ME469B - Spring 2007

COMPUTATIONAL FLUID DYNAMICS USING COMMERCIAL CFD CODES

Gianluca Iaccarino

Dept. Mechanical Engineering

Bldg. 500 RM 204 (RM500-I)
Ph. 650-723-9599
Email: jops@stanford.edu

ME469B - Topics

Introduction to CFD Analysis and CFD Codes

Geometry Modeling and (Unstructured) Grid Generation

Solution Algorithms for Incompressible/Compressible Flows
(Steady/Unsteady, Flow Adaptivity, Moving Meshes, etc.)

Advanced Physical Models (Turbulence, Multiphase Flows, Heat Transfer, etc.)

User Programming and Automation

ME469B - Outline

Theoretical Background: 50%

Applications and Examples: 50%

Lectures:

Regular lectures
Tutorials/Lab
(keep friday 1.15-2.30 free)

Final Project

Software is accessible on the Stanford Network
Several Windows Workstations are Available

ME469B - Grades

No Formal Final Exam at the End of the Quarter

Grade is:

60% Final Project
40% Homeworks

Additional exercises are optional!

ME469B - Final Project

The final project is a "complete" CFD project

…from problem statement to discussion of results…

Proposals for projects are welcome

…let's discuss them in due time.

Lectures end early to leave enough time for the final project

…do not start at the last minute!

ME469B - General Information

Class WEB site:

http://me469b.stanford.edu

- Announcements
- Lecture Notes
- Homeworks and Exercises
- Examples
- Additional References
- Software User Manuals

In ME469A:

- Grid generation
    - Basic information
- Numerical Methods
    - Theoretical background
    - Analysis of discretization schemes
    - Implicit/explicit solvers
    - Convergence acceleration techniques (i.e. multigrid)
    - Methods for Navier-Stokes equations
- Simple codes for specific problems
    - Burgers equations
    - Advection/diffusion problem
    - …

In ME469B:

- Grid generation
    - Background information
    - Emphasis on unstructured-mesh techniques
- Numerical Methods
    - Background information
    - Discretization accuracy
    - Efficiency and Acceleration techniques
- Commercial “general-purpose” CFD packages
    - Complex problems
    - Real-world engineering tools

In ME469B (cont'd):

- Advanced CFD Techniques
    - Grid adaptivity
    - Moving Meshes & Coordinate systems
- Advanced Physical Models
    - Turbulence Models
    - Multiphase Flow Models
    - Heat Transfer
    - …
- Credibility of CFD solutions
    - Validation: Are we solving the correct equations?
    - Verification: Are we solving the equations correctly?
    - Sensitivity analysis

Requirements:

- Basic Fluid Dynamics
    - Laminar/Turbulent Flows
    - Compressible/Incompressible Fluids
    - Navier-Stokes equations
    - Heat transfer…
- Numerical Methods
    - Upwind/Central/… discretizations
    - Implicit/Explicit schemes
    - Convergence/Stability/Accuracy
- Software
    - Basic Unix/Linux
    - Basic Networking
    - Basic of C Programming

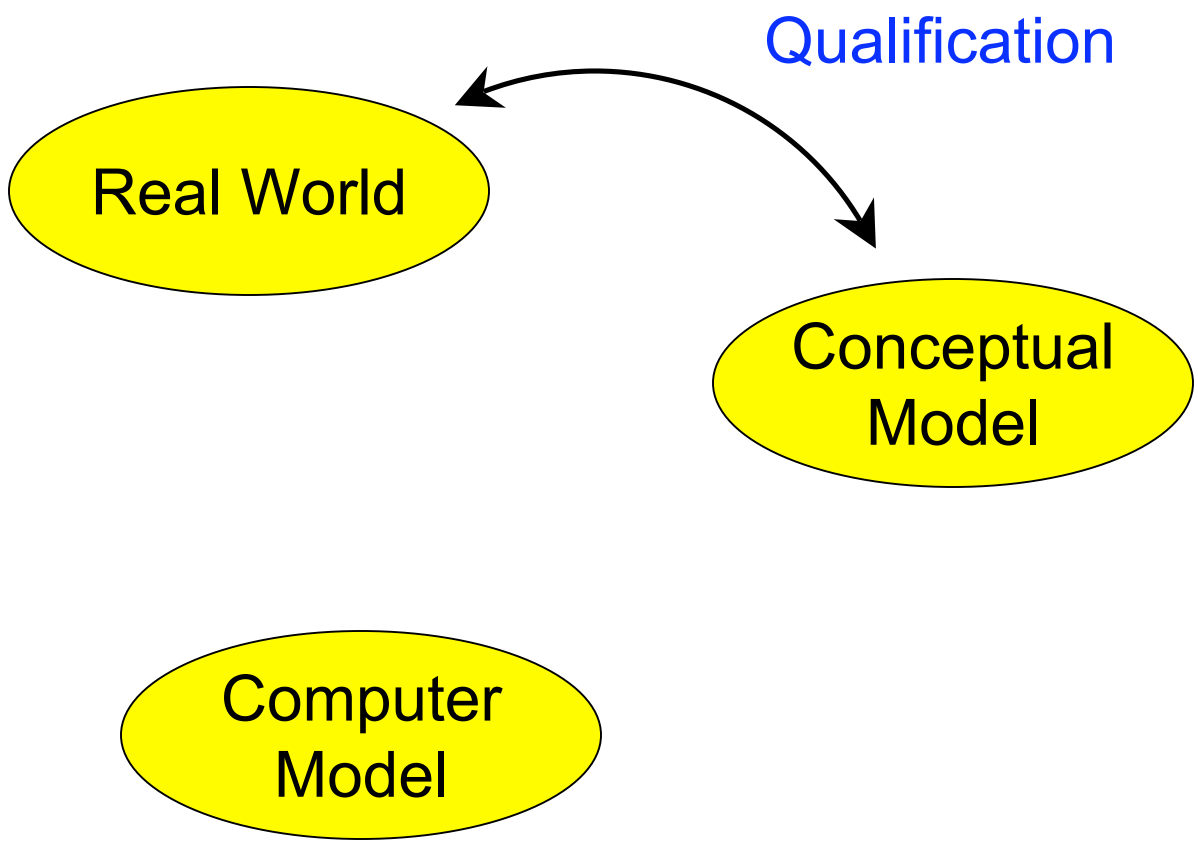
A Typical CFD Project

Real World — The intended application

Conceptual Model — Physical laws, hypothesis and models (e.g. a set of PDEs) and initial and boundary conditions

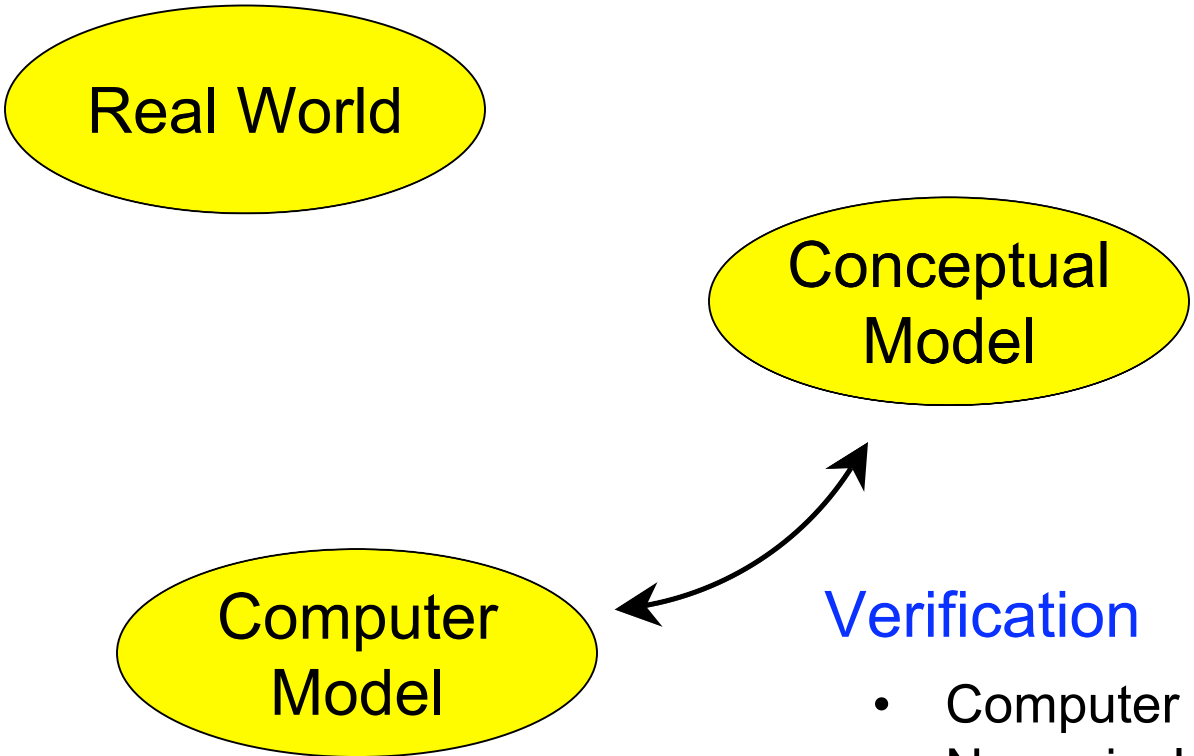
Computer Model — A set of computational algorithms that allow to build a numerical, approximated solution to the conceptual model

A Typical CFD Project



Qualification: Determination of the adequacy of the conceptual model to provide acceptable level of agreement for the intended applications

A Typical CFD Project

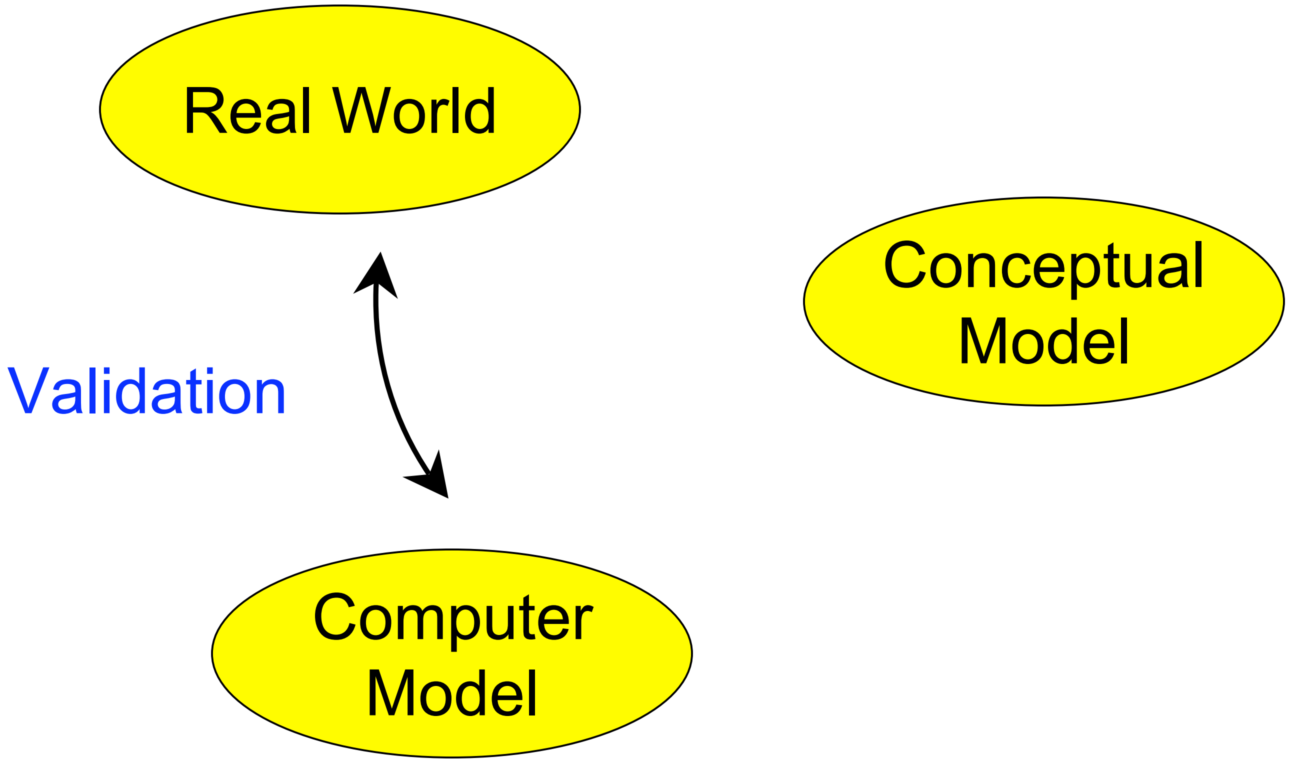


Verification

- Computer codes
- Numerical solutions

Verification: Process of determining that (1) the model implementation accurately represents the conceptual model and (2) the solution to the model is accurate

A Typical CFD Project

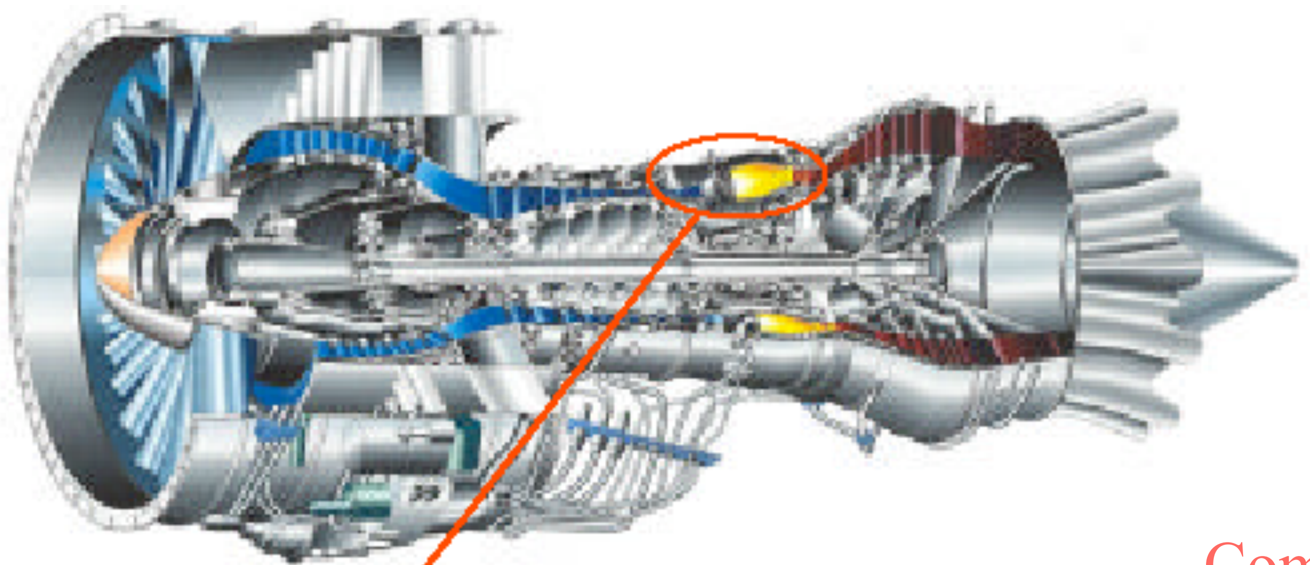


Validation: Process of determining the degree to which the model is an accurate representation of reality from the perspective of the intended application

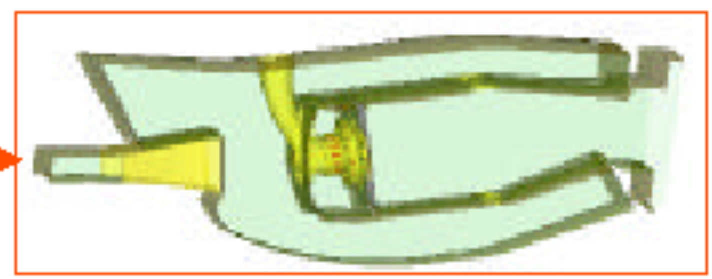
A Typical CFD Project

- Formulate the basic assumptions
  - Compressible/Incompressible fluid (Mach number)
  - Viscous/Inviscid Flow (Reynolds Number)
  - …
- Define the physical and computational problem
  - Geometry definition
  - Grid generation
  - Boundary condition set-up
- Choose a solution strategy
  - Space/Time Discretization scheme
  - Set-up Simulation Monitors
- Analyze the results
  - Evaluate convergence
  - Evaluate grid dependence
  - Postprocess data of interest

Flow in a Jet Engine



Combustor

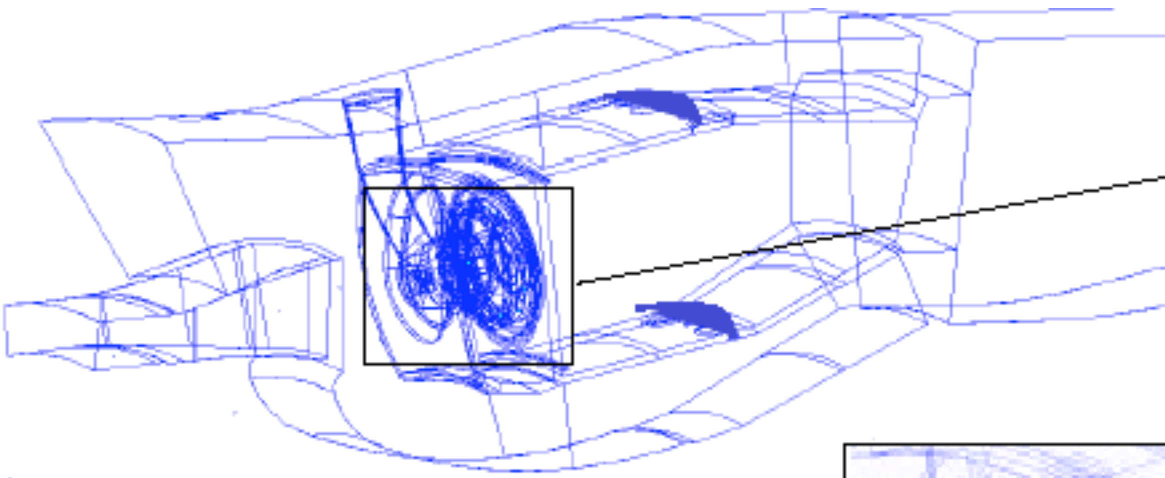


Pratt & Whitney PW6000

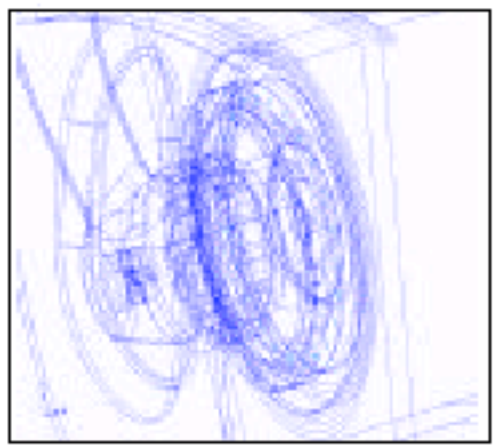
CFD of the flow in a Jet Engine

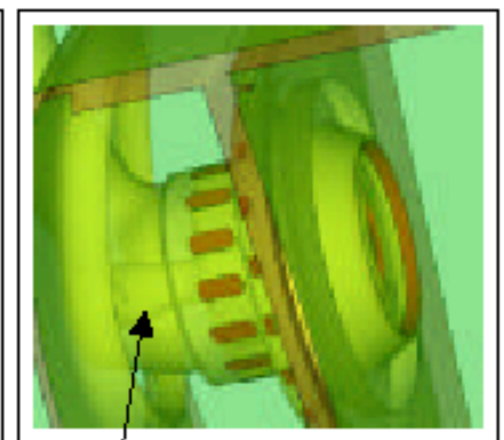
- Complex Geometry
  - Full geometry consists of 18 injectors
  - Intricate passages
  - Large scale overall but several “crucial” small components
- Complex Physics
  - Turbulent, highly unsteady flow
  - Multiphase (air/liquid fuel)
  - Reacting flow
  - Pollutants & Particulate
  - Heat transfer, radiation
- Large scale calculation
  - Order 100 million grid cells
  - Run on largest computer in US (>1000s CPU @ Nat. Lab.)

Geometry Definition



Detail of the Injector Geometry





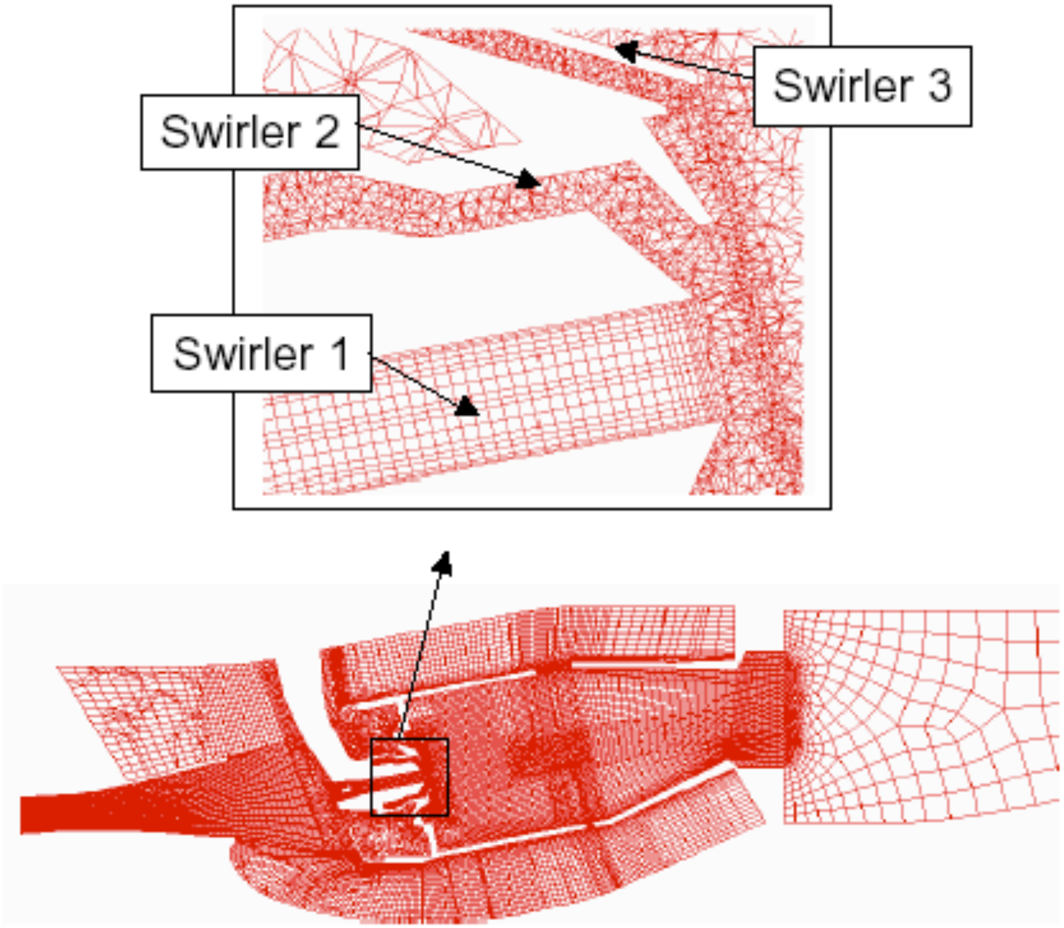
An IGES File with the PW6000 geometry

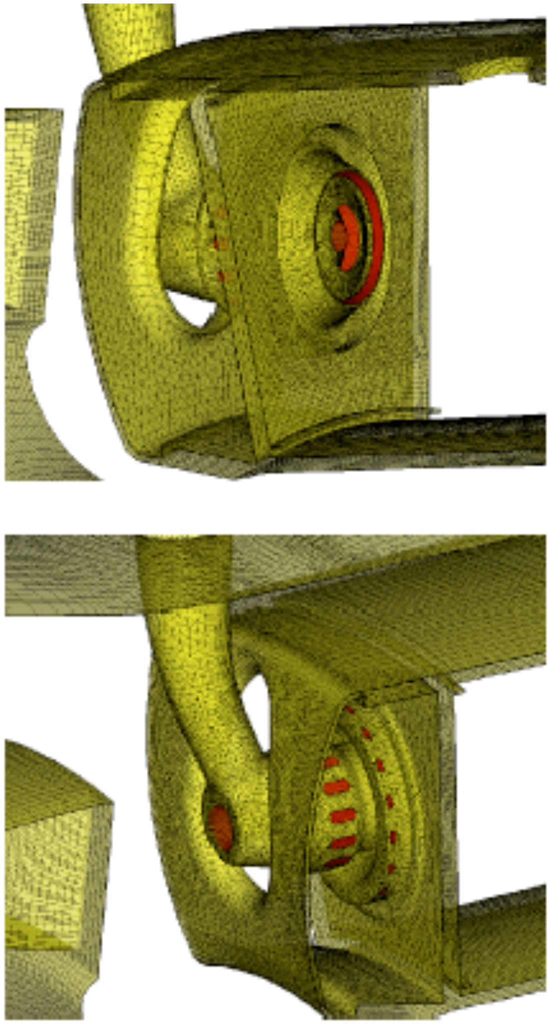
was obtained from PW

It Contained more than 9000 entities (surfaces, edges, vertices)

Both the interior and exterior of the Fuel Injector are modeled

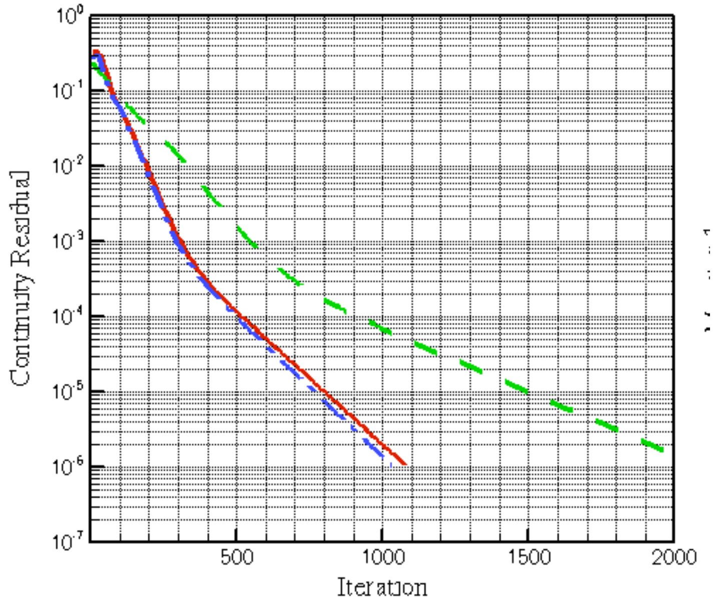
Grid Generation





It requires from few days to several months.

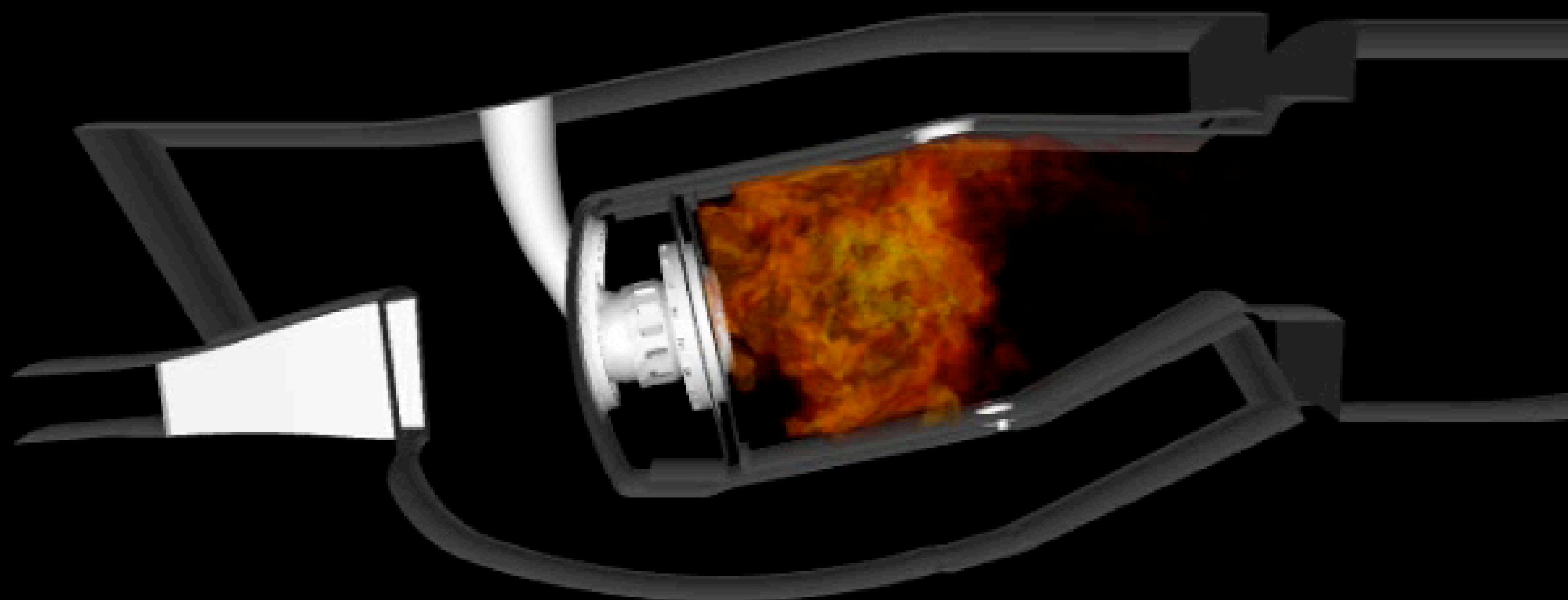
Flow Solution



Typically it consists of an iterative procedure…

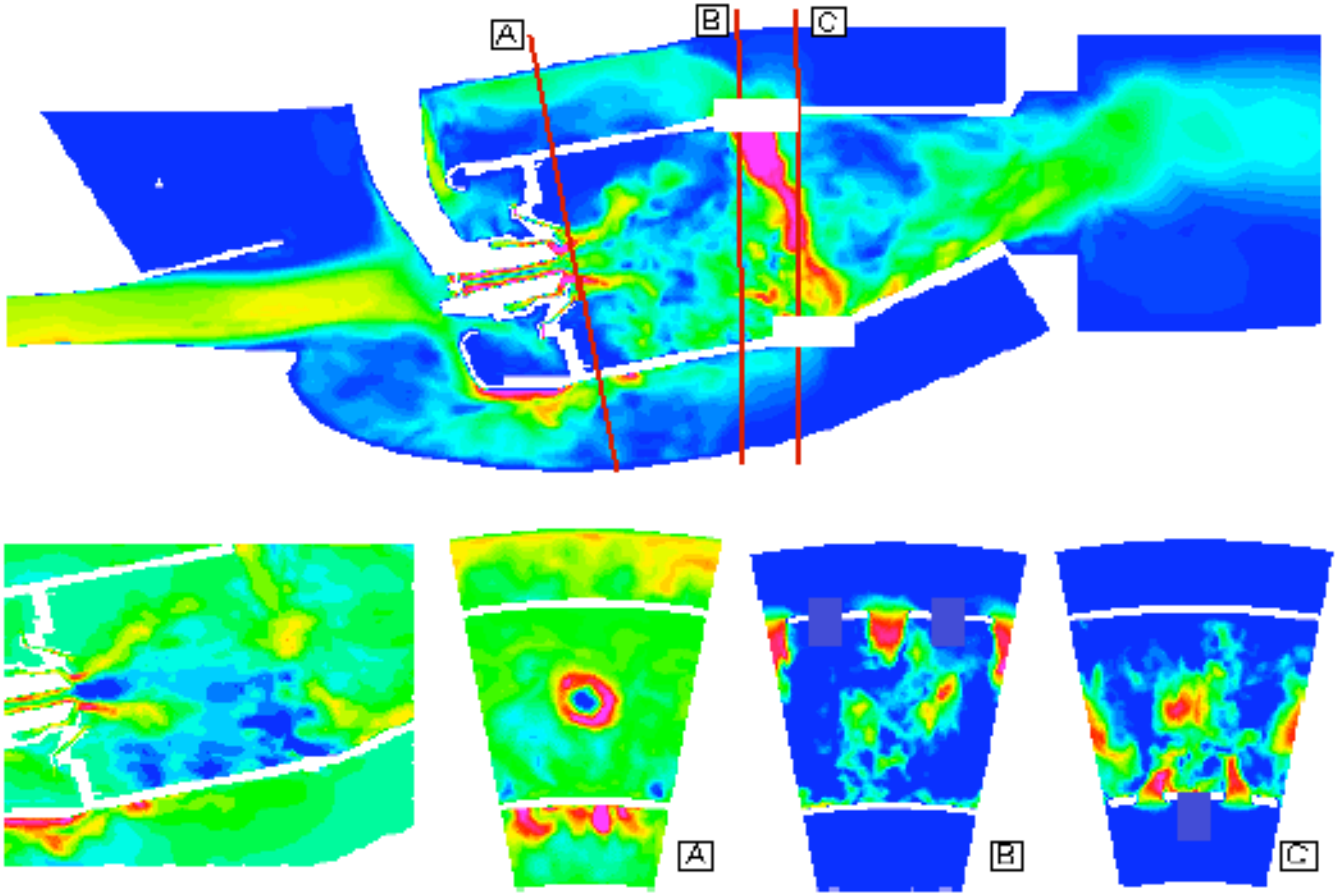
…it might require several thousand CPU hours of a dedicated supercomputers!!!

Stanford ASCI Team



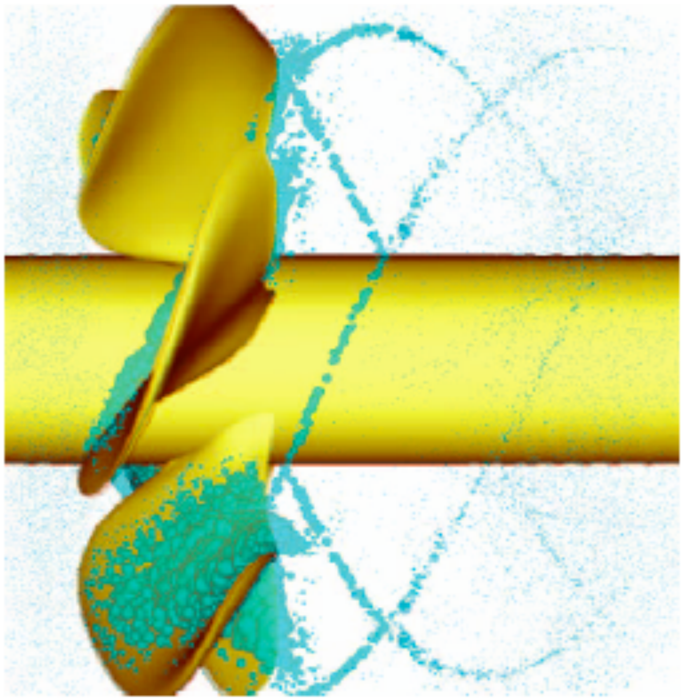
Animation by J. Schluter

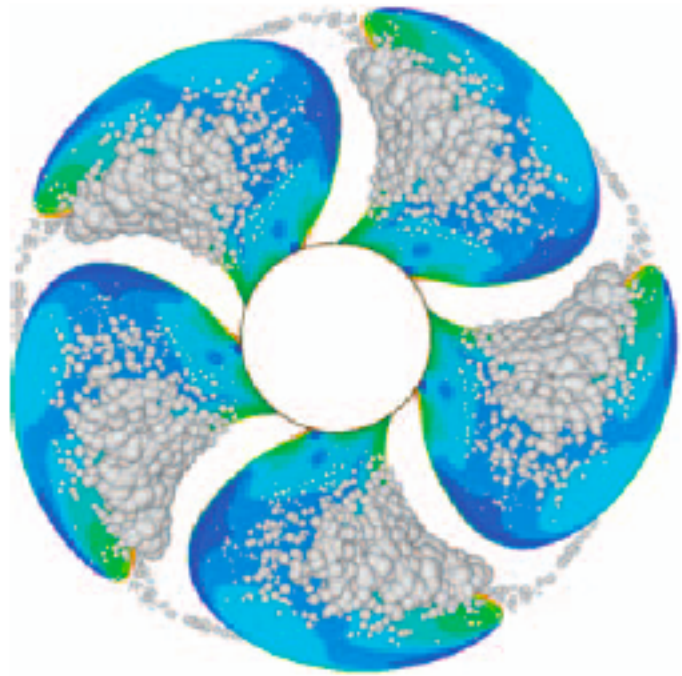
Analysis of the Results



Flow Features in Cross Sections of the Combustor

Bubble Dynamics and Cavitation

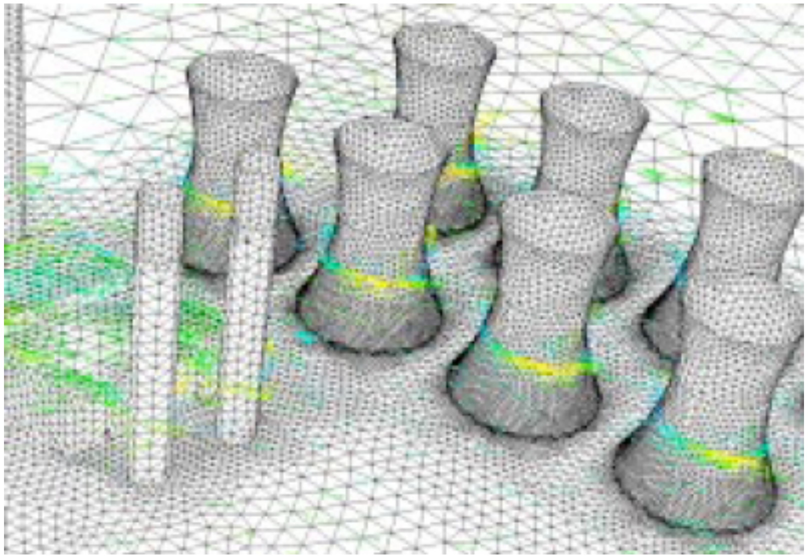




Multiphase, unsteady flow

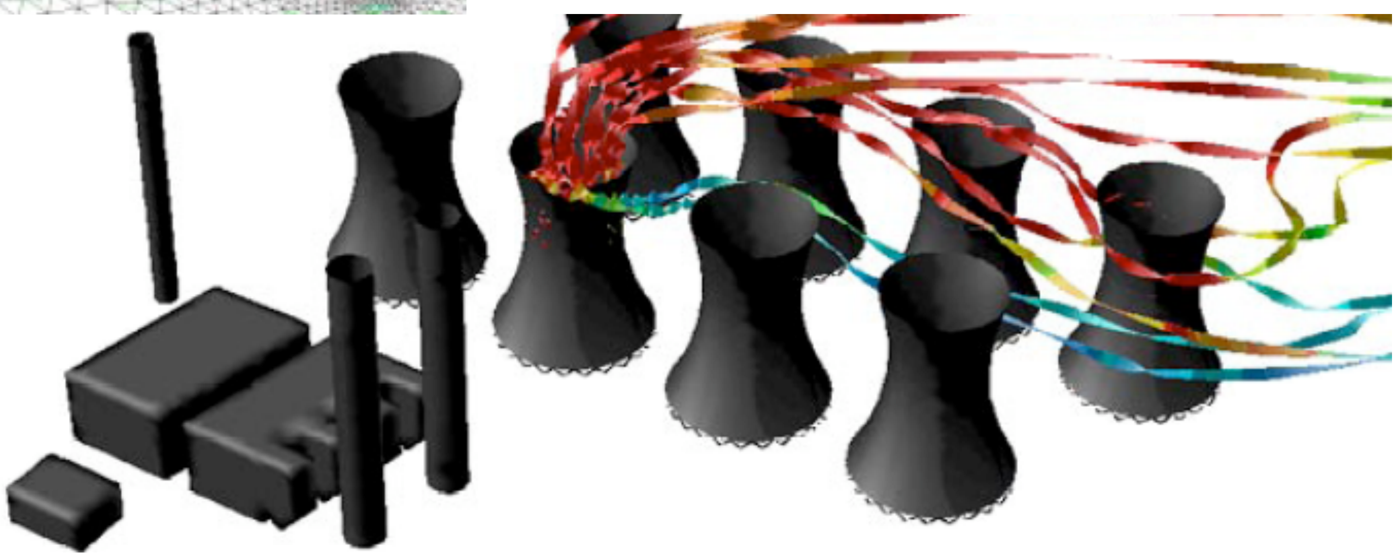
FLUENT ANSYS

Environmental Dispersion

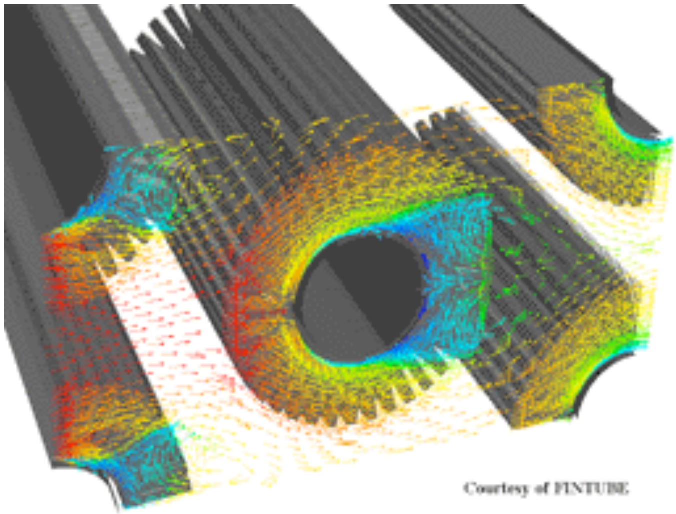


Multispecies, unsteady flow

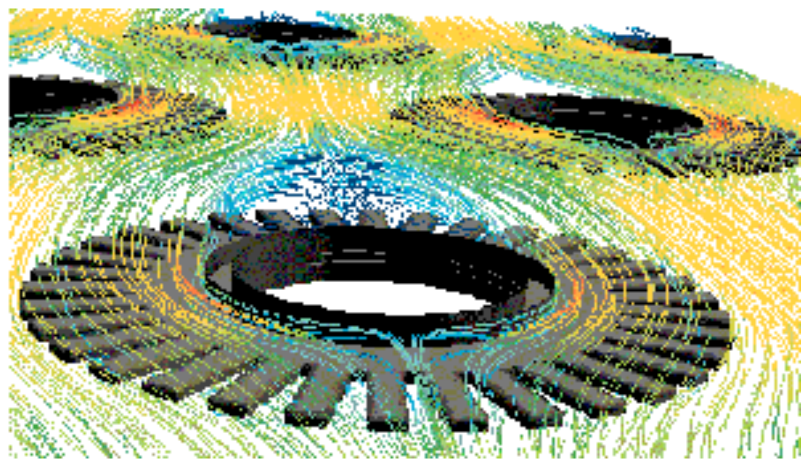
FLUENT
ANSYS



Heat Exchangers

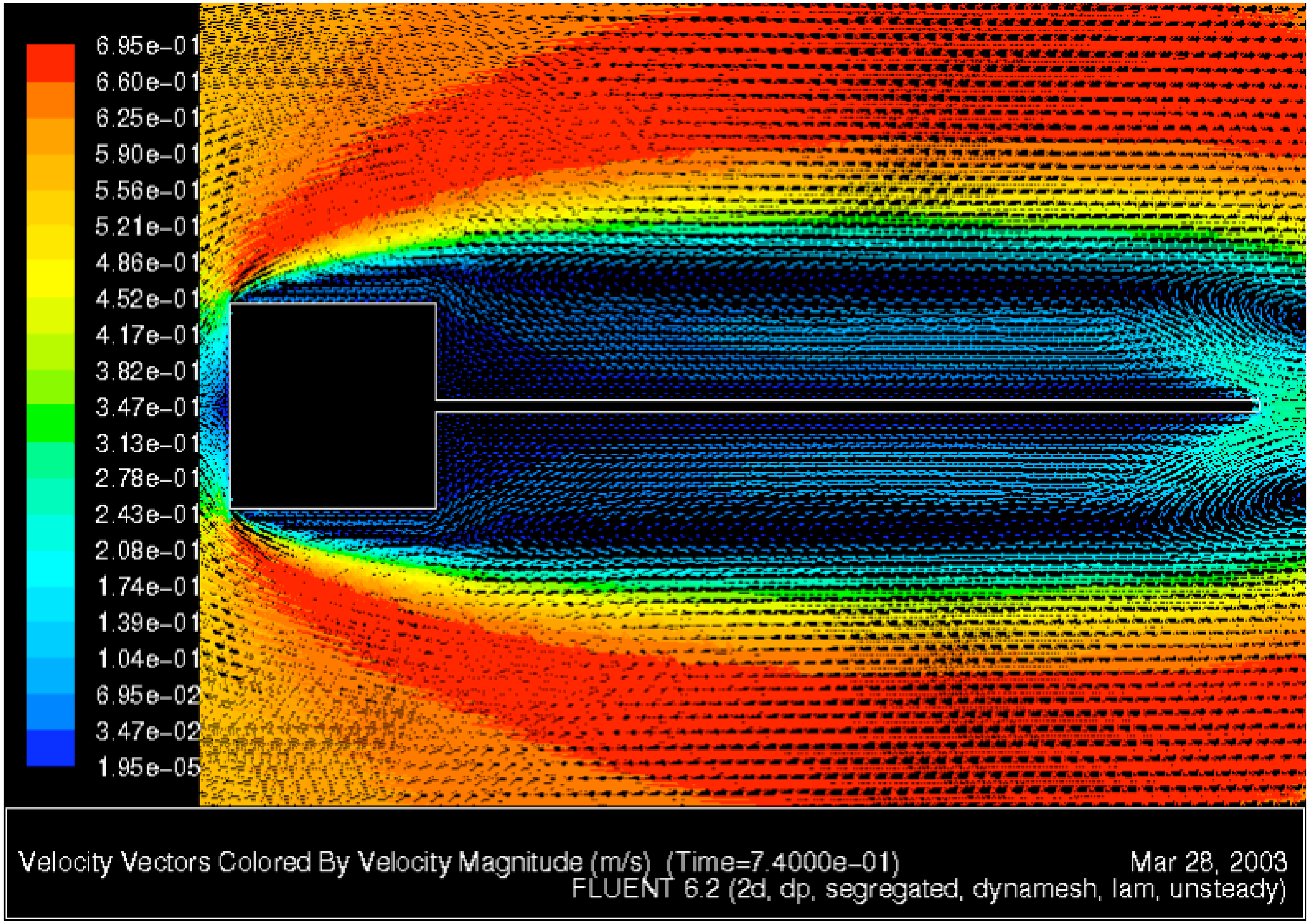


Flow and heat transfer



FLUENT
ANSYS

Fluid/Structure Interactions



Software Tools

SolidWorks	Cubit	CFX	Tecplot
ProE	ICEM	Phoenix	Ensight
AutoCAD	GridPro	StarCD	FieldView
…	…	…	…
	Gambit	Fluent	
Geometry Modeler	**Grid Generator**	**Flow Solver**	**Flow Analysis**

PreProcessing

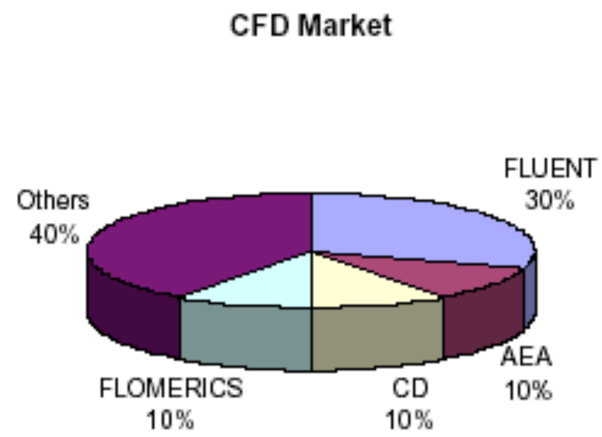
Post Processing

Software Tools

Several Codes with “similar” capabilities are in the market

FLUENT/GAMBIT are available

Use of other codes is welcome (CFX. StarCD, CFDRC have been used in the past)



Software Tools

- Other codes available at ME Stanford

  StarCD

  CFX

  PowerFlow

- Main differences will be outlined
- Results obtained with different codes will be presented

Software access is limited to FLUENT and GAMBIT to facilitate exchange of examples and tutorials

Introduction to CFD Commercial Codes

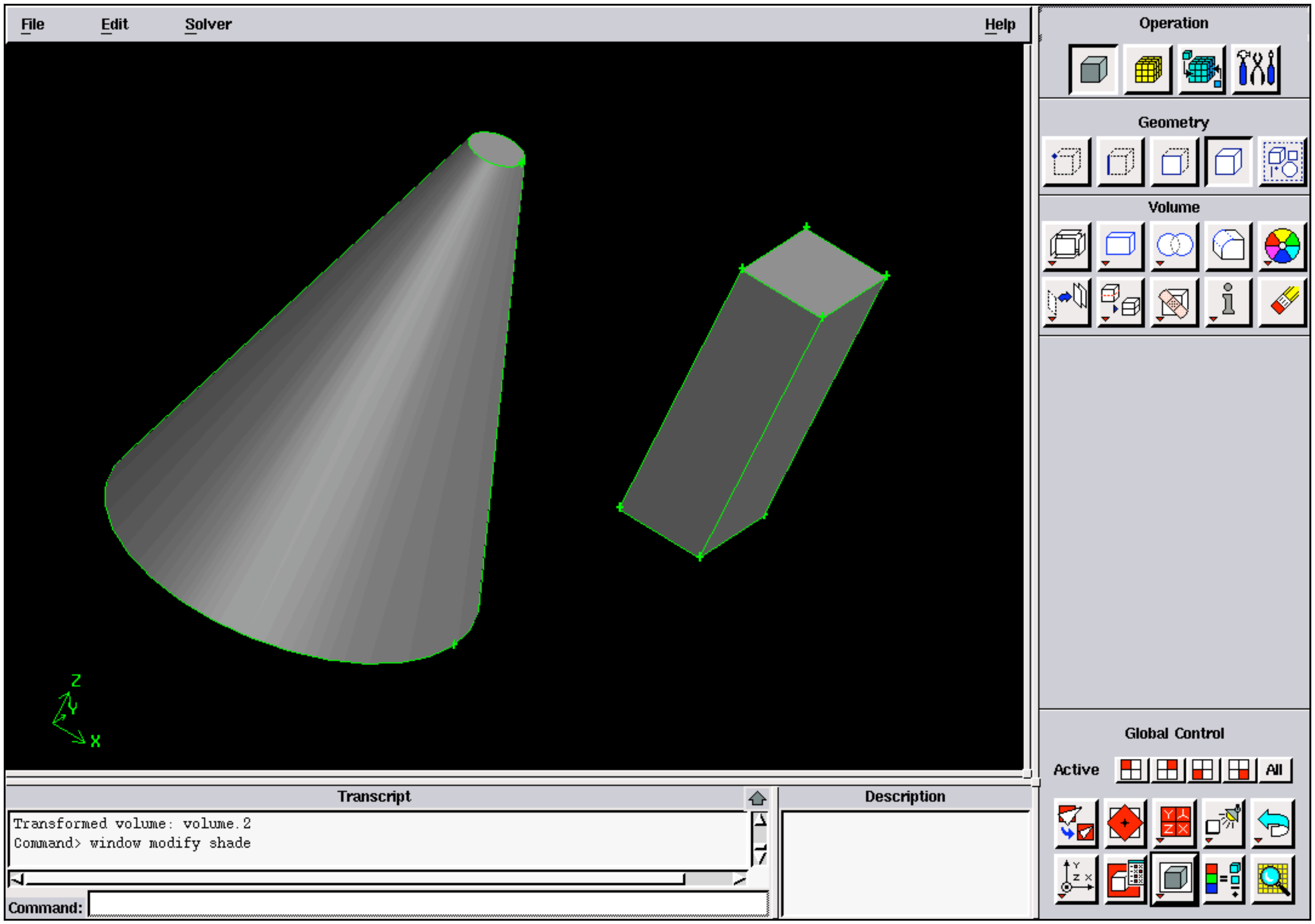
Grid generation: Gambit

- GUI based
- Geometry definition
- CAD geometry import (IGES, STL, etc.)
- Structured & unstructured meshes
- Viscous layers clustering
- Export grids in various format
- Journal files for automation

Grid Quality

- Metric measures
- Mesh improvement and smoothing

GAMBIT



GUI Snapshot

GAMBIT

`gambit -id <namefile>`

Interactive execution with GUI

`gambit -inp <journalfile>`

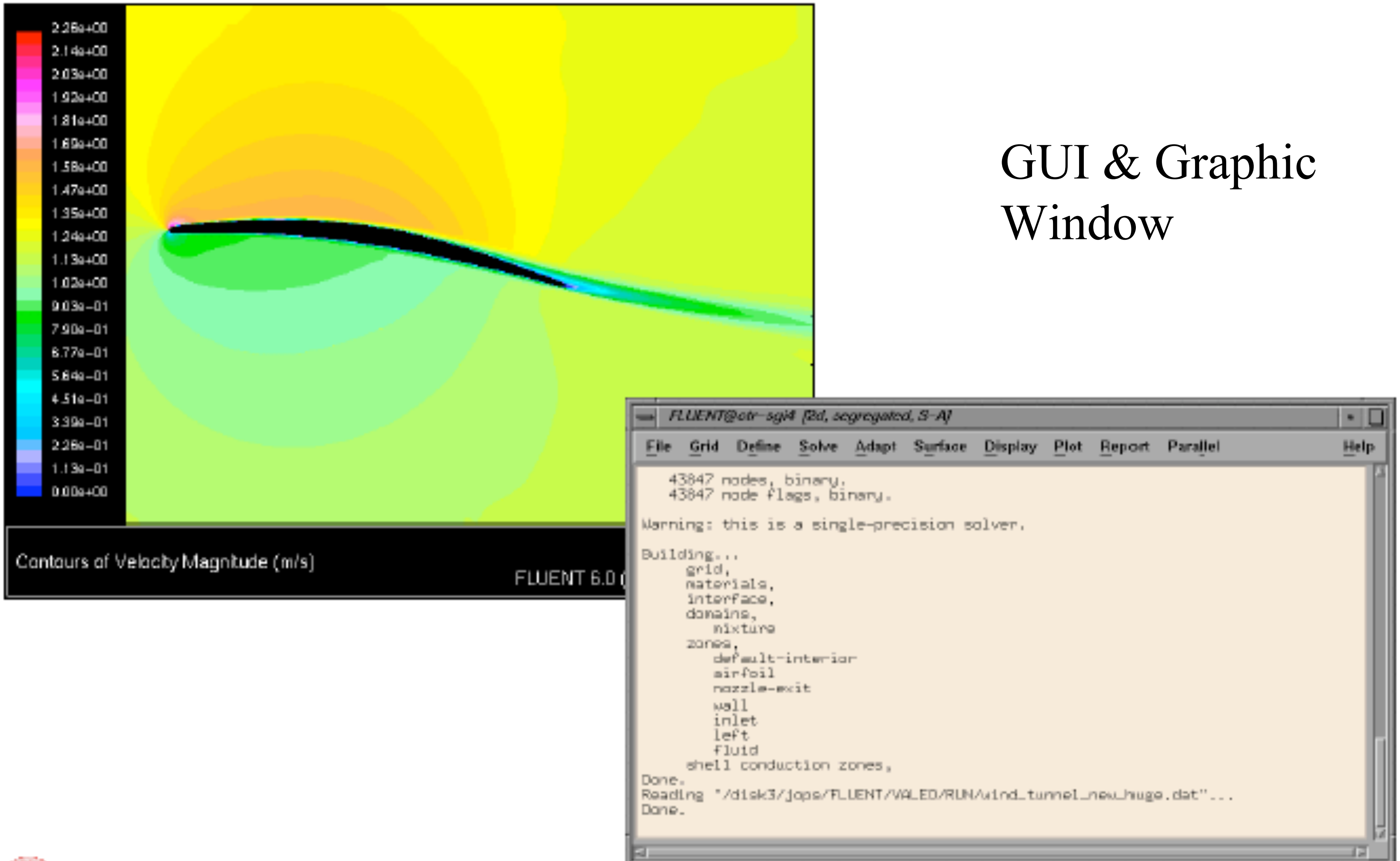
Batch execution without GUI

Geometry & Grid are saved in a *database* file (*.dbs)
The mesh is saved into a solver-dependent file (*.msh)

Introduction to CFD Commercial Codes

- Solver: Fluent
  - GUI based
  - Widely used
  - Several features (physical & numerical models)
  - User-Defined programming
  - Postprocessing capabilities
  - Export results in various format
  - Journal files for automation
- Grid adaptation
  - Mesh refinement schemes
  - Adaptation functions

FLUENT



GUI & Graphic Window

FLUENT

```
fluent 3d
```

Interactive execution with GUI

```
fluent 3d -g -i <journalfile>
```

Batch execution without GUI

Case set-up and boundary conditions are saved in a *case* file (*.cas)
The solution is saved in a *data* file (*.dat)
Selected data can be exported into other formats

Software/Computer Access

Accounts are available on the PCs in Bldg 500-RM 500N2
Fluent and Gambit are pre-installed and ready to go

Please contact Steve Jones (stevejones@stanford.edu) if you have already access to computational resources and want to use Fluent (mention that you are taking ME469B)

For anything else contact me directly:

jops@stanford.edu