Computational Fluid Dynamics

Flow Fields

Important variables

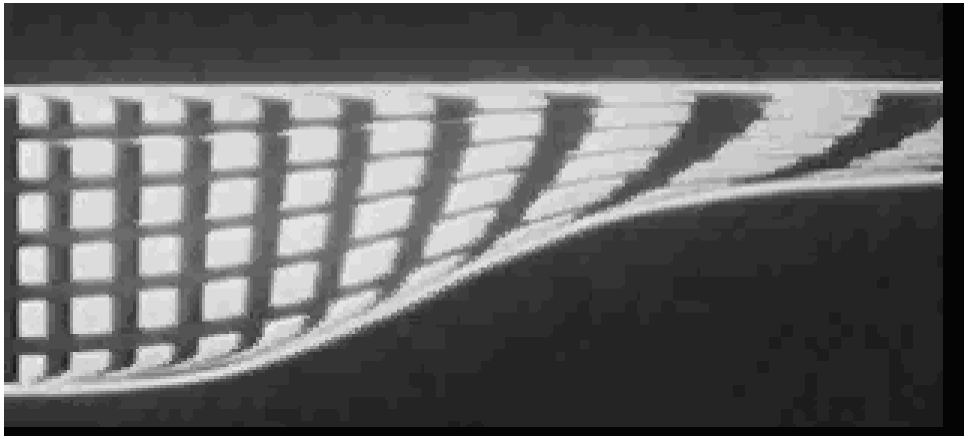
- Pressure and fluid velocities are always calculated in conjunction. Pressure can be used to calculate forces on objects, e.g. for the prediction of drag of a car. Fluid velocities can be visualized to show flow structures.
- From the flow field we can derive other variables such as shear and vorticity. Shear stresses may relate to erosion of solid surfaces. Deformation of fluid elements is important in mixing processes. Vorticity describes the rotation of fluid elements.
- In turbulent flows, turbulent kinetic energy and dissipation rate are important for such processes as heat transfer and mass transfer in boundary layers.
- For non-isothermal flows, the temperature field is important. This may govern evaporation, combustion, and other processes.
- In some processes, radiation is important.

Post-processing

- Results are usually reviewed in one of two ways. Graphically or alphanumerically.
- Graphically:
  - Vector plots.
  - Contours.
  - Iso-surfaces.
  - Flowlines.
  - Animation.
- Alphanumerics:
  - Integral values.
  - Drag, lift, torque calculations.
  - Averages, standard deviations.
  - Minima, maxima.
  - Compare with experimental data.

Fluid motion

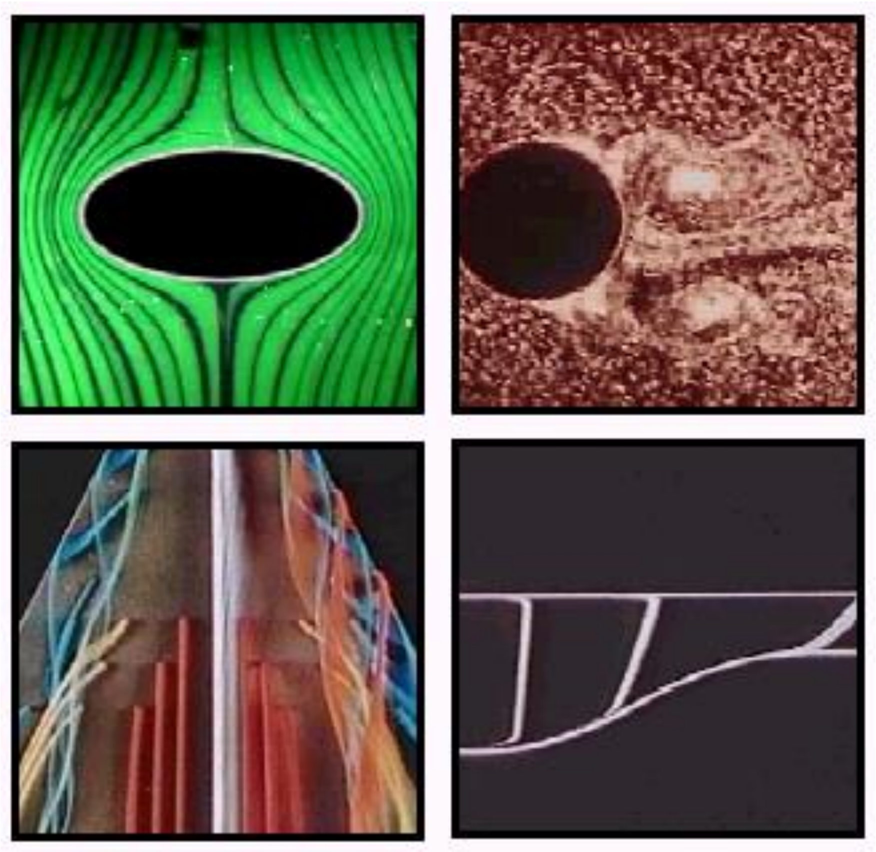
- In a fluid flow field, each fluid element undergoes three different effects:
  - 1. Translation.
  - 2. Deformation.
  - 3. Rotation.



Translation and deformation

Methods to show translation

- Translation can be shown by means of:
  - Velocity vectors.
  - Flowlines:
    - Streamlines.
    - Pathlines.
    - Streaklines.
    - Timelines.
    - Oilflow lines.



Pressure

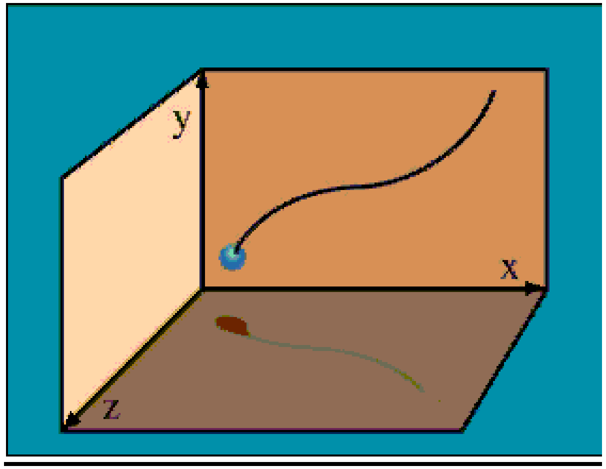
- Pressure can be used to calculate forces (e.g. drag, lift, or torque) on objects by integrating the pressure over the surface of the object.
- Pressure consists of three components:
  - Hydrostatic pressure ρgh.
  - Dynamic pressure $\rho v^2/2$.
  - Static pressure p_s. This can be further split into an operating pressure (e.g. atmospheric pressure) and a gauge pressure.
- When static pressure is reported it is usually the gauge pressure only.
- Total pressure is the static pressure plus the dynamic pressure.

Streamlines

- Streamlines are curves that are everywhere tangent to the velocity vector U.
- The animation shows streamlines for a steady state 3-D flow.
- For 3-D flow fields, instead of streamlines one usually visualizes streaklines or pathlines, which for steady flow are the same.
- For 2-D flow fields, a stream function Ψ can be defined:

$$u = \frac{\partial \psi}{\partial y}; \qquad v = -\frac{\partial \psi}{\partial x}$$

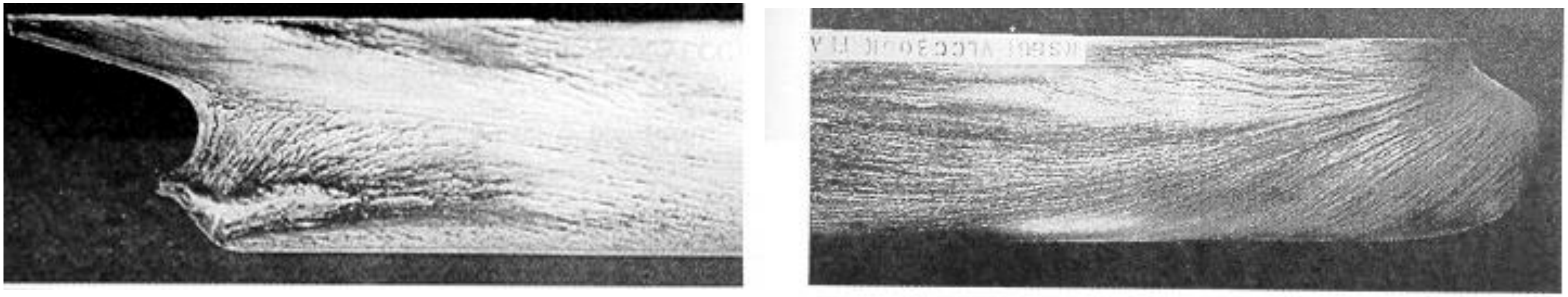
- In 2-D, lines of constant stream function are streamlines. Calculating the stream function and isolines is a more efficient way to calculate streamlines than by integrating particle tracks.



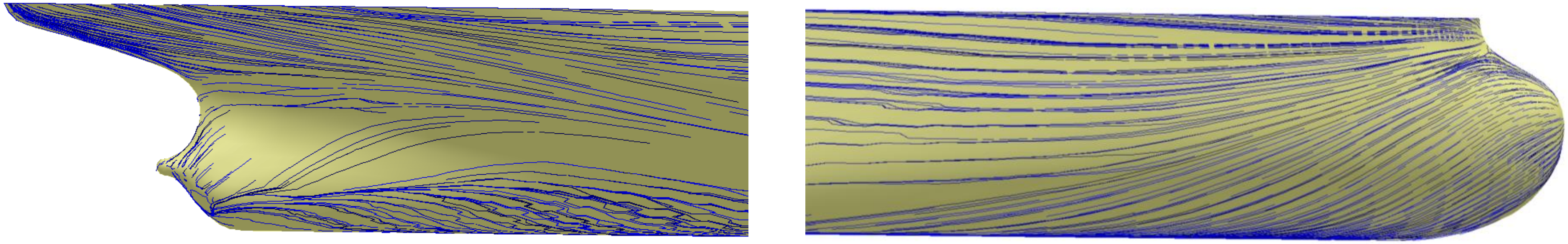
Ship hull surface flow visualization

Stern

Bow



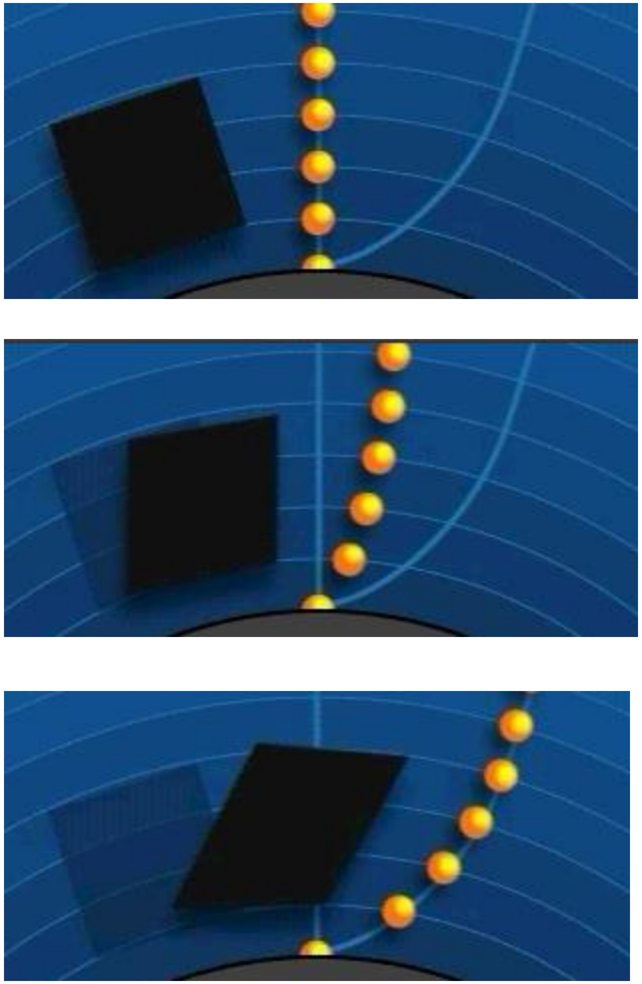
Experimental oil visualization (Van et al., 1998)



Numerical flow visualization

Deformation illustration

- In an incompressible flow field, a fluid parcel may become distorted, but it retains its original volume.
- The divergence of the velocity field is zero: div u = 0. This is the continuity equation.
- De deformation is governed by the rate of strain tensor.



Strain rate

- The deformation rate tensor appears in the momentum conservation equations.
- It is common to report the strain rate S(1/s), which is based on the Euclidian norm of the deformation tensor:

$$S = \sqrt{2 S_{ij} S_{ij}}$$

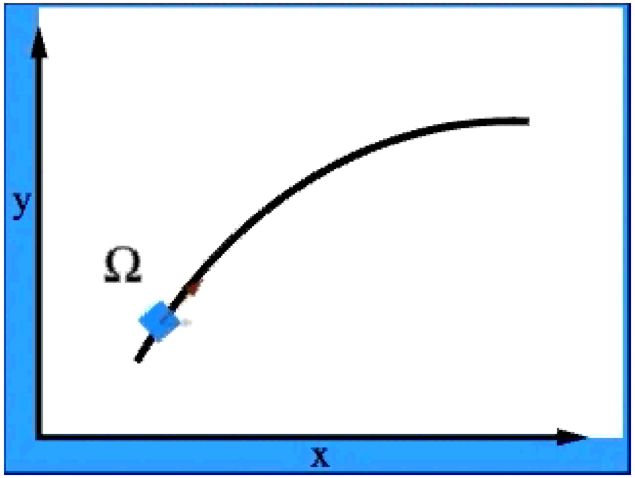
- The strain rate may also be called the shear rate.
- The strain rate may be used for various other calculations:
  - For non-Newtonian fluids, the viscosity depends on the strain rate.
  - In emulsions, droplet size may depend on the strain rate.
  - The strain rate may affect particle formation and agglomeration in pharmaceutical applications.

Vorticity

- As discussed, the motion of each fluid element can be described as the sum of a translation, rotation, and deformation.
- The animation shows a translation and a rotation.
- Vorticity is a measure of the degree of local rotation in the fluid. This is a vector. Unit is 1/s.
- For a 2-D flow this vector is always normal to the flow field plane.
- For 2-D flows, vorticity is then usually reported as the scalar:

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$$

- For 2-D flows, a positive vorticity indicates a counterclockwise rotation and a negative vorticity a clockwise rotation.



Vorticity - 3-D

Three – dimensional velocity vector : $\mathbf{u} = (u, v, w)$

Definition of vorticity: $\boldsymbol{\omega} = \nabla \times \mathbf{u} = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}, \; \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}, \; \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)$

Relationship between vorticity and angular velocity of a fluid element : $\boldsymbol{\omega} = \nabla \times \mathbf{u} = 2\,\boldsymbol{\Omega}$

Vorticity magnitude is calculated using the norm : $\omega\,(1/s) = |\boldsymbol{\omega}| = \sqrt{\omega_x^2 + \omega_y^2 + \omega_z^2}$

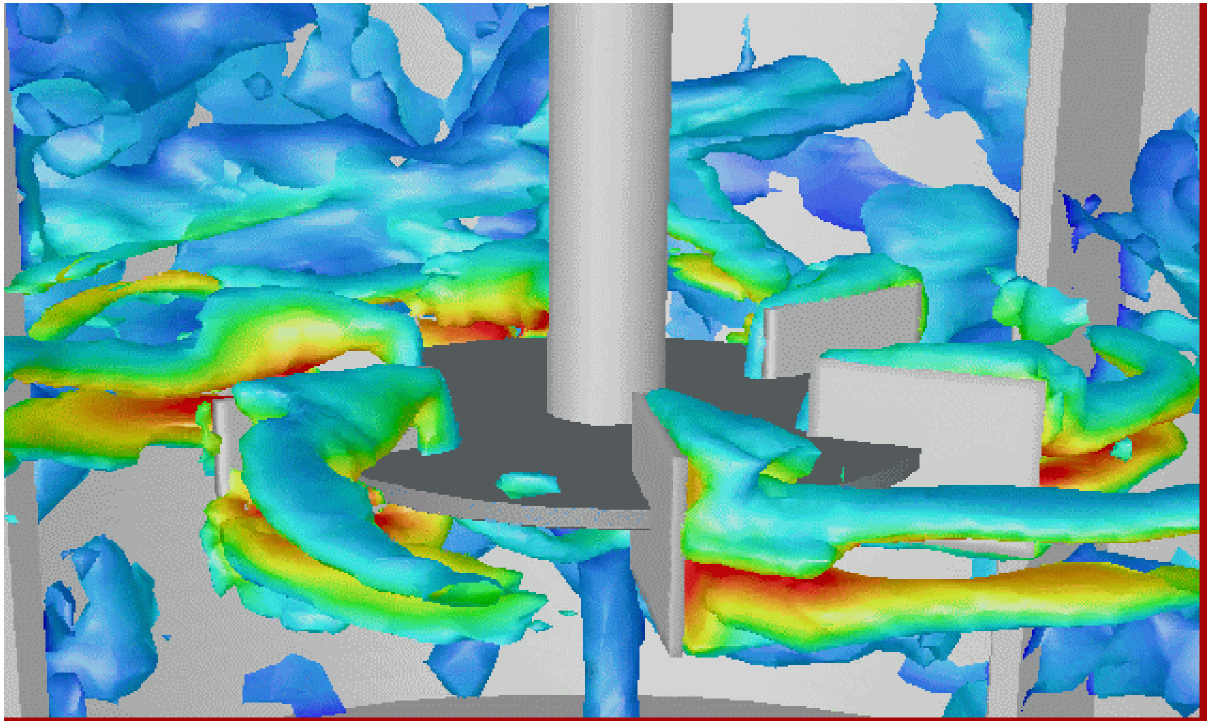
Vortexlines and helicity

- Iso-surfaces of vorticity can be used to show vortices in the flow field.
- Vortex lines are lines that are everywhere parallel to the vorticity vector.
- Vortex cores are lines that are both streamlines and vortexlines.
- The helicity H is the dot product of the vorticity and velocity vectors: $H = \boldsymbol{\omega} \cdot \mathbf{U}$
- It provides insight into how the vorticity vector and the velocity vector are aligned. The angle between the vorticity vector and the velocity vector (which is 0º or 180º in a vortex core) is given by:

$$\alpha = \cos^{-1}(H/(|\boldsymbol{\omega}||\mathbf{U}|))$$

- Algorithms exist that use helicity to automatically find vortex cores. In practice this only works on very fine grids with deeply converged solutions.

Isosurfaces of vorticity magnitude

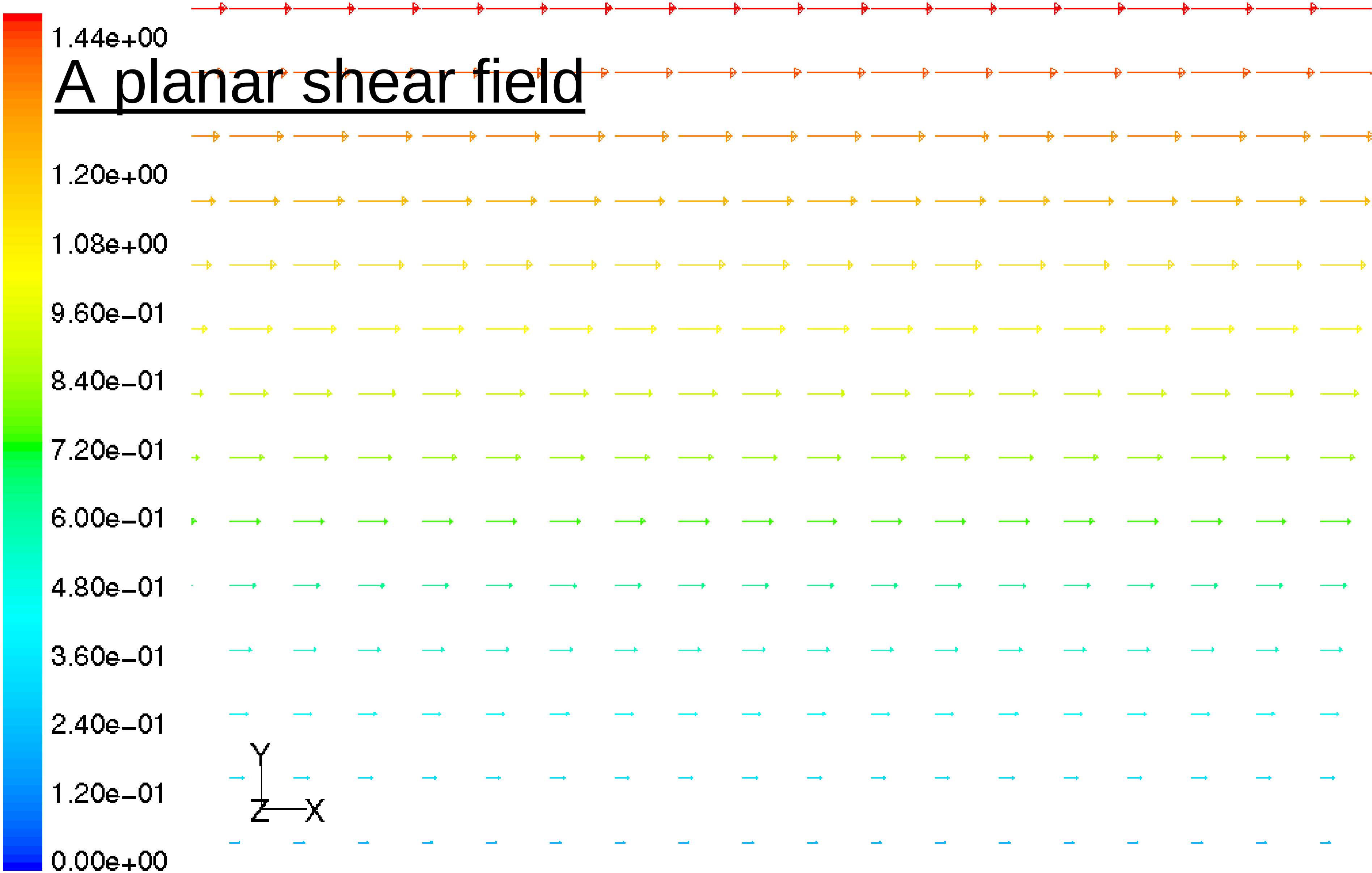


Iso-surface of vorticity magnitude colored by velocity magnitude.

Comparison between strain and vorticity

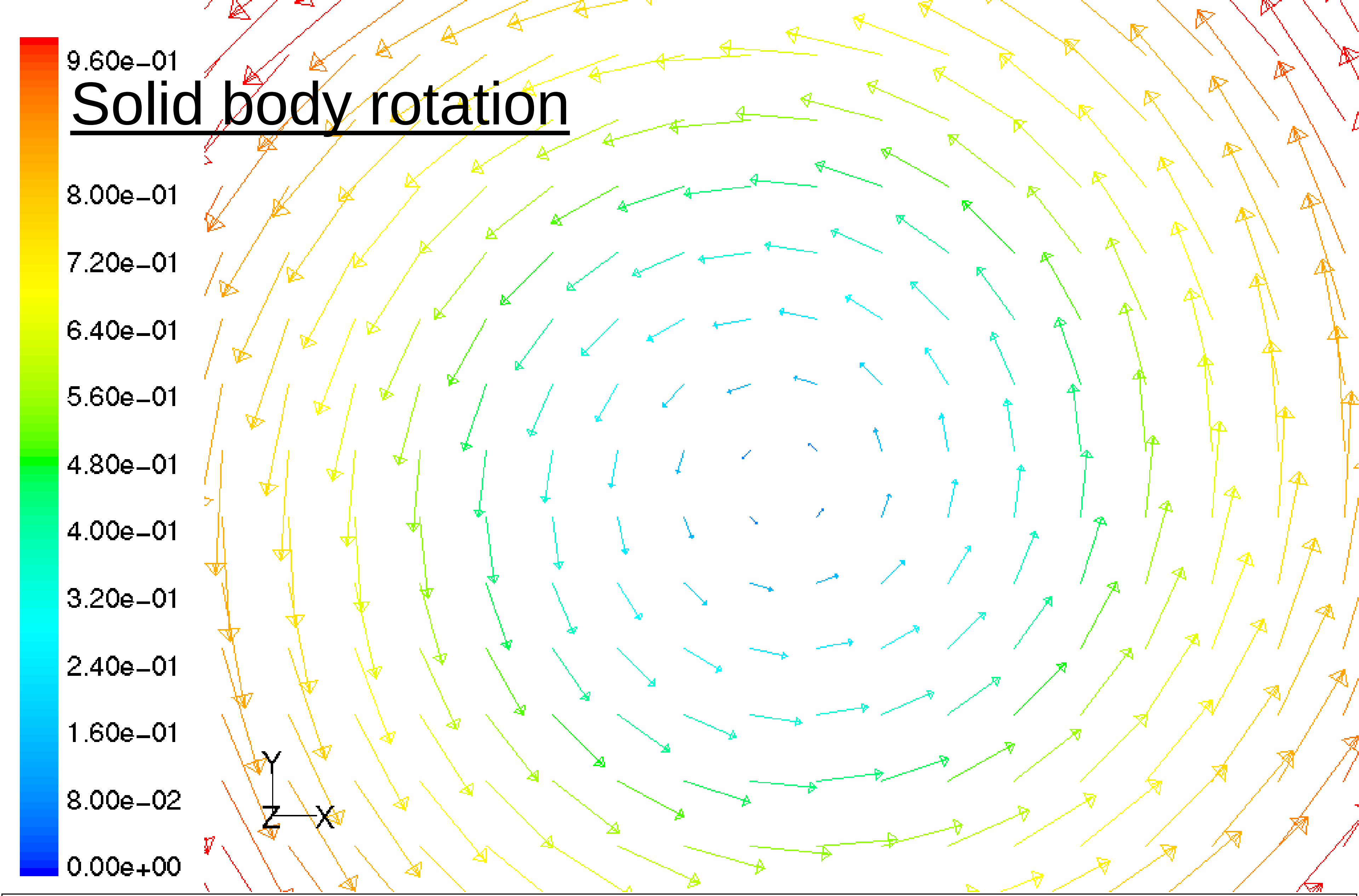
- Both strain and vorticity contain velocity gradients.
- The difference between the two will be shown based on three different flow fields:
  - A planar shear field: both the strain rate and the vorticity magnitude are non-zero.
  - A solid body rotation: the strain rate is 0(!) and the vorticity reflects the rotation speed.
  - Shear field and solid body rotation combined.

A planar shear field



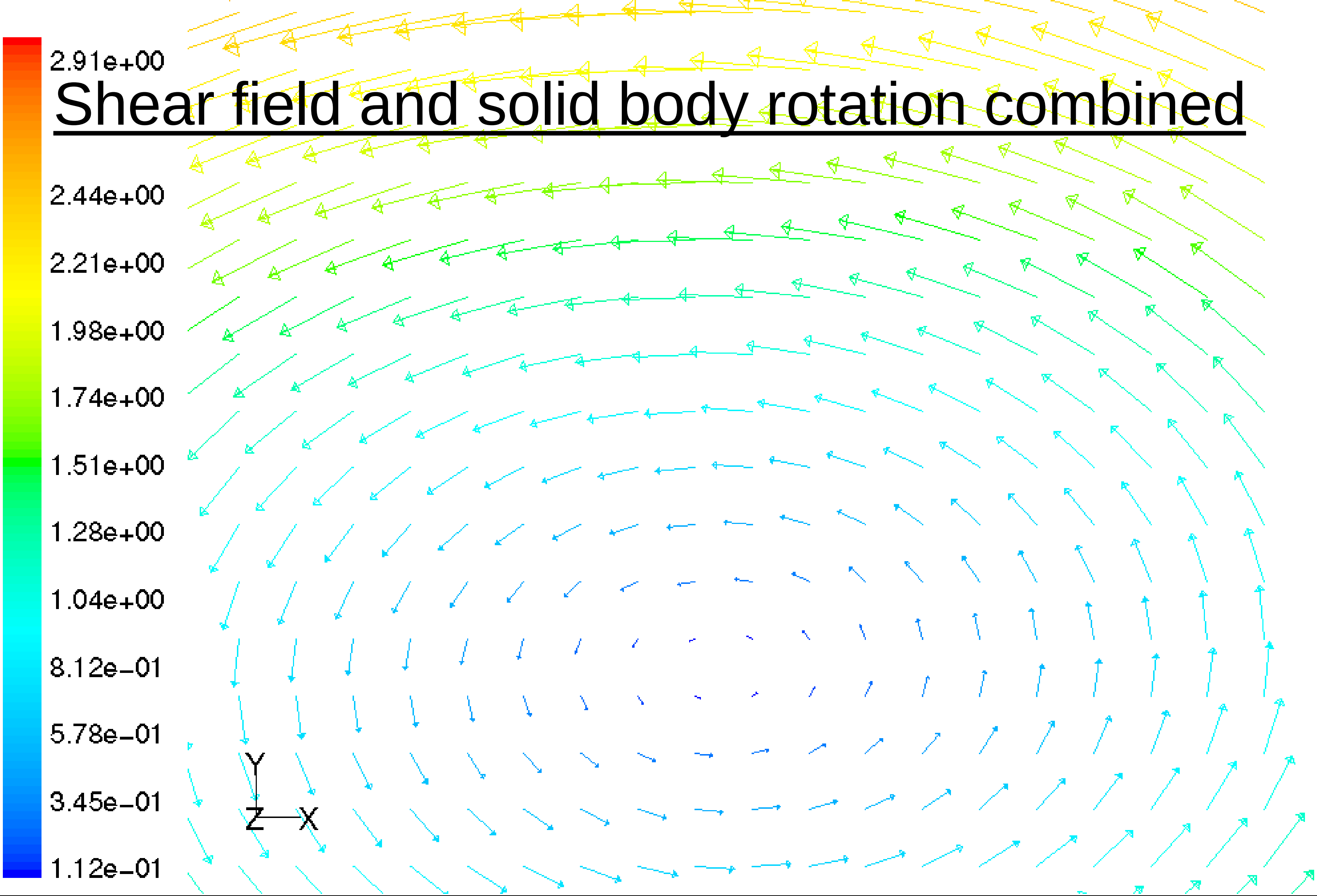
u–velocity = y
Velocity Vectors Colored By Velocity Magnitude (m/s)
vorticity–magnitude=1 1/s; strain–rate=1 1/s; du/dy=1; other derivatives are 0

Solid body rotation



angular velocity is 1 rad/s
Velocity Vectors Colored By Velocity Magnitude (m/s)
vorticity-magnitude=2 1/s; strain-rate=0 1/s; dv/dx=1; du/dy=-1; other derivatives are 0

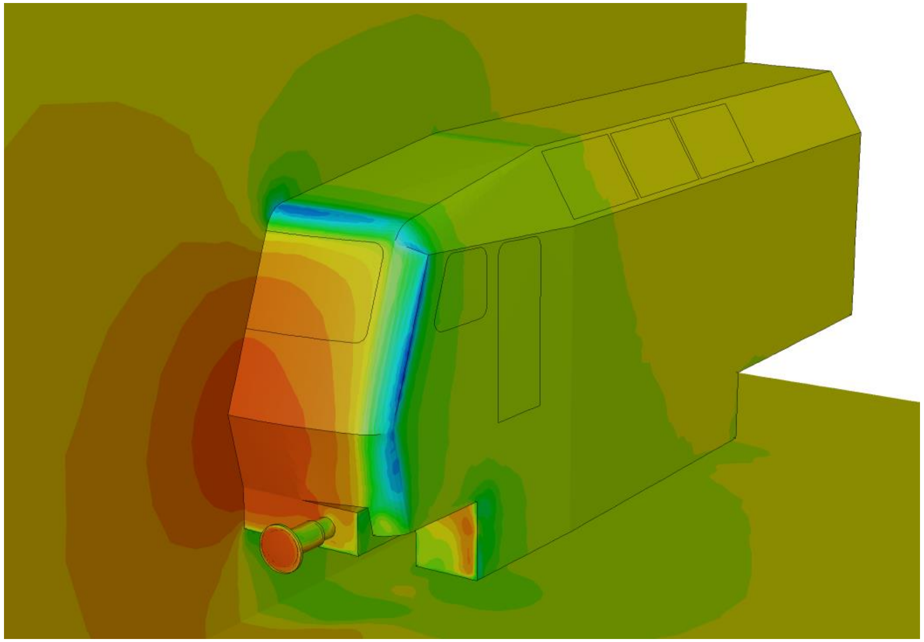
Shear field and solid body rotation combined

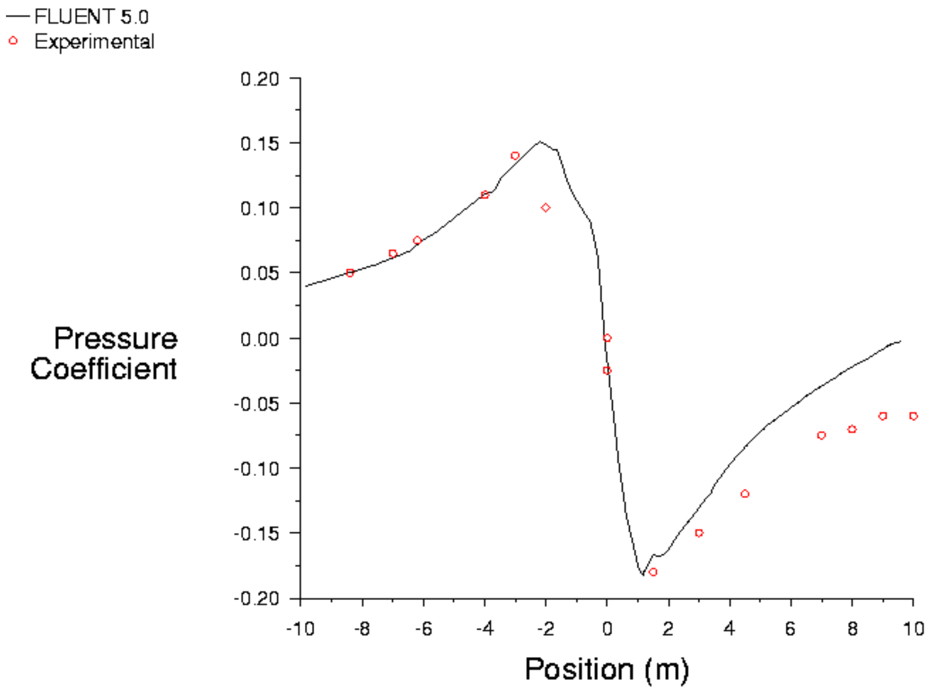


angular velocity is 1 rad/s + translational velocity u=-y
Velocity Vectors Colored By Velocity Magnitude (m/s)
vorticity-magnitude=3 1/s; strain-rate=1 1/s; dv/dx=1; du/dy=-2; other derivatives are 0

Flow over a moving locomotive

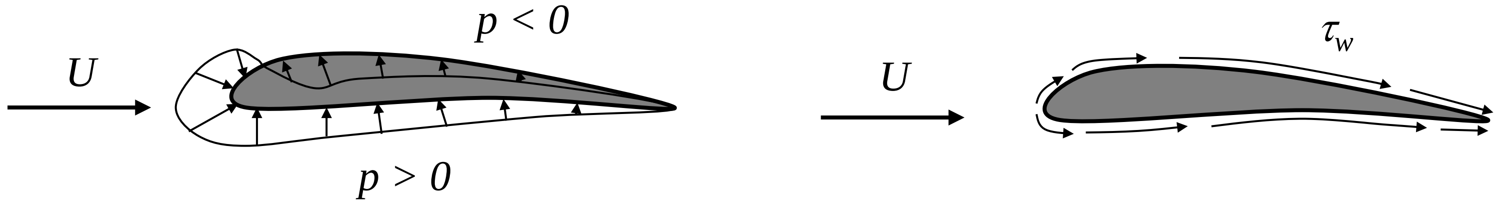
- Pressure contours show the disturbance of the passing train in the near field region.
- The figure on the left shows the pressure field over the locomotive.
- Predictions of pressure coefficient alongside the train agree reasonably well with experimental data.





Quantitative validation - NACA airfoil

- The surrounding fluid exerts pressure forces and viscous forces on the airfoil:



- The components of the resultant force acting on the object immersed in the fluid are the drag force and the lift force. The drag force D acts in the direction of the motion of the fluid relative to the object. The lift force L acts normal to the flow direction.

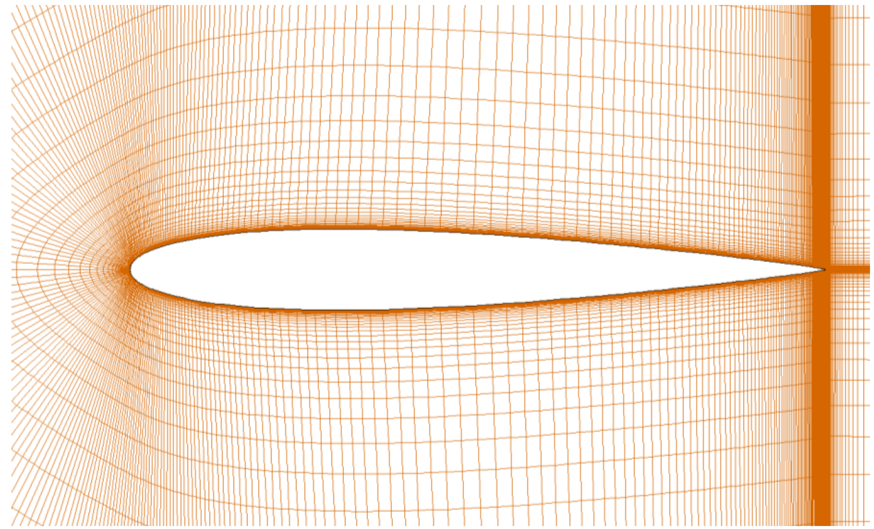
$$L = C_L . A . \frac{1}{2} \rho U^2 \qquad D = C_D . A . \frac{1}{2} \rho U^2$$

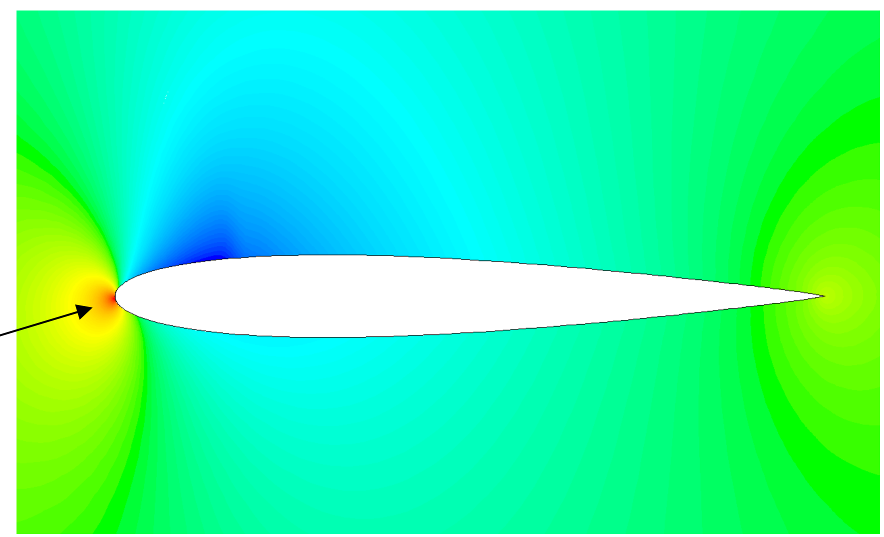
Lift *Drag*

- Lift and drag are obtained by integrating the pressure field and viscous forces over the surface of the airfoil.

Quantitative validation - NACA airfoil

- Transonic, compressible flow over the NACA 0012 airfoil is modeled using FLUENT.
  - Free stream mach number = 0.7.
  - 1.49º angle of attack.
- The realizable k-ε turbulence model with 2-layer zonal model for near-wall treatment is used.
- Pressure contours.



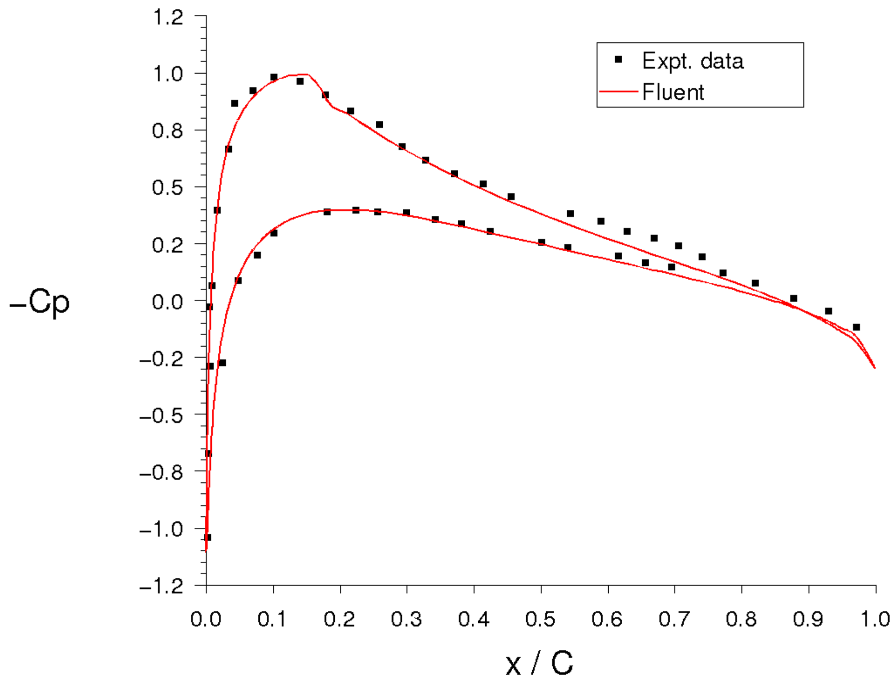


Transonic flow over NACA airfoil

- The pressure coefficient is calculated as follows:

$$c_p = \frac{p - p_0}{\frac{1}{2}\rho v_0^2}$$

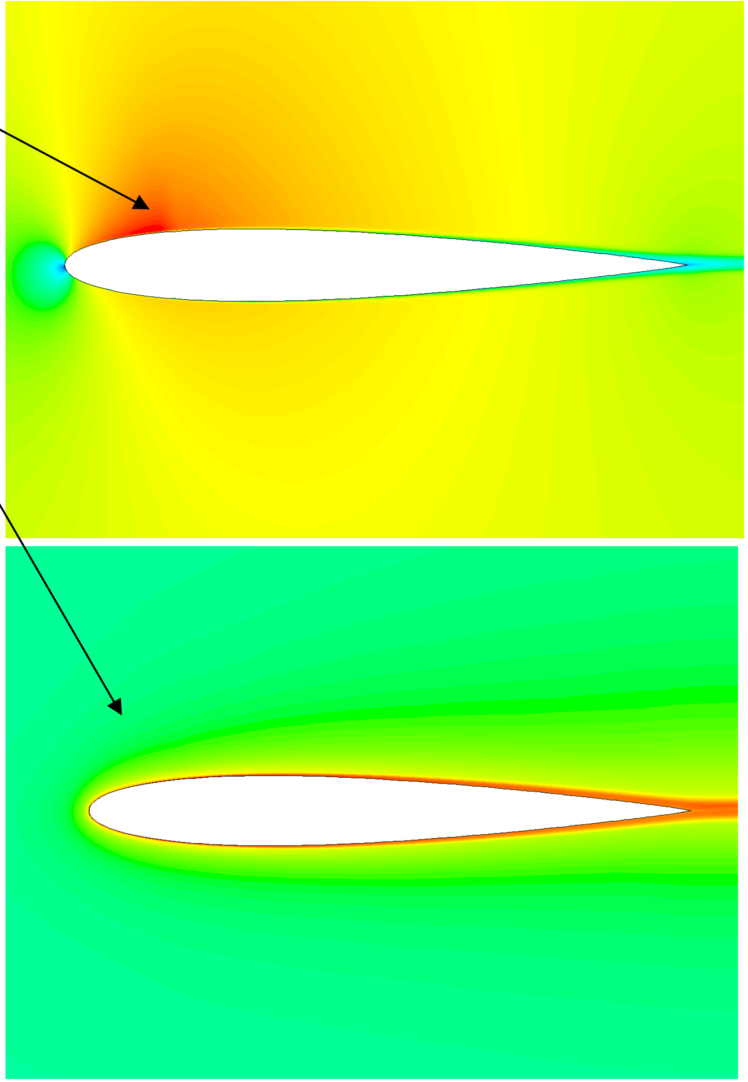
- Here p_0 is the far-field pressure and v_0 the free stream velocity.
- Pressure coefficient for upper (top) and lower airfoil surfaces shows very good agreement with data.
- Drag coefficient:
  - FLUENT: 0.0084
  - Coakley[1]: 0.0079



[1]*Thomas J. Coakley, "Numerical simulation of viscous transonic airfoil flows", AIAA-87-0416, 1987.*

Transonic flow over NACA airfoil

- Mach number contours exhibit transonic flow, with maximum (red) of 1.08.
- Turbulence kinetic energy contours show generation primarily in boundary layer.
- Overall CFD can be very useful in validating lift and drag for airfoils.



Summary

- CFD simulations result in data that describes a flow field.
- Proper analysis and interpretation of this flow field data is required in order to be able to solve the original engineering problem.
- The amount of data generated by a CFD simulation can be enormous. Analysis and interpretation are not trivial tasks and the time it takes to do this properly is often underestimated.