primary source of design information.** Progress is being made in computational models for these flows.

1.2 Methods for Solving Fluid Dynamics Problems (4)

- *Analytical Fluid Dynamics (AFD)*
  Mathematical analysis of governing equations, including exact and approximate solutions.
- *Computational Fluid Dynamics (CFD)*
  Numerical solution of the governing equations
- *Experimental Fluid Dynamics (EFD)*
  Observation and data acquisition.

1.2 Methods for Solving Fluid Dynamics Problems (5)
Analytical Fluid Dynamics

How fast do tsunamis travel in the deep ocean?

Incompressible Navier-Stokes equations

$$\frac{\partial \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla)\mathbf{U} = \frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{U}$$

Linearized wave equation for inviscid, irrotational flow

$$\nabla^2 \phi = 0, \mathbf{U} = \nabla \phi$$

$$\frac{\partial \phi}{\partial z} = 0 \text{ on } z = -h$$

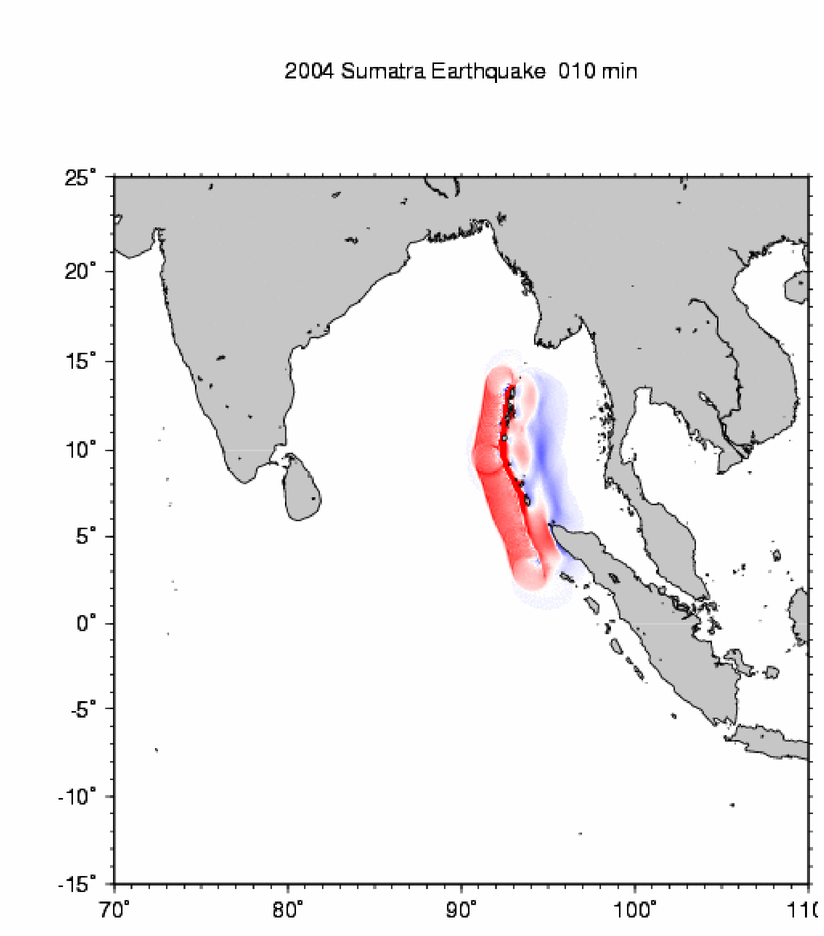
$$\frac{\partial^2 \phi}{\partial t^2} = -g\frac{\partial \phi}{\partial z} \text{ on } z = 0$$

Shallow-water approximation, l/h >> 1

$$c = \sqrt{\frac{g}{k}\tanh kh} \Longrightarrow c = \sqrt{gh}$$

For g = 32.2 ft/s^2 and h=10000 ft, c=567 ft/s = 387 miles/hr

1.2 Methods for Solving Fluid Dynamics Problems (6)
Computational Fluid Dynamics



Animation by Vasily V. Titov, Tsunami Inundation Mapping Efforts, NOAA/PMEL

- In comparison to analytical methods, which are good for providing solutions for simple geometries or behavior for limiting conditions (such as linearized shallow water waves), CFD provides a tool for solving problems with nonlinear physics and complex geometry.

1.2 Methods for Solving Fluid Dynamics Problems (7)
Experimental Fluid Dynamics

- Oregon State University Wave Research Laboratory
- Model-scale experimental facilities
  - Tsunami Wave Basin
  - Large Wave Flume
- Dimensional analysis

is very important in designing a model experiment which represents physics of actual problem

❑Comparison of Approaches:

Approach	Advantages	Disadvantages
Experimental	(1) Capable of being most realistic	(1) Equipment required (2) Scaling problems (3) Tunnel corrections (4) Measurement difficulties (5) Operating costs
Theoretical	(1) Clean, general information, which is usually in formula form	(1) Restricted to simple geometry and physics (2) Usually restricted to linear problems
Computational	(1) No restriction to linearity (2) Complicated physics can be treated (3) Time evolution of flow can be obtained	(1) Truncation errors (2) Boundary conditions problems (3) Computer costs

Why use CFD?

- Analysis and Design
  - 1. Simulation-based design instead of "build & test"
    - More cost effective and more rapid than EFD
    - CFD provides high-fidelity database for diagnosing flow field
  - 2. Simulation of physical fluid phenomena that are difficult for experiments
    - Full scale simulations (e.g., ships and airplanes)
    - Environmental effects (wind, weather, etc.)
    - Hazards (e.g., explosions, radiation, pollution)
    - Physics (e.g., planetary boundary layer, stellar evolution)
- Knowledge and exploration of flow physics

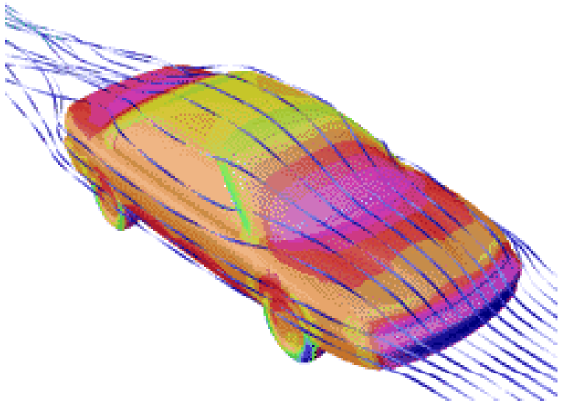
1.3 The Impact of Computational Fluid Dynamics---Some Other Examples (1)

- Historically, the early development of **CFD in the 1960s and 1970s was driven by the needs of the aerospace community.** However, **modern CFD cuts across all disciplines where the flow of a fluid is important.**

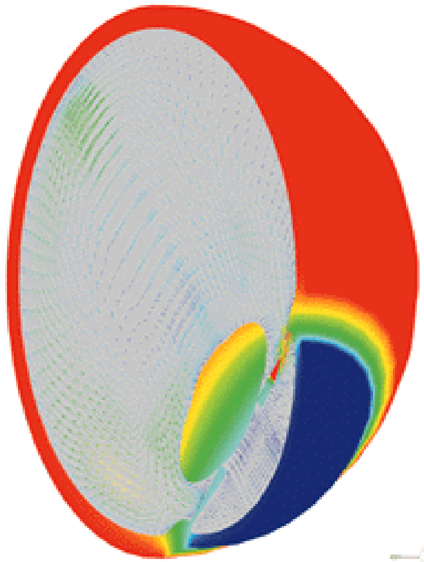
1.3 The Impact of Computational Fluid Dynamics---Some Other Examples (2)

- Where is CFD used?
  - ***Aerospace***
  - ***Automotive***
  - ***Biomedical***
  - Chemical Processing
  - HVAC
  - Hydraulics
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports

F18 Store Separation



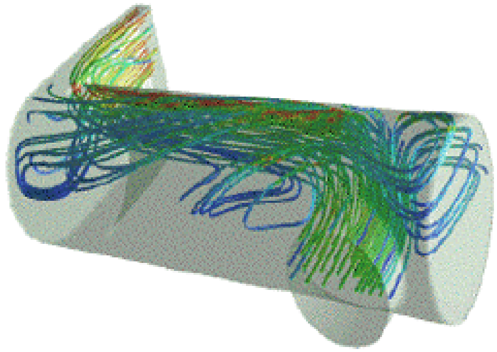
Automotive



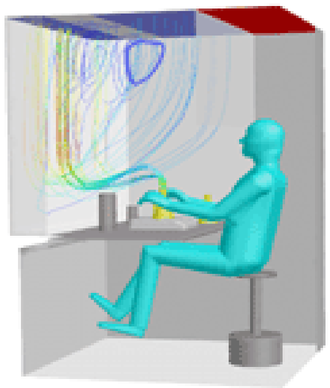
Temperature and natural convection currents in the eye following laser heating.

1.3 The Impact of Computational Fluid Dynamics---Some Other Examples (3)

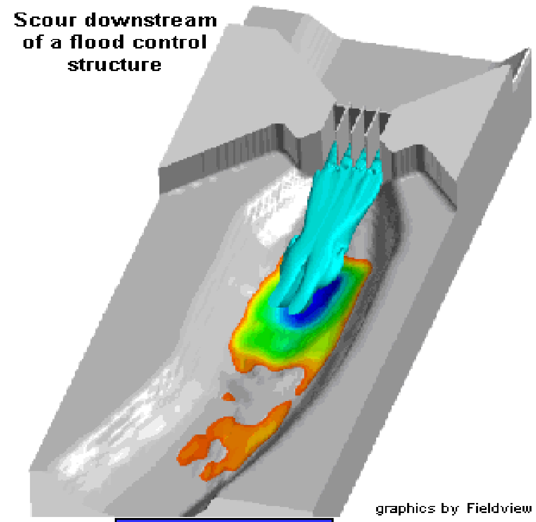
- Where is CFD used?
  - Aerospacee
  - Automotive
  - Biomedical
  - ***Chemical Processing***
  - ***HVAC***
  - ***Hydraulics***
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports



Chemical Processing



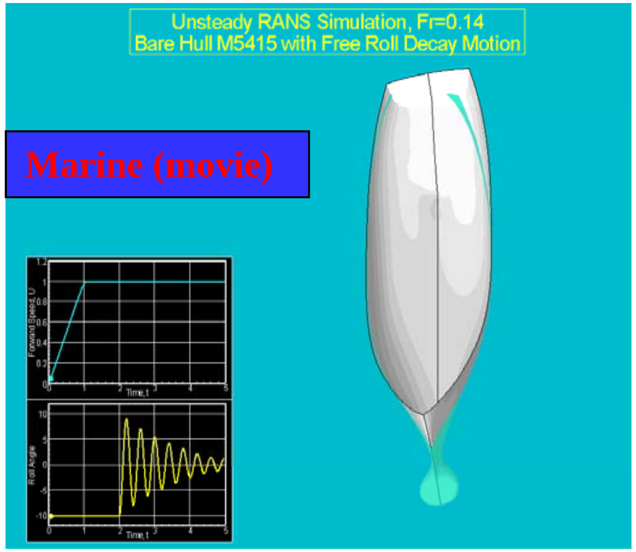
HVAC



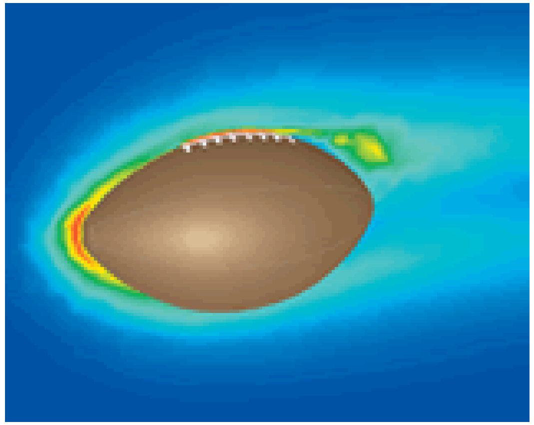
Hydraulics

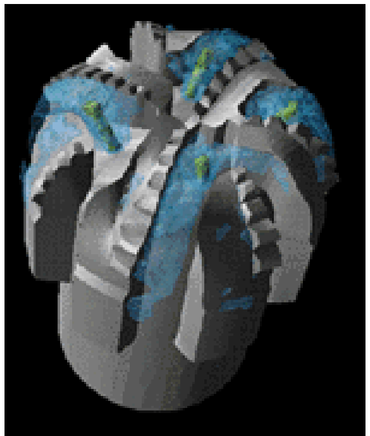
1.3 The Impact of Computational Fluid Dynamics---Some Other Examples (4)

- Where is CFD used?
  - Aerospace
  - Automotive
  - Biomedical
  - Chemical Processing
  - HVAC
  - Hydraulics
  - ***Marine***
  - ***Oil & Gas***
  - ***Power Generation***
  - ***Sports***

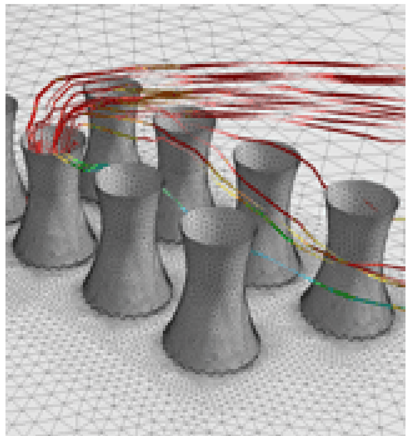


Sports





Oil & Gas



Power Generation

1.4 CFD Process (1)

- Outline of CFD Process
  - Model Equations
  - Discretization
  - Grid Generation
  - Boundary Conditions
  - Solve
  - Post-Processing
  - Uncertainty Assessment
- Conclusions
- Example: FLOWLAB

1.4 CFD Process (2)
Model Equations

- Most commercial CFD codes solve the continuity, Navier-Stokes, and energy equations
  - Coupled, ***non-linear***, partial differential equations
  - For example, incompressible form

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial \rho v}{\partial y}+w\frac{\partial \rho v}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{\mu}{\rho}\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

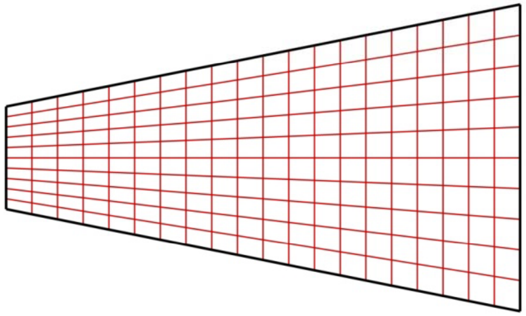
1.4 CFD Process (3)

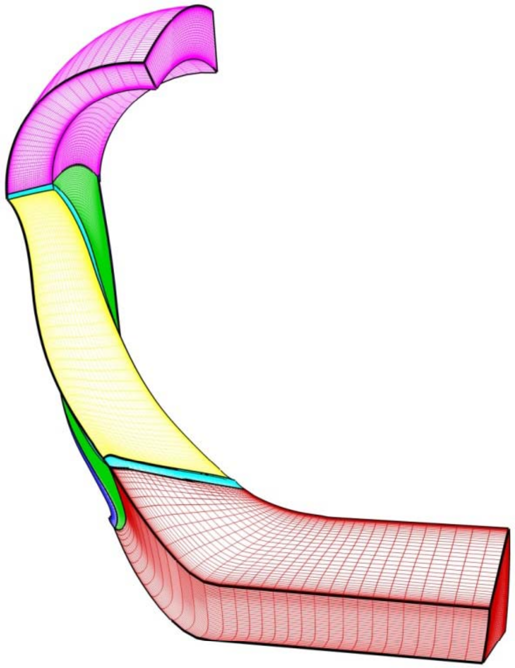
Discretization
Grid Generation

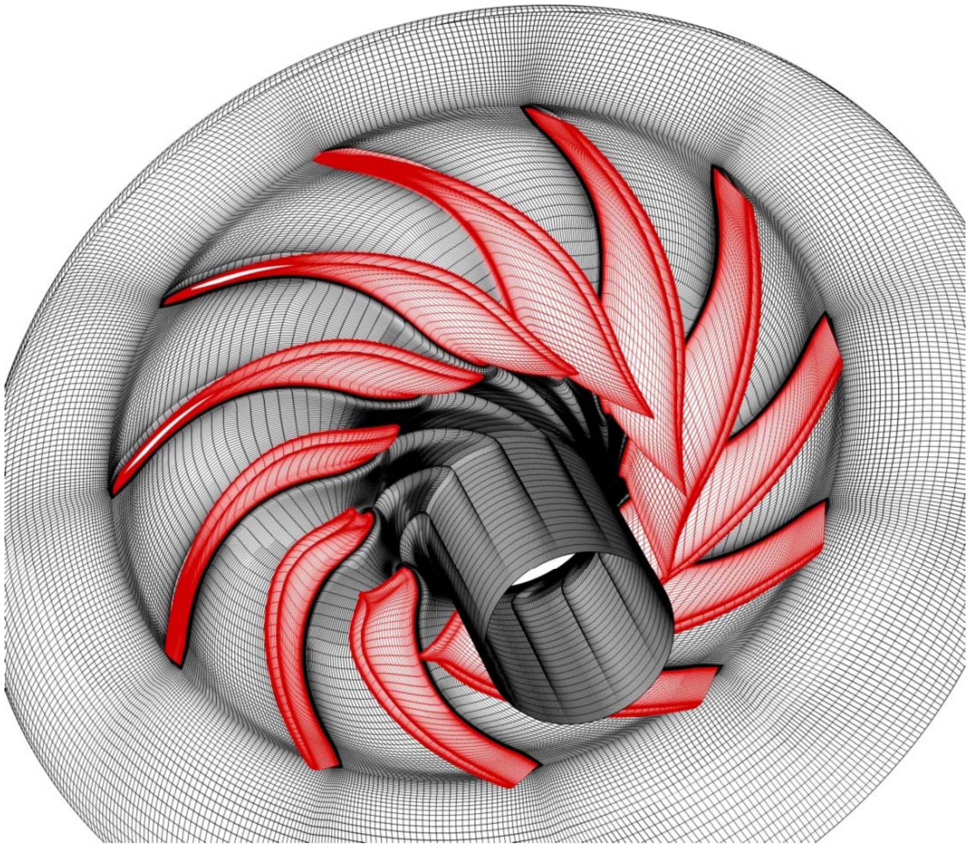
- Flow field must be treated as a discrete set of points (or volumes) where the governing equations are solved.
- Many types of grid generation: type is usually related to capability of flow solver.
  - Structured grids
  - Unstructured grids
  - Hybrid grids: some portions of flow field are structured (viscous regions) and others are unstructured
  - Overset (Chimera) grids

1.4 CFD Process (4)

Grid Generation: Structured Grids



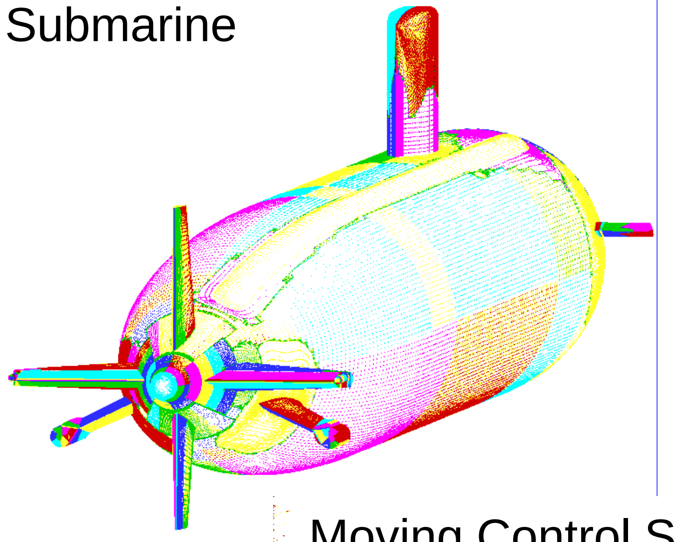




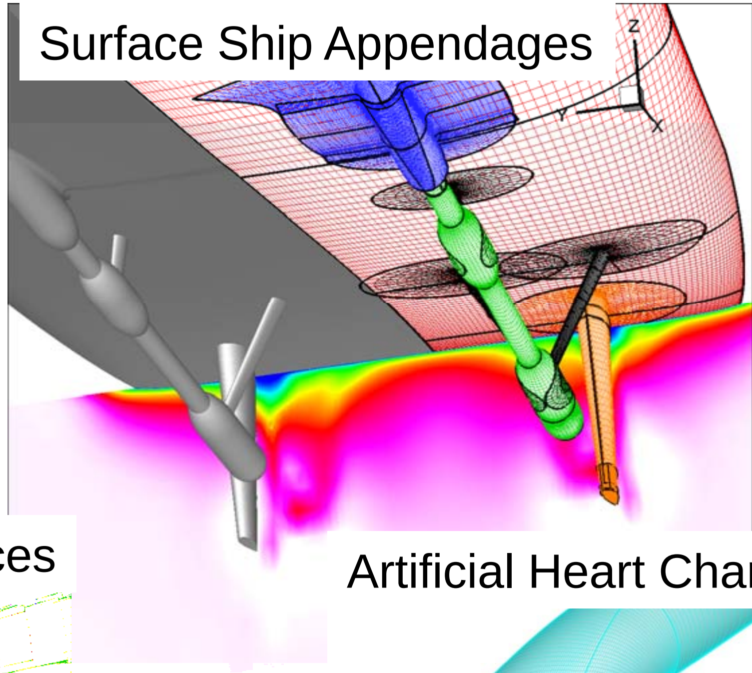
1.4 CFD Process (5)

Grid Generation: Structured Overset Grids

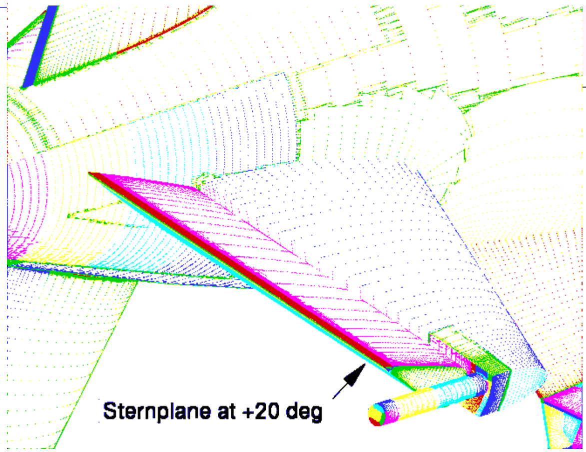
Submarine



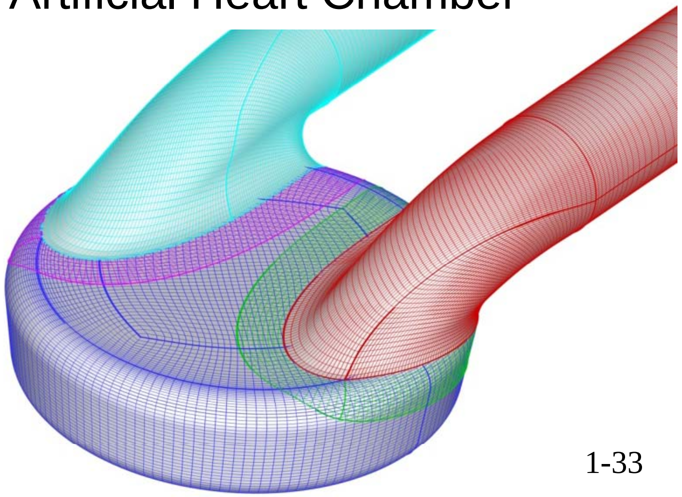
Surface Ship Appendages



Moving Control Surfaces

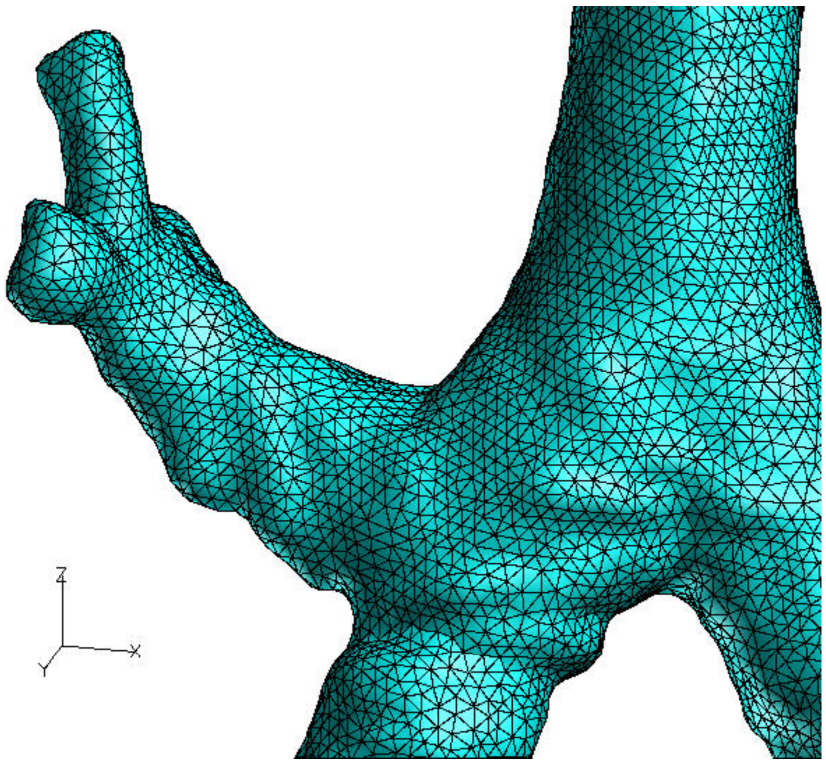


Artificial Heart Chamber

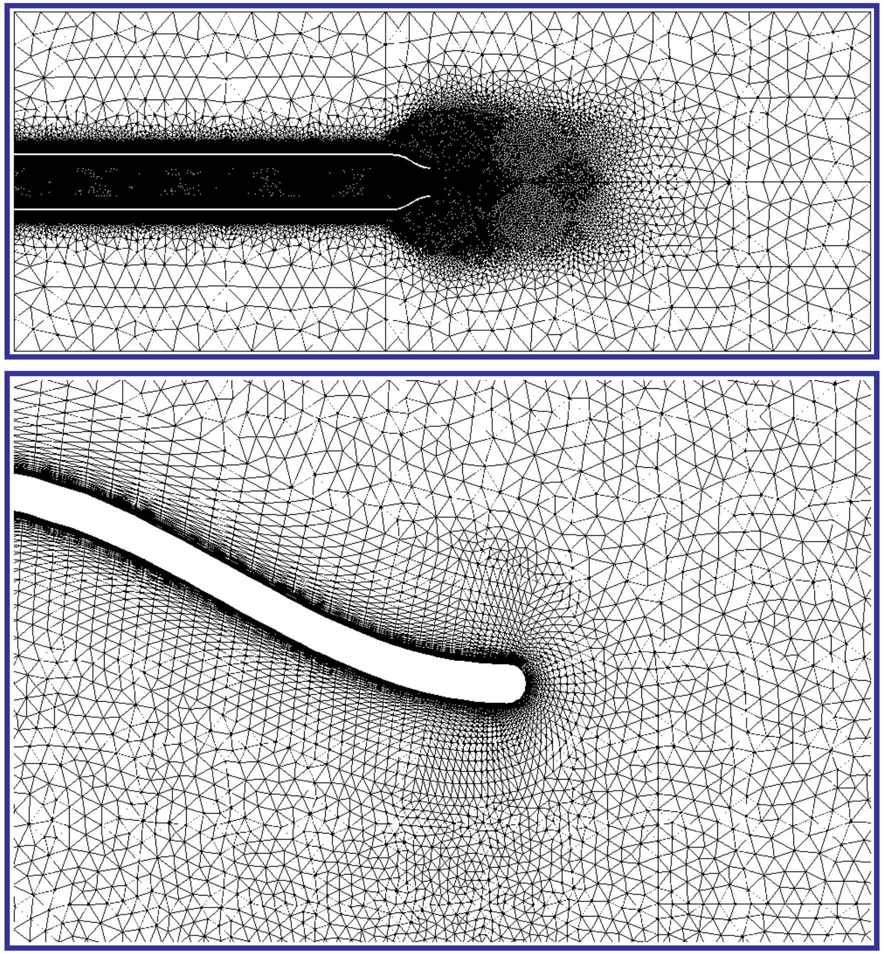


1.4 CFD Process (6)

Grid Generation: Unstructured Grids



Branches in Human Lung



Structured-Unstructured Nozzle Grid

1.4 CFD Process (7)
Discretization: Algebraic equations

- To solve NSE, we must convert governing PDE's to algebraic equations
  - Finite difference methods (FDM)
    - Each term in NSE approximated using Taylor series, e.g.,

$$\frac{\partial U}{\partial x} = \frac{U_{i+1} - U_i}{\Delta x} + O(\Delta x)$$

$$\frac{\partial^2 U}{\partial x^2} = \frac{U_{i+1} - 2U_i + U_{i-1}}{(\Delta x)^2} + O(\Delta x)^2$$

  - Finite volume methods (FVM)
    - Use CV form of NSE equations on each grid cell ! Most popular approach, especially for commercial codes
  - Finite element methods (FEM)
    - Solve PDE's by replacing continuous functions by piecewise approximations defined on polygons, which are referred to as elements. Similar to FDM.

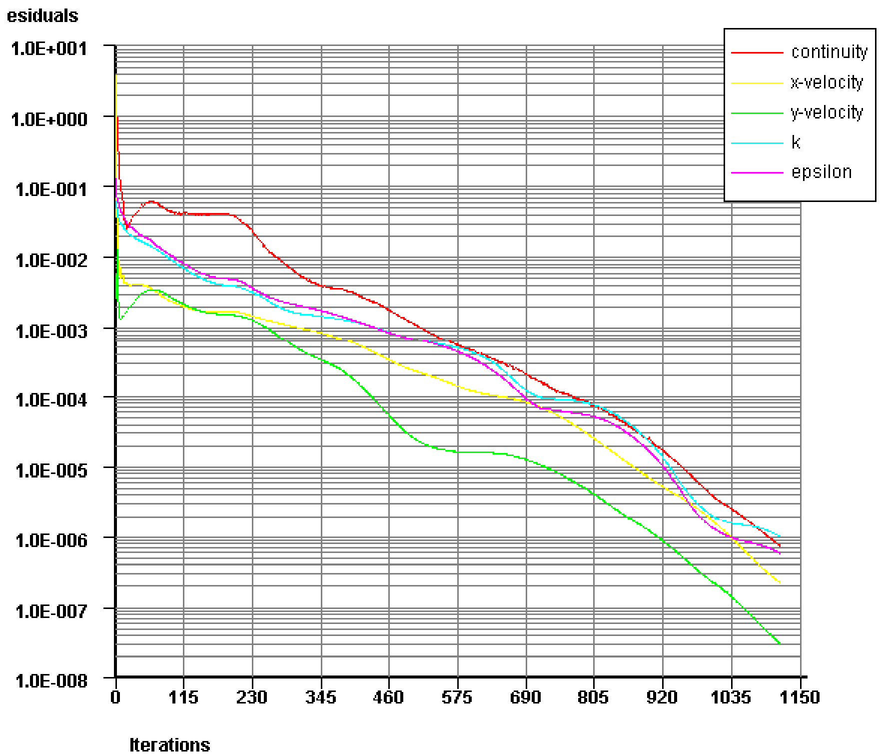
1.4 CFD Process (8)
Boundary Conditions

- Typical conditions
  - Wall
    - No-slip (u = v = w = 0)
    - Slip (tangential stress = 0, normal velocity = 0)
    - With specified suction or blowing
    - With specified temperature or heat flux
  - Inflow
  - Outflow
  - Interface Condition, e.g., Air-water free surface
  - Symmetry and Periodicity
- Usually set through the use of a graphical user interface (GUI) – click & set

1.4 CFD Process (9)

Solve

- Run CFD code on computer
  - 2D and small 3D simulations can be run on desktop computers (e.g., FlowLab)
  - Unsteady 3D simulations still require large parallel computers
- Monitor Residuals
  - Defined two ways
    - Change in flow variables between iterations
    - Error in discrete algebraic equation



1.4 CFD Process (10)
Uncertainty Assessment

- Process of estimating errors due to numerics and modeling
  - Numerical errors
    - Iterative non-convergence: monitor residuals
    - Spatial errors: grid studies and Richardson extrapolation
    - Temporal errors: time-step studies and Richardson extrapolation
  - Modeling errors (Turbulence modeling, multi-phase physics, closure of viscous stress tensor for non-Newtonian fluids)
    - Only way to assess is through comparison with benchmark data which includes EFD uncertainty assessment.

1.4 CFD Process (11)
Conclusions-1

- Capabilities of Current Technology
  - Complex real-world problems solved using Scientific Computing
  - Commercial software available for certain problems
  - Simulation-based design (i.e., logic-based) is being realized.
  - Ability to study problems that are either expensive, too small, too large, or too dangerous to study in laboratory
    - Very small : nano- and micro-fluidics
    - Very large : cosmology (study of the origin, current state, and future of our Universe)
    - Expensive : engineering prototypes (ships, aircraft)
    - Dangerous : explosions, response to weapons of mass destruction

1.4 CFD Process (12)

Conclusions-2

- Limitations of Current Technology
  - For fluid mechanics, many problems not adequately described by Navier-Stokes equations or are beyond current generation computers.
    - Turbulence
    - Multi-phase physics: solid-gas (pollution, soot), liquid-gas (bubbles, cavitation); solid-liquid (sediment transport)
    - Combustion and chemical reactions
    - Non-Newtonian fluids (blood; polymers)
  - Similar modeling challenges in other branches of engineering and the sciences

1.4 CFD Process (13)
Conclusions-3

- Because of limitations, need for experimental research is great
- However, focus has changed
  - From
    - Research based solely upon experimental observations
    - Build and test (although this is still done)
  - To
    - High-fidelity measurements in support of validation and building new computational models.
- Currently, the best approach to solving engineering problems often uses simulation and experimentation

1.4 CFD Process (14)
Example: FlowLab-1

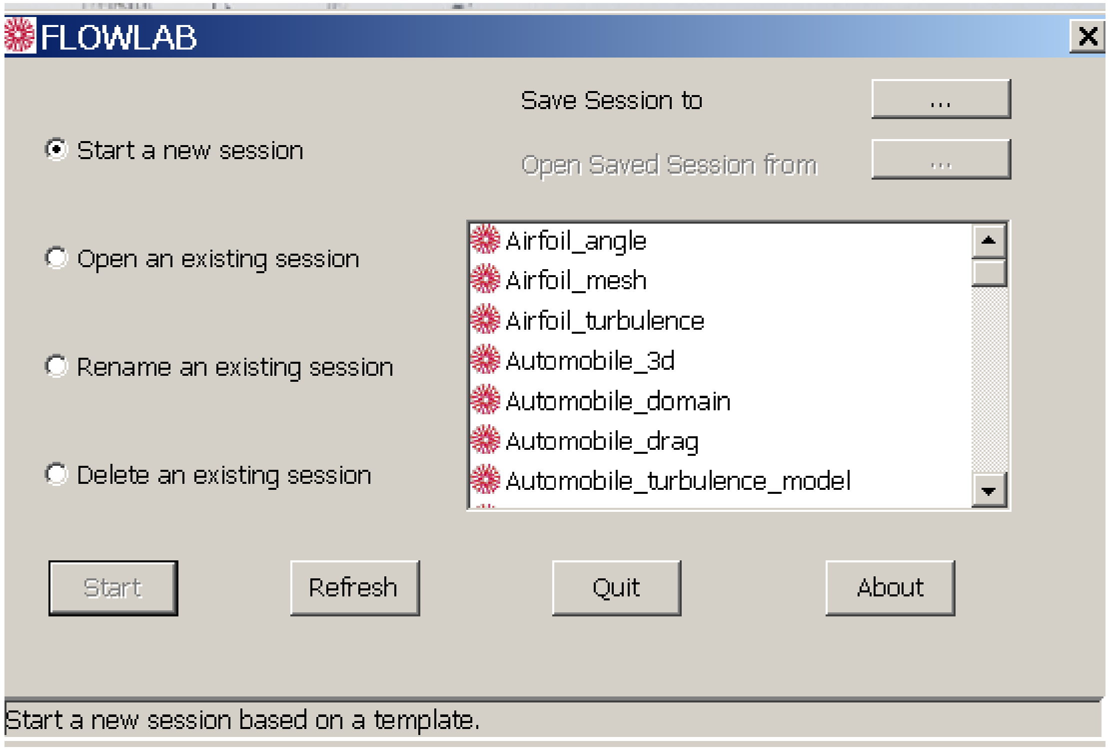
- FlowLab (http://www.flowlab.fluent.com/)
  - Educational software that uses the power of flow visualization through CFD to teach basic fluid mechanics principles in the engineering classroom.
  - Runs Fluent's general purpose CFD code, FLUENT, and pre-processor, GAMBIT, in the background, with a user-friendly, student-specific graphical user interface (GUI) on its front end.
  - Based on ready-to-use exercises, FlowLab eliminates the long learning curve associated with general fluid flow modeling packages, making it easy to deploy as part of the undergraduate or masters-level curriculum.
    - Templates for Problems 15-26 – 15-88 in Cengel and Cimbala.

1.4 CFD Process (15)
Example: FlowLab-2

- Templates are
  - Nearly self-guided
  - Designed to teach specific lessons
    - Effect of grid resolution
    - Effect of domain size
    - Flow physics and sensitivity to parameters, e.g., diffuser angle, airfoil angle of attack, etc.
  - GUI for post-processing is easy to learn
    - Contours
    - Streamlines
    - Vector Plots

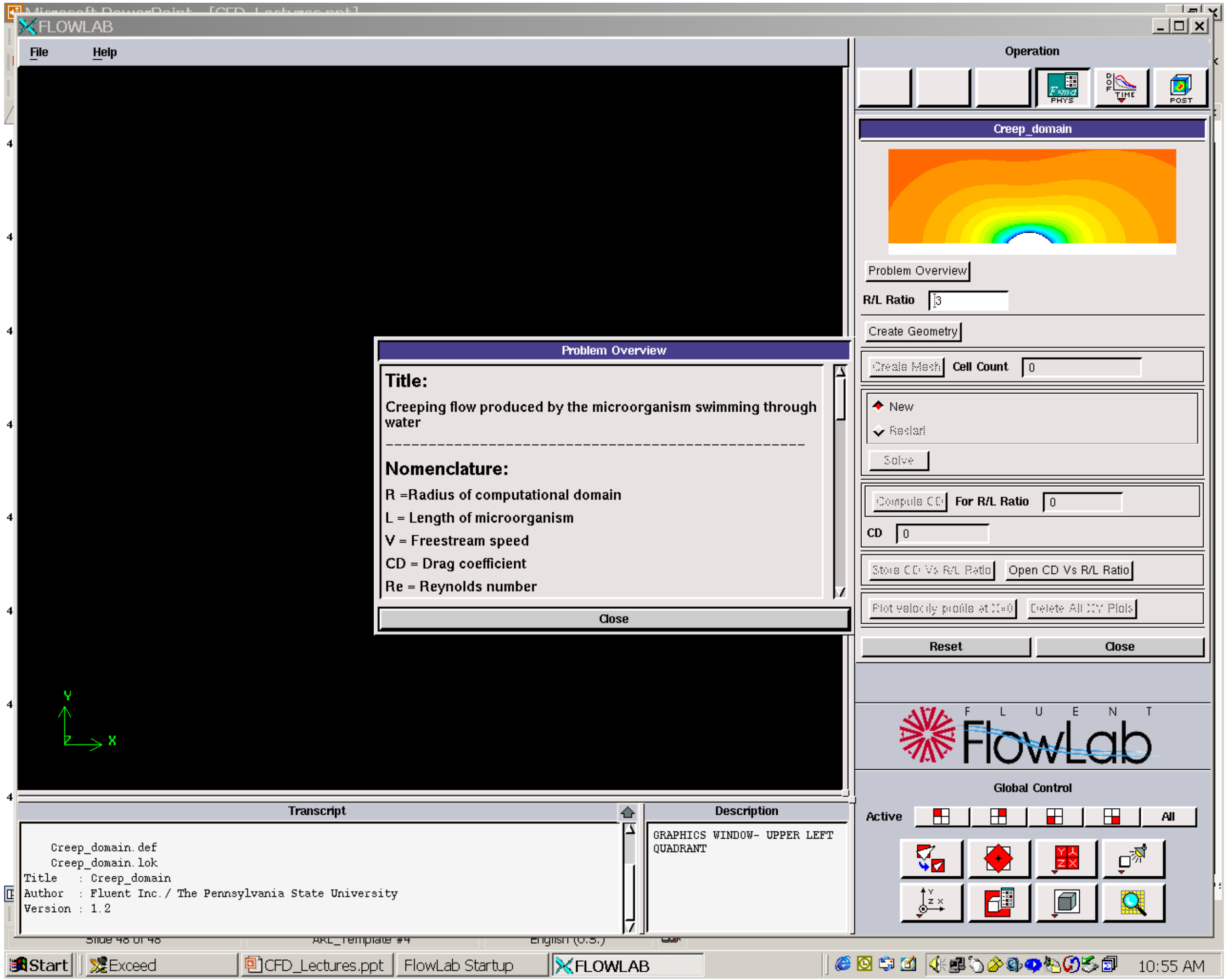
1.4 CFD Process (16)
Example: FlowLab-3

- Step 1: Select template from startup menu



1.4 CFD Process (17)
Example: FlowLab-4

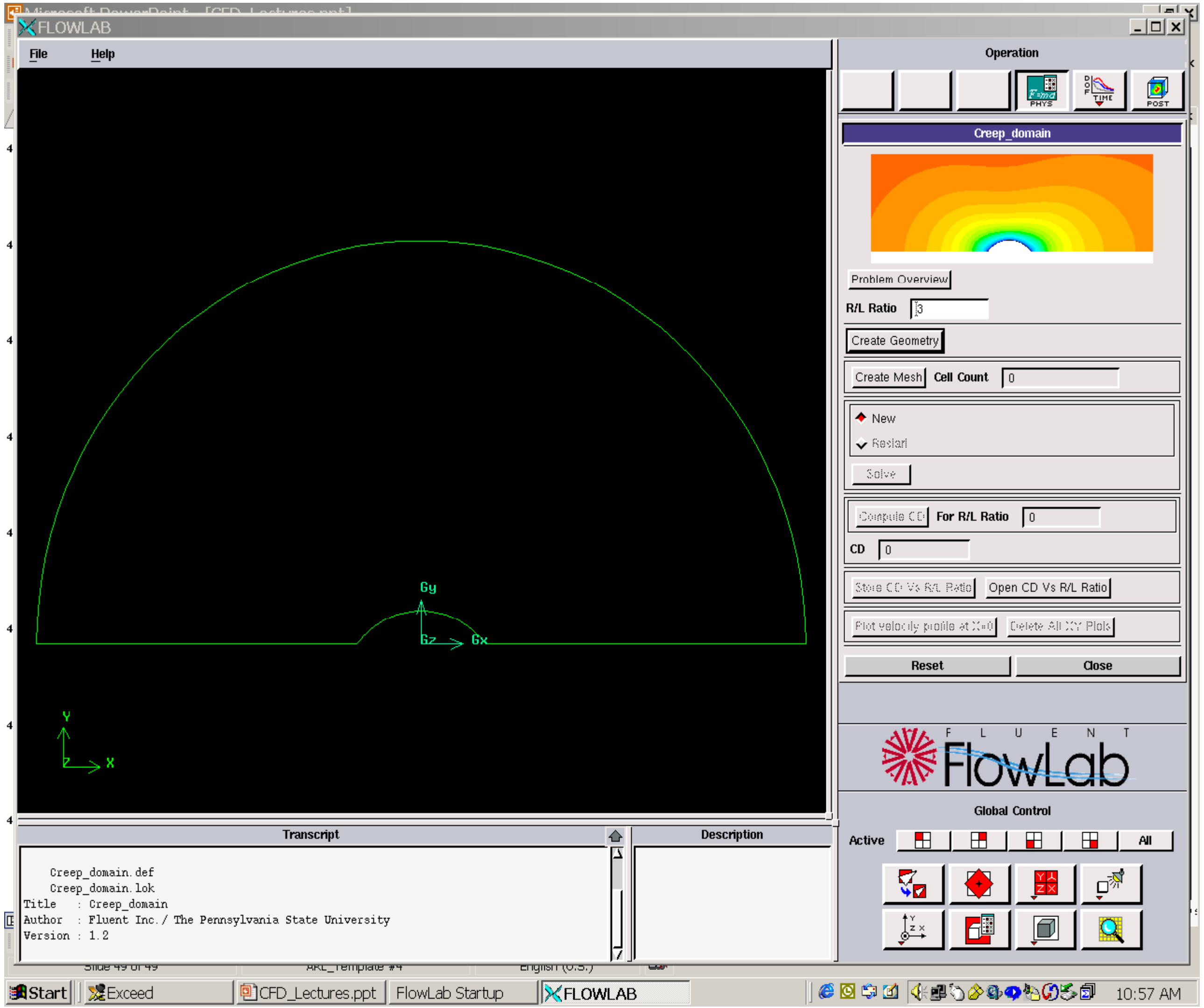
- Step 2: Read problem overview



1.4 CFD Process (18)

Example: FlowLab-5

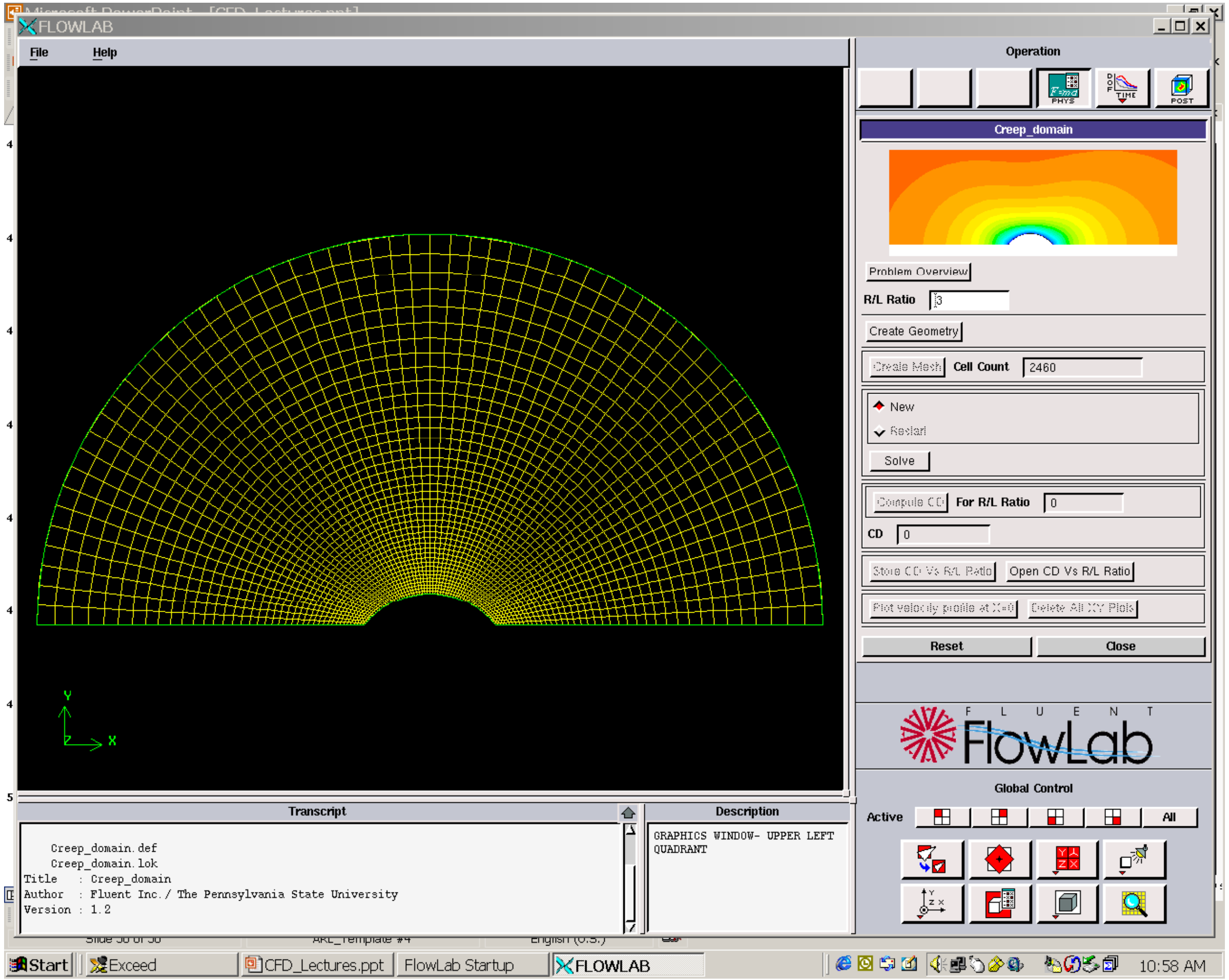
- Step 3: Create Geometry



1.4 CFD Process (19)

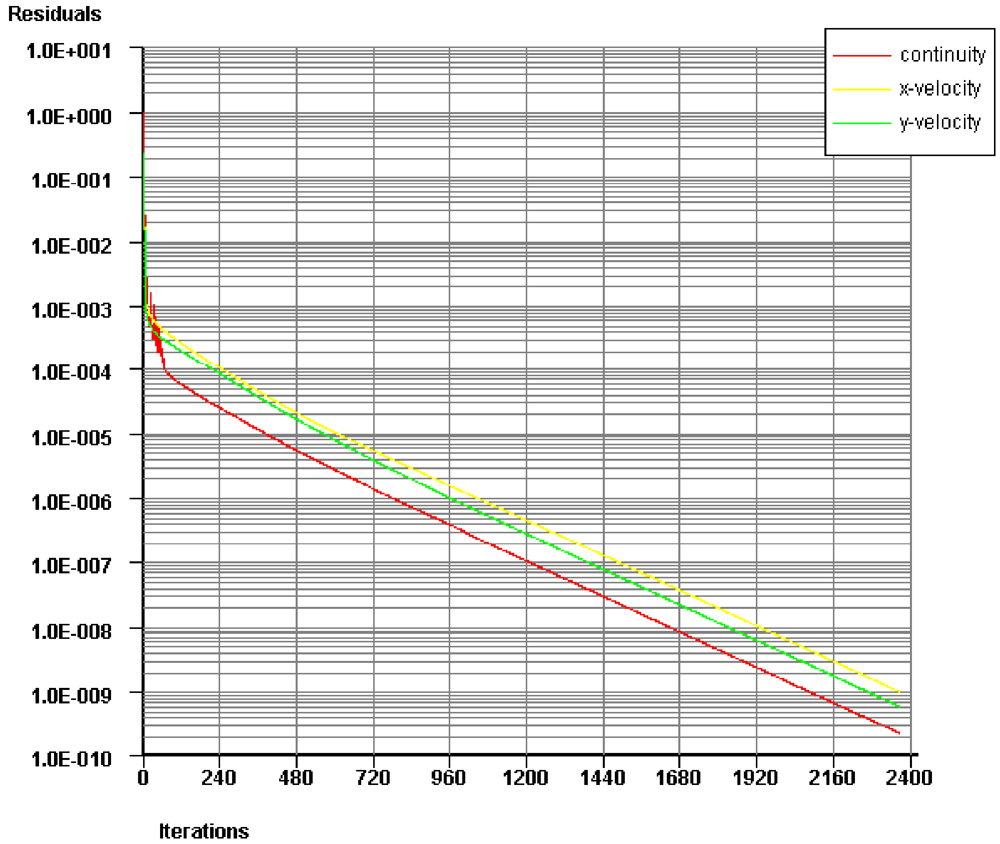
Example: FlowLab-6

- Step 4: Create Mesh



1.4 CFD Process (20)
Example: FlowLab-7

- Step 5: Solve, monitor residuals



1.4 CFD Process (21)

Example: FlowLab-8

- Step 6: Post-process
- Compute C_D
- Plot velocity profile
- Other contours