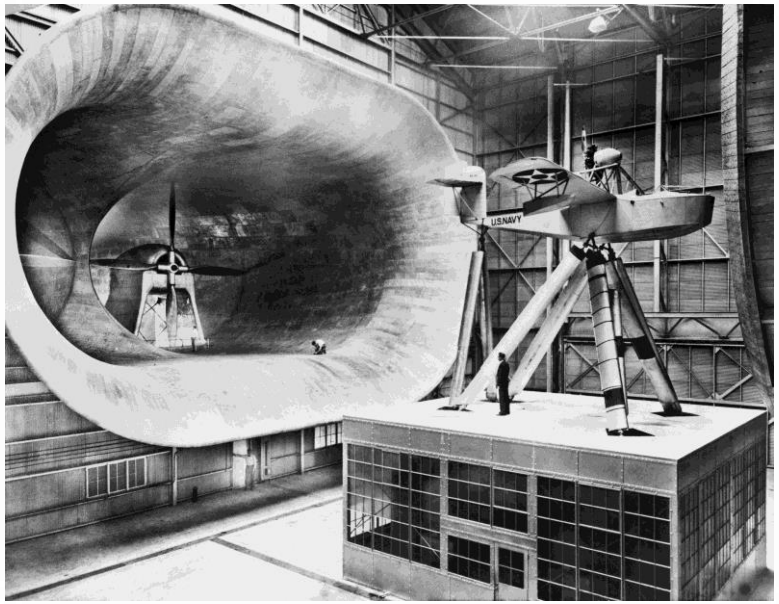
Fluid Flow Visualization

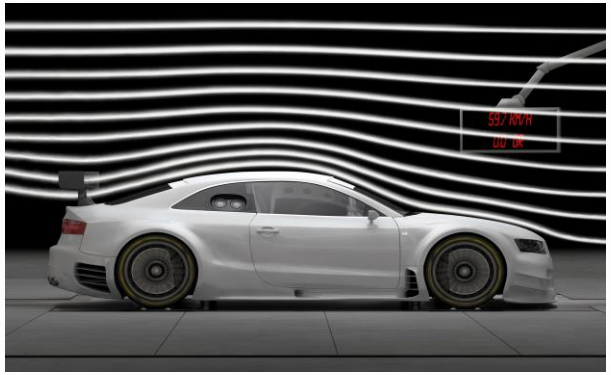
Fluid Kinematics – Lesson 4

Wind Tunnel

- A wind tunnel uses the concept of streamlines to understand the aerodynamics of an automobile or an aircraft by mimicking its motion.
- A scaled model is kept stationary and high-speed air is blown on the model using a large rotating fan.
- Smoke is used to create streamlines around these scaled models.
- The interaction of the smoke lines with the body helps the engineer understand the impact of its shape, providing a macroscopic understanding of the overall flow around the body.
- A general sense of the airflow and the generated vortices can be obtained from these smoke lines.



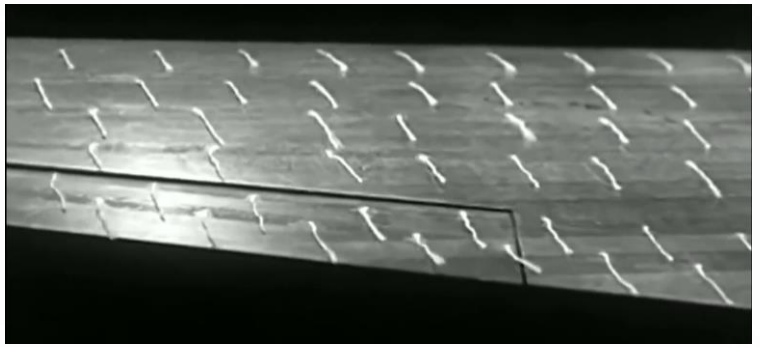
Model of an aircraft in a wind tunnel

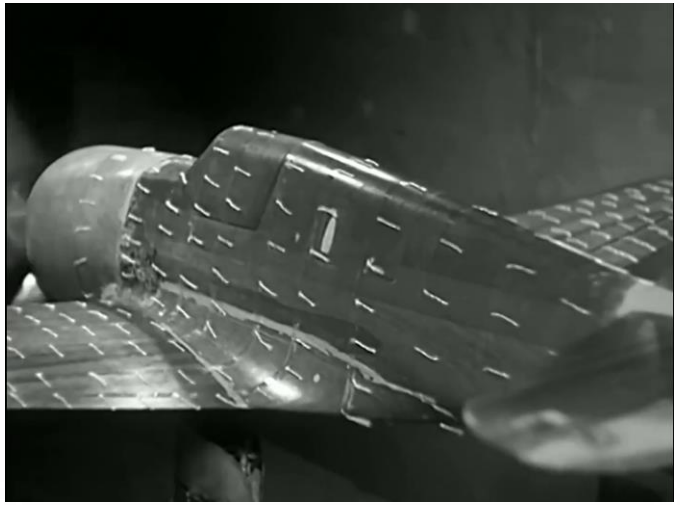

Smoke lines over an automobile

©2020 ANSYS, Inc.

Tufts

- Tufts are used to understand and visualize flow very close to the body.
- Tufts are small strips of yarn or stings about $15\ cm$ long. One end is securely glued with the surface of the model being tested while the other end is free to move with the flow.
- Length and weight of tuft is carefully selected so that they can provide a good understanding of the flow very close to the body without altering the flow.
- It helps to obtain qualitative information on cross-flow, reverse flow and flow separation close to the body being analyzed.
- These flow behaviors influence the overall resistance offered by the flow.





Tufts being used for Flow Visualization in Wind Tunnel

©2020 ANSYS, Inc.

Surface “Oil Flow” Visualization

- The surface of the body of interest is coated with a viscous oil which is transported by high-speed air flowing around the body, creating streaks on the surface.
- The resulting streaks can be used to visualize the flow on the surface of the body of interest.
- These oil streaks act as good indicators of flow separation.
- The initial thickness of the oil film is an important parameter to obtain useful results.

©2020 ANSYS, Inc.

Water Tunnel Visualization

- Water tunnels are used to visualize and evaluate the hydrodynamic behavior of submerged bodies under water.
- Like wind tunnels, the model is kept stationary as water flows over it.

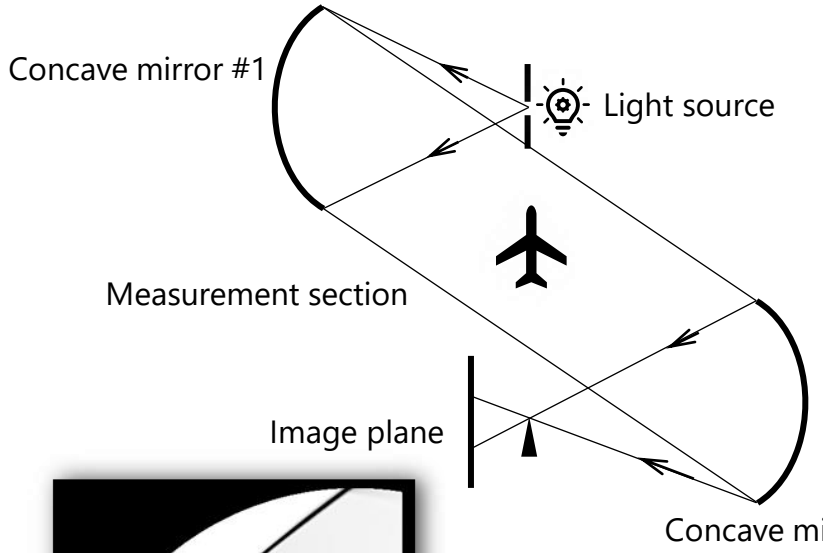
- Colored dyes are injected into the flow, which helps to visualize macroscopic flow around the bodies and the locations of vortices.
- Certain water tunnels can increase or decrease the static pressure to perform cavitation studies, which play a crucial role in the design process of propeller blades of submarines and ships.

Dye injection in water

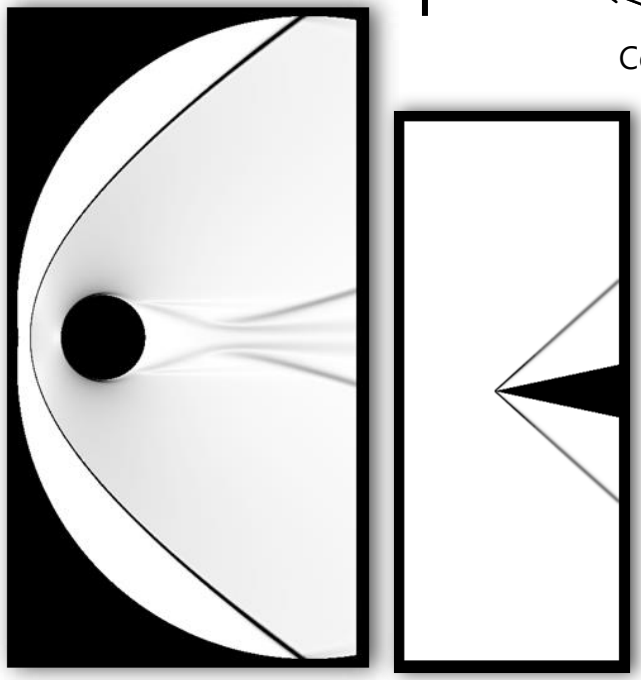
©2020 ANSYS, Inc.

Schlieren Photography

- Schlieren photography is an optical visualization technique that distinguishes fluids of varying densities.
- It uses the principle of refraction, i.e., light rays bend when traveling from a denser medium to a rarer medium and vice-versa.
- The setup consists of a bright source of light, a knife edge for focusing the light and two concave mirrors on either side of the flow being visualized.
- This method was primarily developed for studying high-speed supersonic flows.



Schlieren setup



Bow shock & normal shock
(Ansys Fluent simulation)

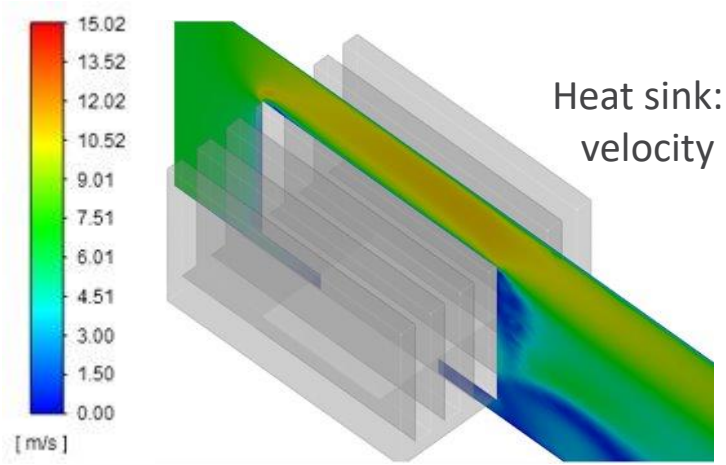
©2020 ANSYS, Inc.

Visualization in Simulation

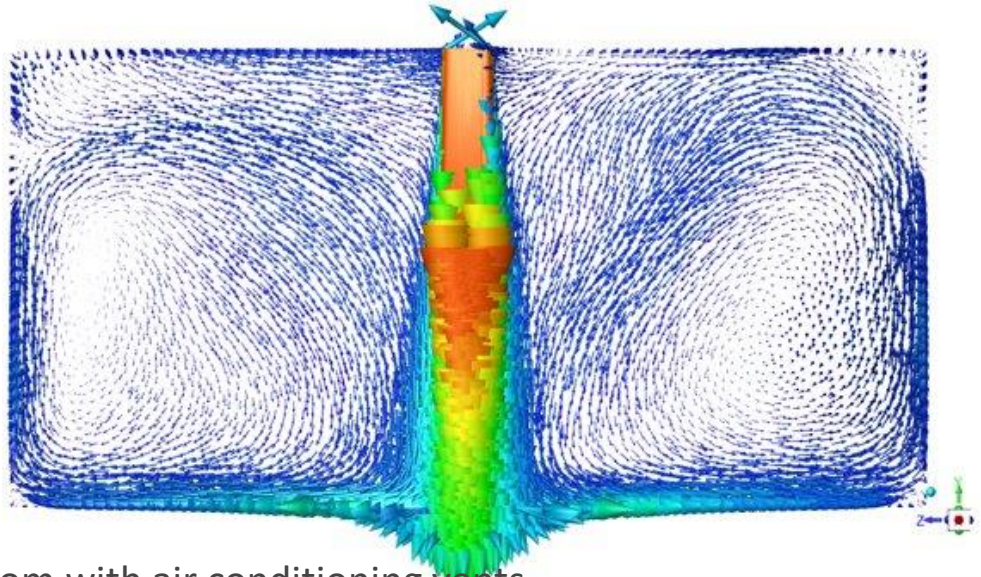
- While experimental visualization techniques provide invaluable insight into the flow physics, the information they provide is oftentimes limited by:
  - The amount of information they can provide
  - Placement of the instrument
  - Optical blockage
  - The flow regimes of the measurement devices used

- Experimental visualizations are also expensive, both in terms of time it takes to set up the apparatus, conduct and post-process the experiment, and the cost of equipment and staff time.

- On the other hand, visualization of a numerical flow field calculated with a high-fidelity Computational Fluid Dynamics code is free of these limitations and can present a complete virtual visualization of the fluid flow, oftentimes at a fraction of the cost of the experiment



Heat sink: contours of velocity magnitude

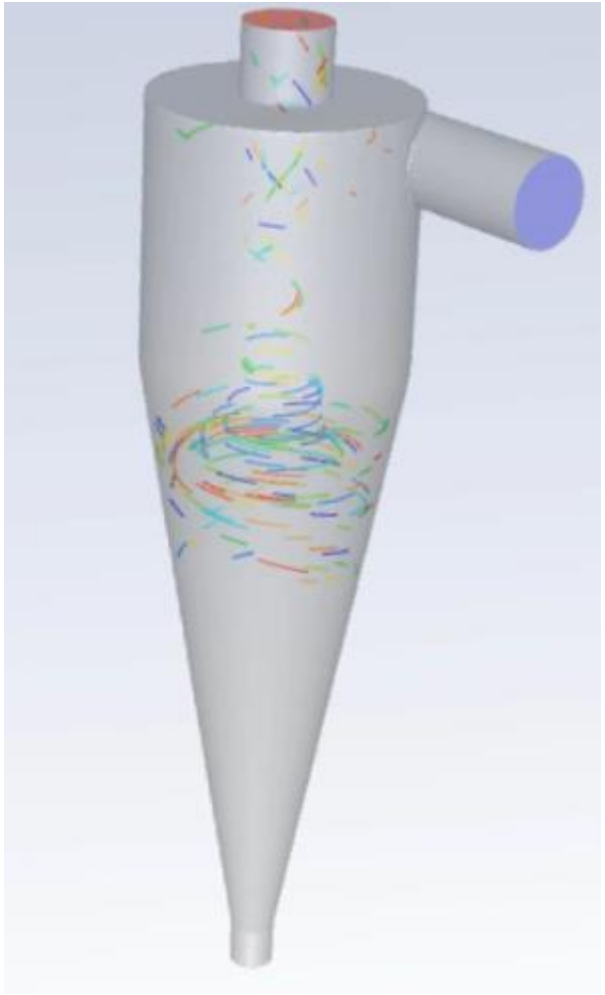


Room with air conditioning vents – velocity vectors

©2020 ANSYS, Inc.

Visualization in Simulation

- A multitude of options is available to visualize and examine details of a simulated flow field:
  - Contour plots of flow variables on arbitrary defined surfaces
  - Distribution of vector fields
  - Streamlines, pathlines, particle tracks
  - Animations of flow fields
  - Visualization of 3D flow structures such as vortices and turbulent eddies
  - 3D submerged virtual reality, or virtual travel through the fluid system
  - …

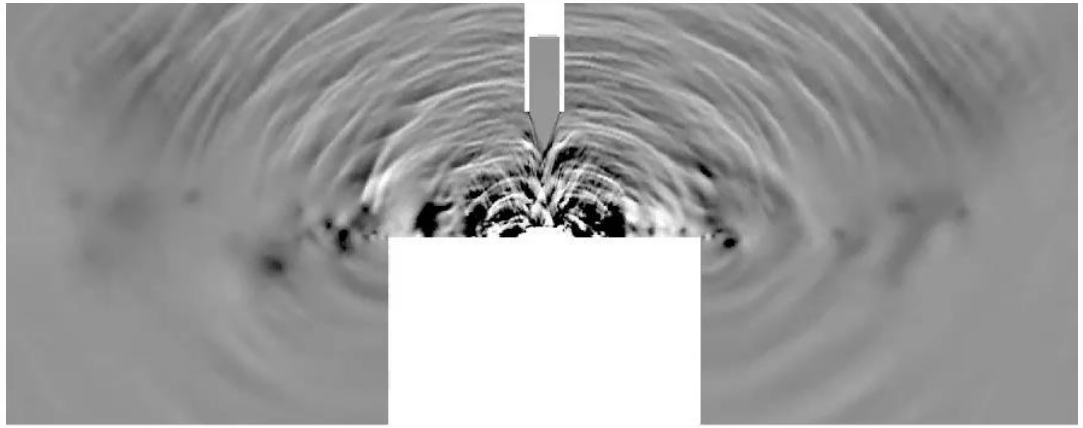


Pulsing pathlines in a cyclone separator

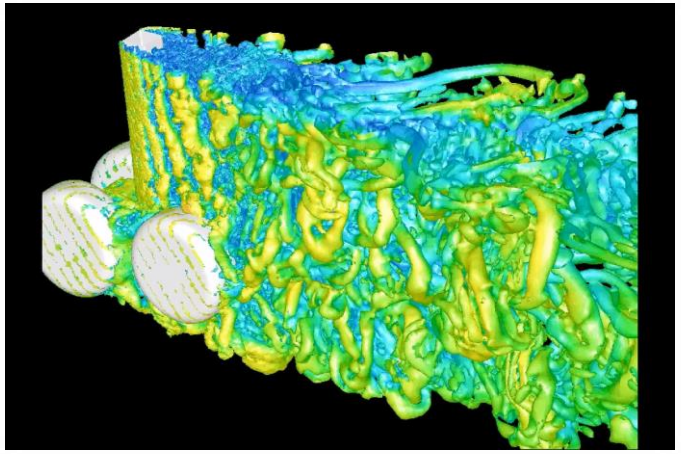
©2020 ANSYS, Inc.

Visualization in Simulation

- These animation examples illustrate the power of virtual visualization of numerical simulation results which would be difficult or impossible to obtain using experimental techniques:



Generation and propagation of sound waves by supersonic impinging jet



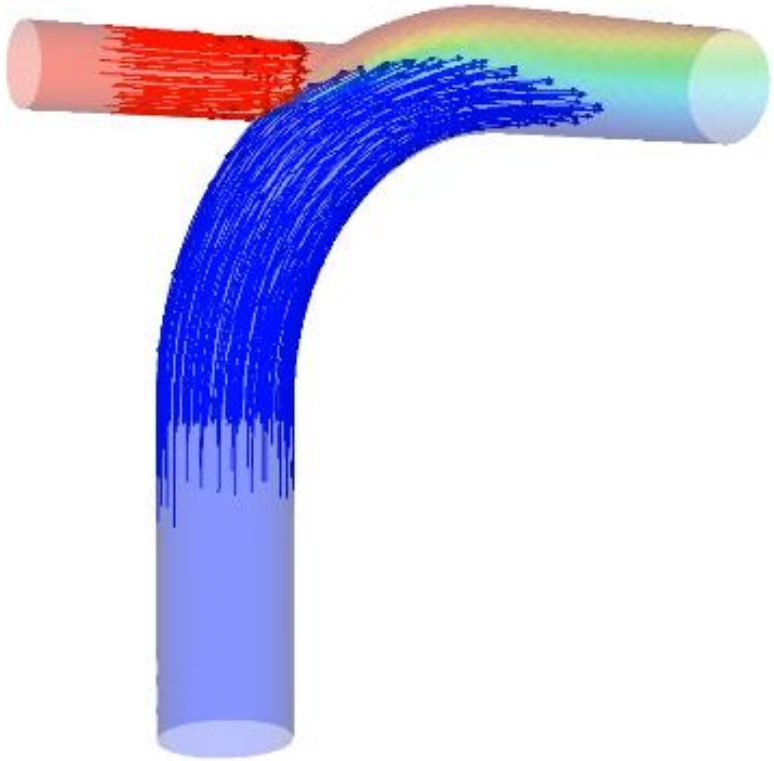
Details of instantaneous turbulent structures behind a landing gear

©2020 ANSYS, Inc.

Summary

- Experimental visualization of fluid flows provides important insights into flow physics and expands our understanding of fluid dynamics.

- Visualization of simulation results calculated by a high-fidelity CFD code adds capabilities to examine flow features otherwise not observable by the experimental techniques and further contributes to broadening our knowledge of fluid flows.



Mixing of fluids at different temperatures in a mixing elbow

©2020 ANSYS, Inc.


Ansys