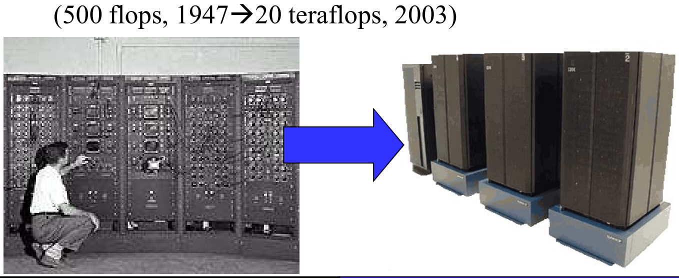
Chapter 1
Introduction

Ibrahim Sezai
Department of Mechanical Engineering
Eastern Mediterranean University

What is CFD?

- CFD is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)
- Historically only Analytical Fluid Dynamics (AFD) and Experimental Fluid Dynamics (EFD).
- CFD made possible by the advent of digital computer and advancing with improvements of computer resources

(500 flops, 1947→20 teraflops, 2003)



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

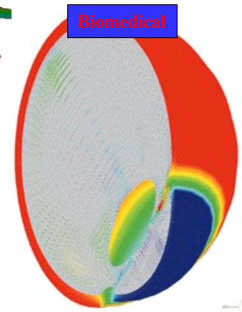
Where is CFD used?

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

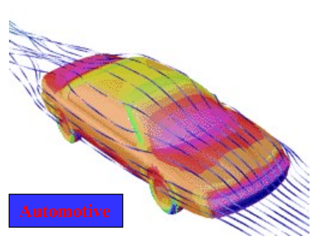
Aerospace

F18 Store Separation

Biomedical



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

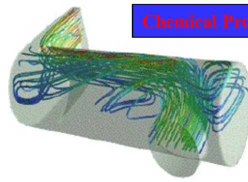


Automotive

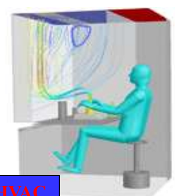
Where is CFD used?

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

Chemical Processing



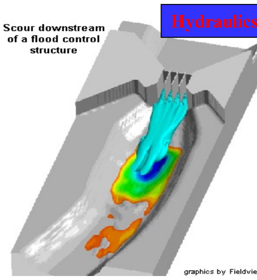
Polymerization reactor vessel - prediction of flow separation and residence time effects.



HVAC

Streamlines for workstation ventilation

Hydraulics

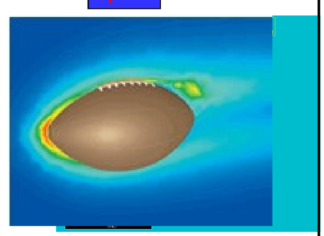


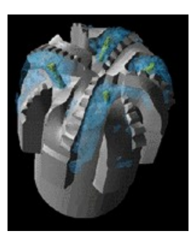
Where is CFD used?

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

Marine (movie)

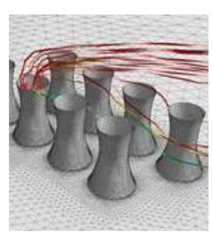
Sports





Oil & Gas

Flow of lubricating mud over drill bit



Power Generation

Flow around cooling towers

COMMERCIAL SOFTWARE

The market is currently dominated by four codes:

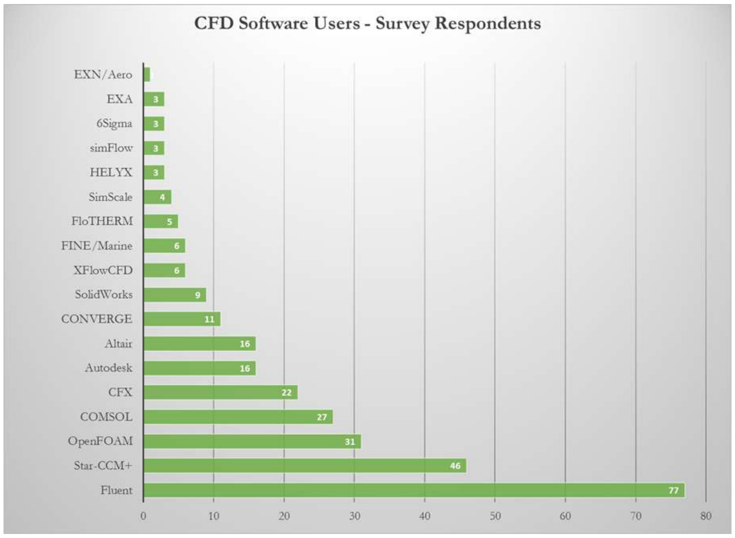
1) PHOENICS
2) FLUENT
3) FLOW3D
4) STAR-CD

Prices of the commercial software range between £10 000 and £ 50 000.

NON-COMMERCIAL SOFTWARE

- There are also non-commercial CFD software. One of the best non-commercial software is OpenFOAM.
- The OpenFOAM® (Open Field Operation and Manipulation) CFD Toolbox is a free, open source CFD software package.
- OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to two-phase flows, solid dynamics and electromagnetics.

Preference of CFD packages (2016)



https://www.resolvedanalytics.com/theflux/comparing-popular-cfd-software-packages

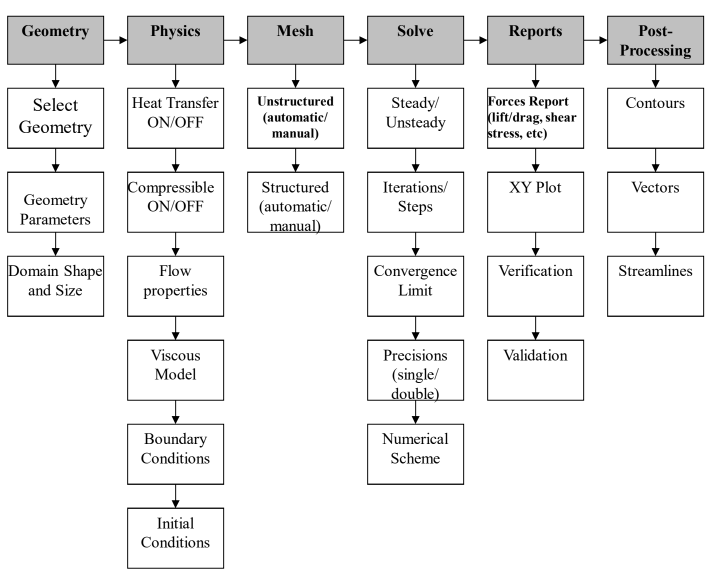
Advantages of CFD over EFD

- Substantial reduction of lead times and costs of new designs.
- Ability to study systems where controlled experiments are difficult or impossible to perform (e.g. very large systems).
- Ability to study systems under hazardous conditions at and beyond their normal performance limits (e.g. safety studies and accident scenarios).
- Practically unlimited level of detail of results.

CFD process

- Purposes of CFD codes will be different for different applications:
  - investigation of bubble-fluid interactions for bubbly flows,
  - study of wave induced massively separated flows for free-surface, etc.
- Depending on the specific purpose and flow conditions of the problem, different CFD codes can be chosen for different applications (aerospace, marines, combustion, multi-phase flows, etc.)
- Once purposes and CFD codes chosen, “CFD process” is the steps to set up the IBVP problem and run the code:

How does a CFD code work?



All commercial codes contain three basic elements:

1. Pre-processor
2. Solver
3. Post-processor

1) Pre-processor

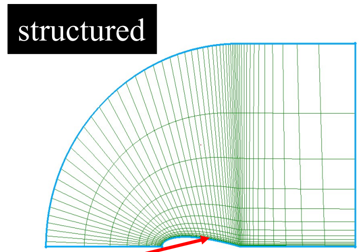
Provides the input of the problem and transforms this input in a form suitable for use by the solver. Preprocessing involve:

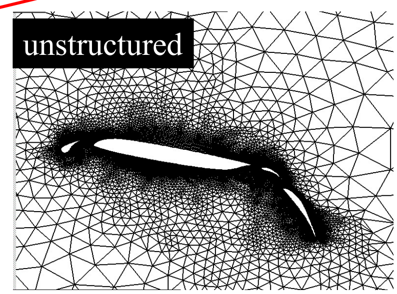
Definition of the geometry of the region of interest: the computational domain.

- Selection of an appropriate coordinate
- Determine the domain size and shape
- Any simplifications needed?
- What kinds of shapes needed to be used to best resolve the geometry? (lines, circular, ovals, etc.)
- For commercial code, geometry is usually created using commercial software (either separated from the commercial code itself, like Gambit, or combined together, like FlowLab)
- One of the commercial geometry and mesh modeller is Gridgen.

b) Grid Generation

- Grids can either be structured (hexahedral) or unstructured (tetrahedral). Depends upon type of discretization scheme and application
  - Scheme
    - ❑ Finite differences: structured
    - ❑ Finite volume or finite element: structured or unstructured
  - Application
    - ❑ Thin boundary layers best resolved with highly-stretched structured grids
    - ❑ Unstructured grids useful for complex geometries
    - ❑ Unstructured grids permit automatic adaptive refinement based on the pressure gradient, or regions interested (FLUENT)





2) SOLVER

There are three basic numerical solution techniques:

a) Finite difference methods

b) Finite element methods

c) Spectral methods

All of these numerical methods perform the following steps:

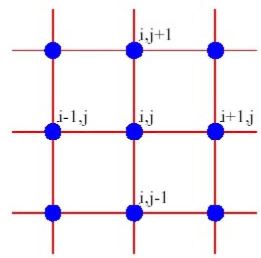
- Approximation of the unknown flow variables by means of simple functions.
- Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

a) Finite Difference Method

- The unknowns ϕ at each grid point are approximated by using Taylor series expansion of the derivatives of ϕ

- Discretise the governing **differential equations directly; e.g.**

$$0 = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \approx \frac{u_{i+1,j} - u_{i-1,j}}{2\Delta x} + \frac{v_{i,j+1} - v_{i,j-1}}{2\Delta y}$$



b) Finite Element Method

Use simple, piecewise functions valid on the elements to describe the local variations of unknown flow variables ϕ

$$\phi(x) = \sum \phi_\alpha S_\alpha(x)$$

where S_α is the shape function.

The finite element method is popular in solid mechanics.

c) Spectral Methods

Approximate the unknowns by means of

Fourier Series

Or series of Chebysev polynomials.

Finite Volume Method

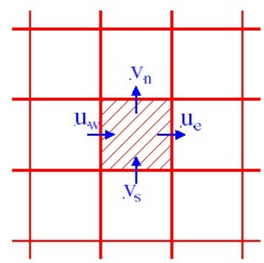
- is a special form of the finite difference methods.
- 4 of the 5 commercially available CFD codes use this method.

Discretise the governing **integral equations directly; e.g.**

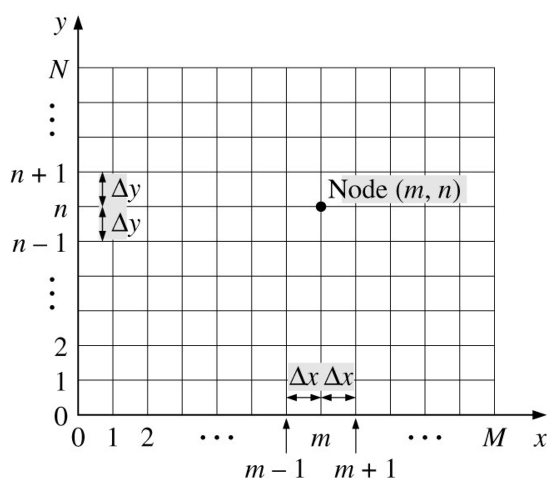
Net mass flow =

$$(\rho uA)_e - (\rho uA)_w + (\rho vA)_n - (\rho vA)_s = 0$$

The **finite-volume** method is popular in **fluid mechanics** because:
· it rigorously enforces **conservation**;
· it is **flexible** in terms of both **geometry** and the variety of **fluid phenomena**;
· it is directly relatable to **physical quantities** (mass flux, etc.).



- A mesh is generated in the solution domain



The conservation of a general flow variable ϕ, within a control volume can be expressed as

$$\begin{bmatrix}\text{Rate of change}\\ \text{of } \phi \text{ in the control}\\ \text{volume with}\\ \text{respect to time}\end{bmatrix} = \begin{bmatrix}\text{Net flux of}\\ \phi \text{ due to}\\ \text{convection into}\\ \text{the control volume}\end{bmatrix} + \begin{bmatrix}\text{Net flux of}\\ \phi \text{ due to}\\ \text{diffusion into the}\\ \text{control volume}\end{bmatrix} + \begin{bmatrix}\text{Net rate of creation}\\ \text{of } \phi \text{ inside the}\\ \text{control volume}\end{bmatrix}$$

For a 2D flow, the above relationship can be expressed mathematically as

$$\frac{\partial}{\partial t}(\rho\phi)+\frac{\partial}{\partial x}(\rho u\phi)+\frac{\partial}{\partial y}(\rho v\phi)=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial\phi}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma\frac{\partial\phi}{\partial y}\right)+S_\phi \qquad (1\text{-}1)$$

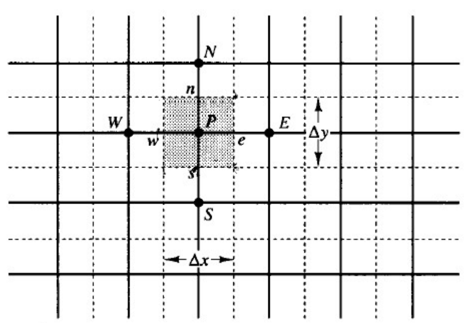
where Γ = diffusion coefficient

S_ϕ = Source term

ϕ = general flow variable (u, v, w, T, …etc)

- The discretized form of the conservation equation (1-1) for a 2-dimensional control volume is of the form

$$a_P\phi_P = a_W\phi_W + a_E\phi_E + a_S\phi_S + a_N\phi_N + S_u$$



- A similar equation is written for each control volume.
- At the end, a system of linear algebraic equations is obtained.
- The system of equations are expressed in matrix form as

$$[\mathbf{A}][\phi] = [\mathbf{b}]$$

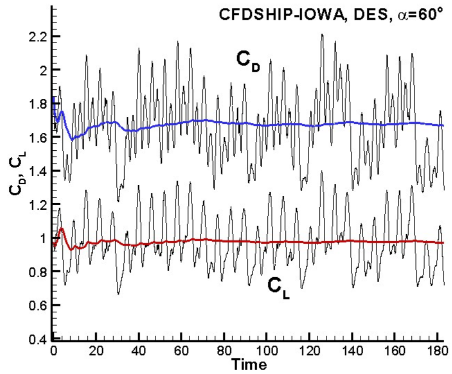
- Use any matrix solution method to solve the system of equations for the unknown variable ϕ.
- An iterative solution approach is used. Most popular is the TDMA line-by-line solver for the set of algebraic equations.

3) Post Processor

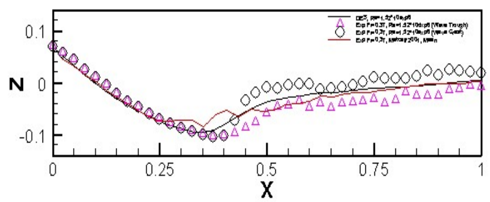
The leading CFD packages are now equipped with versatile data visualization tools. These include:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc)
- Color postscript output.

Post-Processing (visualization, XY plots)

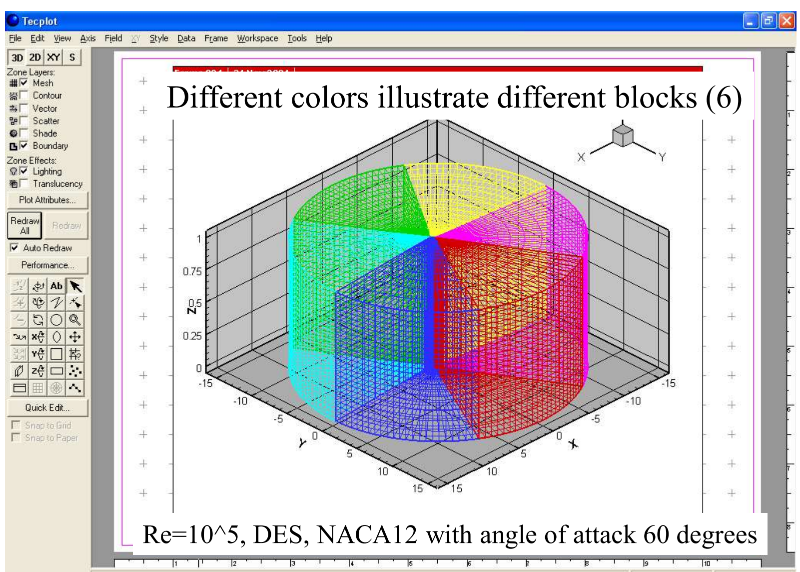


Lift and drag coefficients of NACA12 with 60° angle of attack (CFDSHIP-IOWA, DES)



Wave profile of surface-piercing NACA24, Re=1.52e6, Fr=0.37 (CFDSHIP-IOWA, DES)

Post-Processing (visualization, Tecplot)

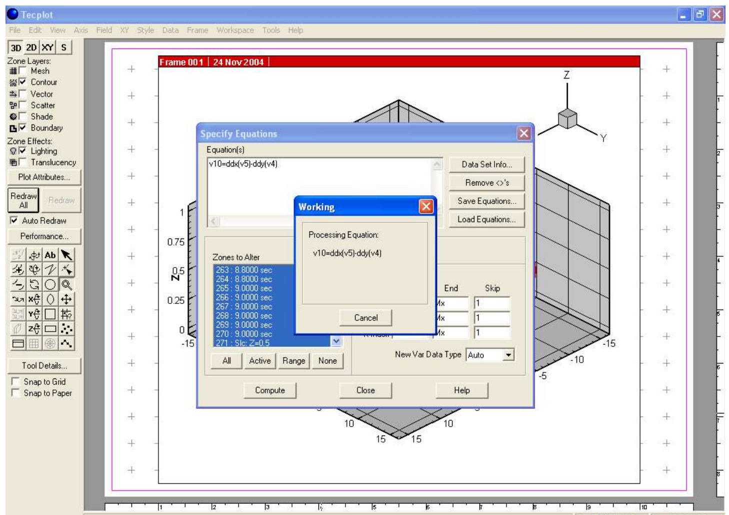


Different colors illustrate different blocks (6)

Re=10^5, DES, NACA12 with angle of attack 60 degrees

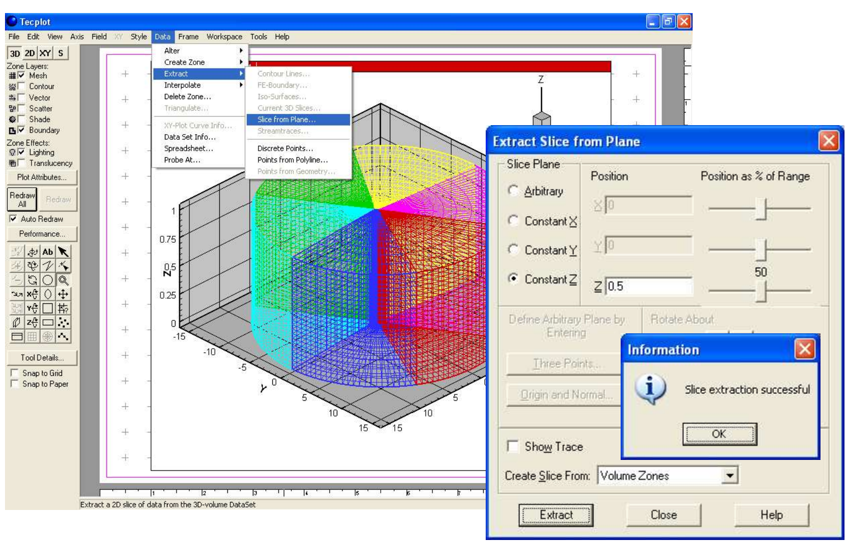
Post-Processing (NACA12, 2D contour plots, vorticity)

- Define and compute new variable: “Data”→”Alter”→”Specify equations”→”vorticity in x,y plane: v10”→”compute”→”OK”.



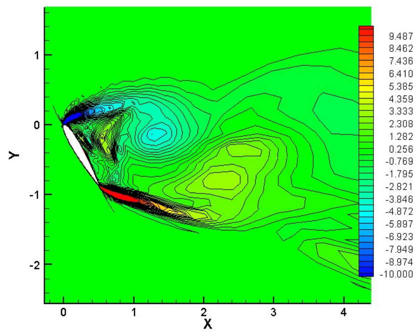
Post-Processing (NACA12, 2D contour plot)

- Extract 2D slice from 3D geometry: “Data”→”Extract”→”Slice from plane”→”z=0.5”→”extract”

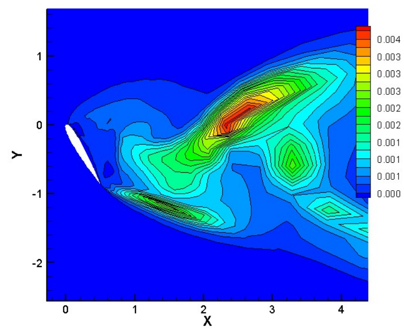


Post-Processing (NACA12, 2D contour plots)

- 2D contour plots on z=0.5 plane (vorticity and eddy viscosity)



Vorticity ω_z

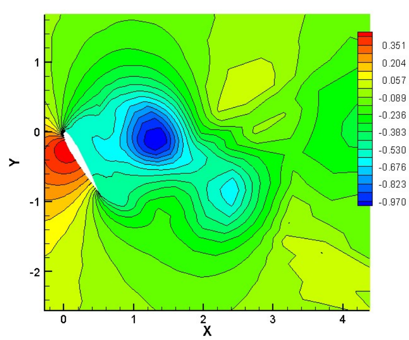


Eddy viscosity

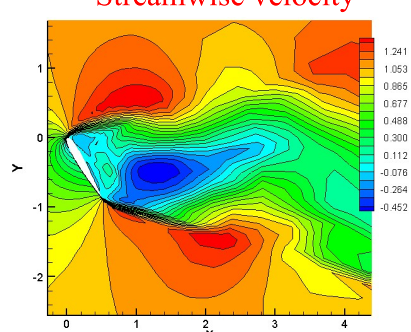
Post-Processing (NACA12, 2D contour plots)

- 2D contour plots on z=0.5 plane (pressure and streamwise velocity)

Pressure

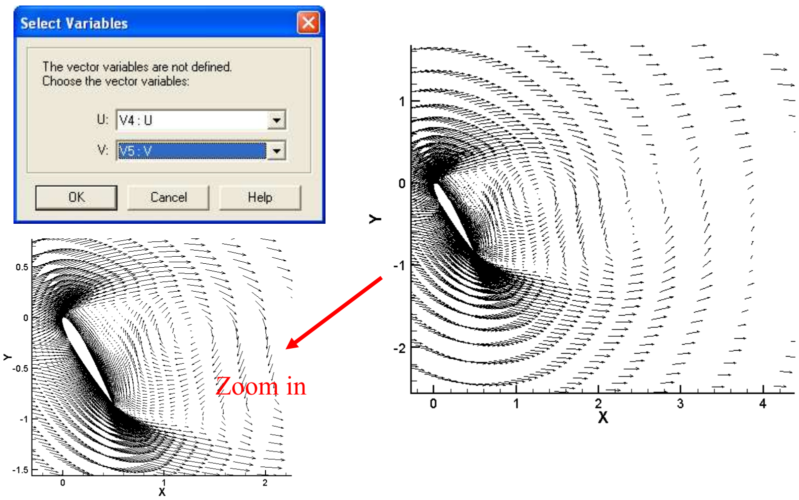


Streamwise velocity



Post-Processing (2D velocity vectors)

- 2D velocity vectors on z=0.5 plane: turn off “contour” and activate “vector”, specify the vector variables.



Post-Processing (3D Iso-surface plots, cont’d)

- 3D Iso-surface plots: pressure, p=constant
- 3D Iso-surface plots: vorticity magnitude

$$\Omega = \sqrt{\omega_x^2 + \omega_y^2 + \omega_z^2}$$

- 3D Iso-surface plots: λ_2 criterion

Second eigenvalue of $\frac{1}{2\rho}\nabla^2 p$

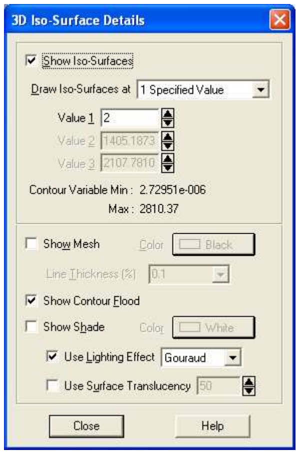
- 3D Iso-surface plots: Q criterion

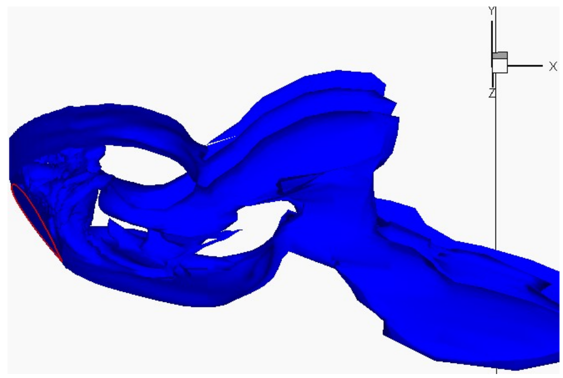
$$Q = \frac{1}{2}\left(\Omega_{ij}\Omega_{ij} - S_{ij}S_{ij}\right) \qquad \Omega_{ij} = \left(u_{i,j} - u_{j,i}\right)/2$$

$$S_{ij} = \left(u_{i,j} + u_{j,i}\right)/2$$

Post-Processing (3D Iso-surface plots)

- 3D Iso-surface plots: used to define the coherent vortical structures, including pressure, voriticity magnitude, Q criterion, λ_2, etc.

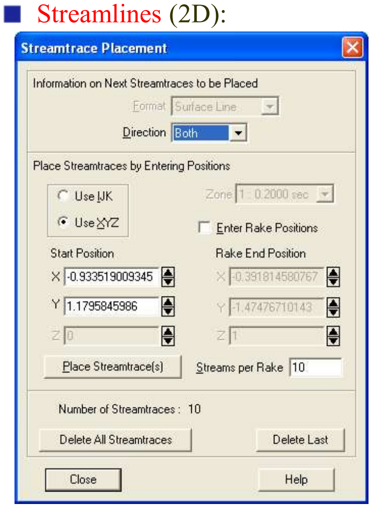


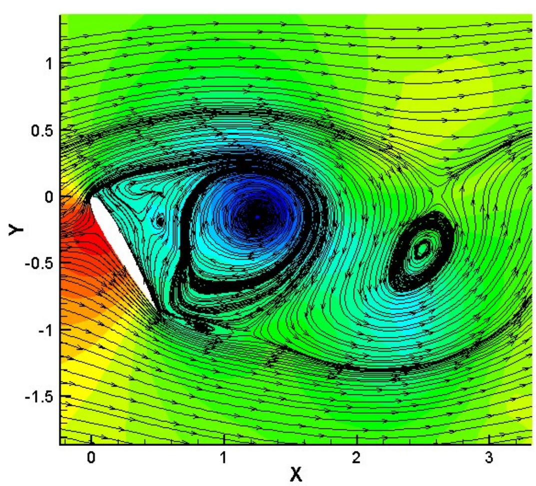


Iso-surface of vorticity magnitude

Post-Processing (streamlines)

- Streamlines (2D):



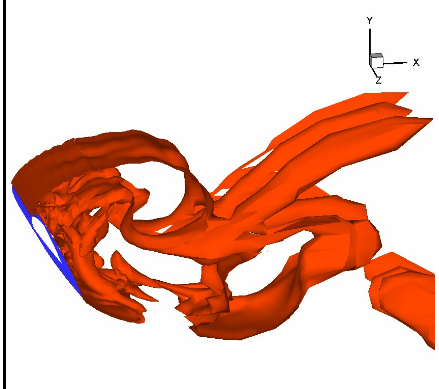


Streamlines with contour of pressure

- Streaklines and pathlines (not shown here)

Post-Processing (Animations)

- Animations (3D): animations can be created by saving CFD solutions with or without skipping certain number of time steps and playing the saved frames in a continuous sequence.
- Animations are important tools to study time-dependent developments of vortical/turbulent structures and their interactions

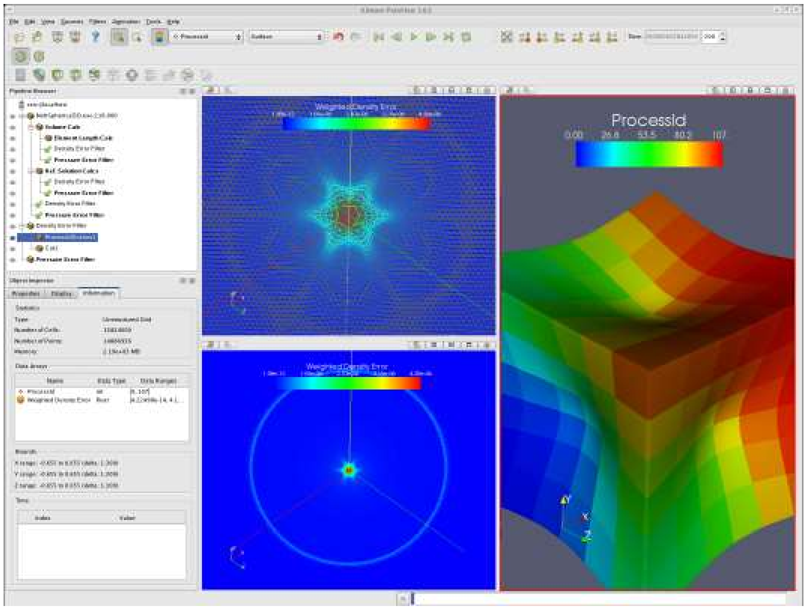


Q=0.4

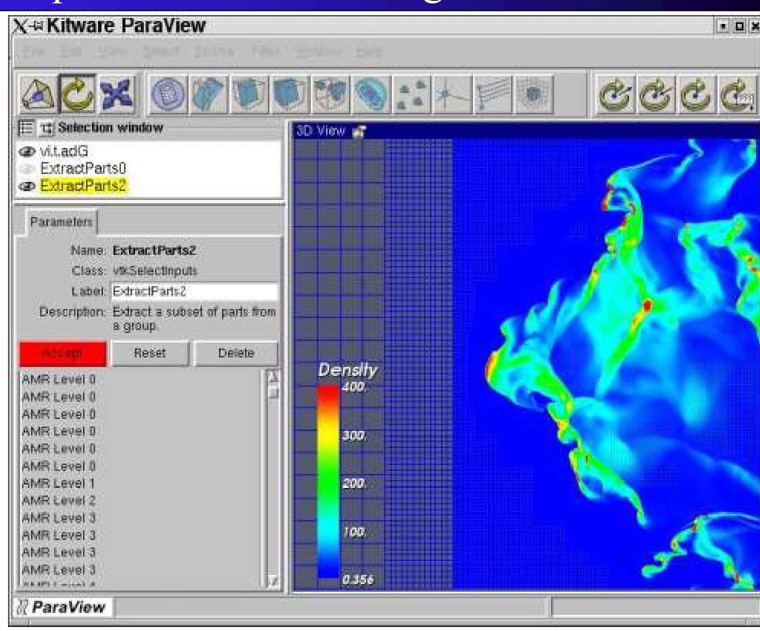
ParaView Post Processor

- **ParaView** is an open-source, multi-platform data analysis and visualization tool for CFD results.
- ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques.
- The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities.

Examples of ParaView Images



Examples of ParaView Images



Problem Solving with CFD

- The results of a CFD code are:
  - at best as good as the physics embedded in it.
  - at worst as good as its operator.
- Three mathematical concepts are useful in determining the success of CFD codes:

1) Convergence:

   - is a property of a numerical method to produce a solution which approaches the exact solution as the grid spacing is reduced to zero.

2) Consistency:

   - consistent numerical schemes produce systems of algebraic equations which are equivalent to the original governing equations as the grid spacing tend to zero.

3) Stability:

   - is associated with damping of errors as the numerical method proceeds.

- A CFD code should also have the following properties:
- **Conservativeness:**
  - Conservation of a fluid property ϕ for each control volume.
  - A numerical scheme which possesses the conservativeness property also ensure global conservation of the fluid property over the entire geometry.
  - Is achieved by means of consistent fluxes of ϕ through the cell faces of adjacent control volumes.
  - The finite volume approach guarantees conservativeness.

- **Boundedness:**
  - is crucial for stability and requires that in a linear problem without sources the solution is bounded by the maximum and minimum boundary values of the flow variable.
- **Transportiveness**
  - is a property that accounts for the directional property of convection terms.
  - in convection phenomena, a point only experiences effects due to changes at upstream locations.
  - a finite volume scheme should consider the relative strength of diffusion to convection.

Free Fortran compilers:

- GNU Fortran
- G95

ANSYS Free Student Software Downloads

- Student version of ANSYS products can be downloaded freely.
- Download ANSYS student 19.2
- Limited by 512K cells/nodes.
- License duration: Renewable, twelve-month lease.
- Download time: 55 mins for 10 Mbps, 15 mins for 51 Mbps
- We will use mainly ANSYS CFX