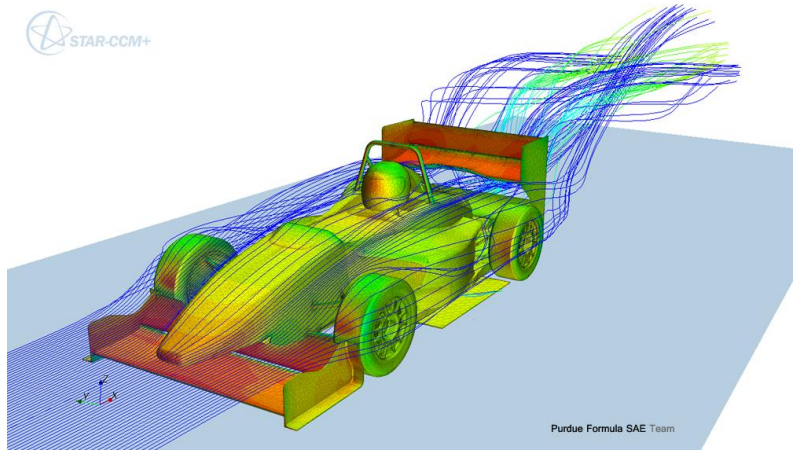
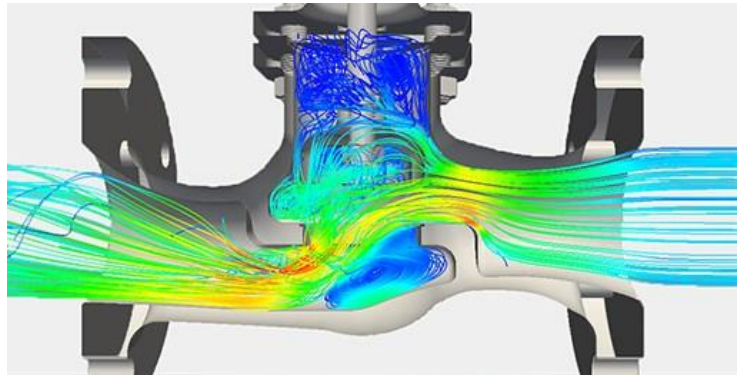
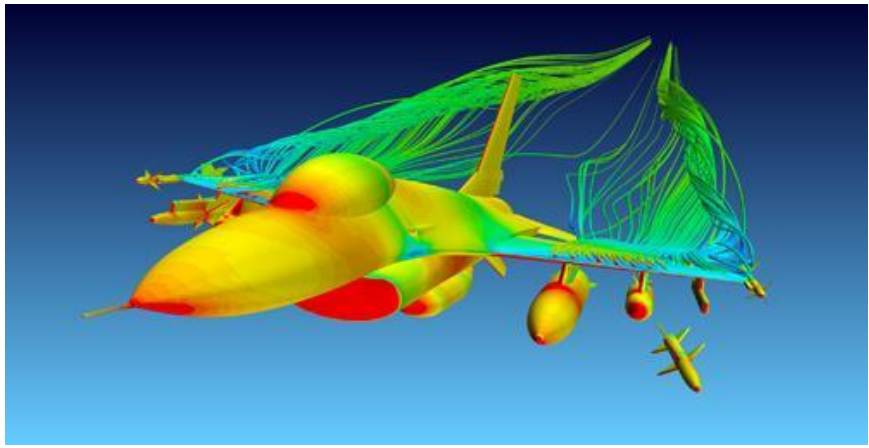
Computational Fluid Mechanics
Lecture 1







Dr./ Ahmed Nagib Elmekawy

Oct 13, 2018

Course Description

- The course is an introductory course to computational fluid dynamics for graduate students.
- Finite difference method will be used to solve different type of Partial Differential Equations (PDEs) that descript different fluid dynamics and heat transfer problems.
- Several model equations will be considered for finite difference discretization, stability and error analysis.

Course Description (Cont.)

- Solution schemes and boundary conditions treatment will be selected based on the PDEs classification.
- Scalar form of Navier-Stokes (NS) equations will be solved along with two-dimensional incompressible NS equations.

Course Contents

- Introduction (What is CFD?, Physical classifications of fluid dynamics problems, governing equations)
- Classification of PDEs (Characteristics lines and boundary conditions)
- Finite Difference Method (Taylor series, truncation error, consistency, stability, and convergence)
- Stability analysis
- Parabolic PDEs

Course Contents (Cont.)

- Elliptic PDEs
- Hyperbolic PDEs
- Scalar NS equation
- Introduction to Finite Volume

Course Contents (Cont.)

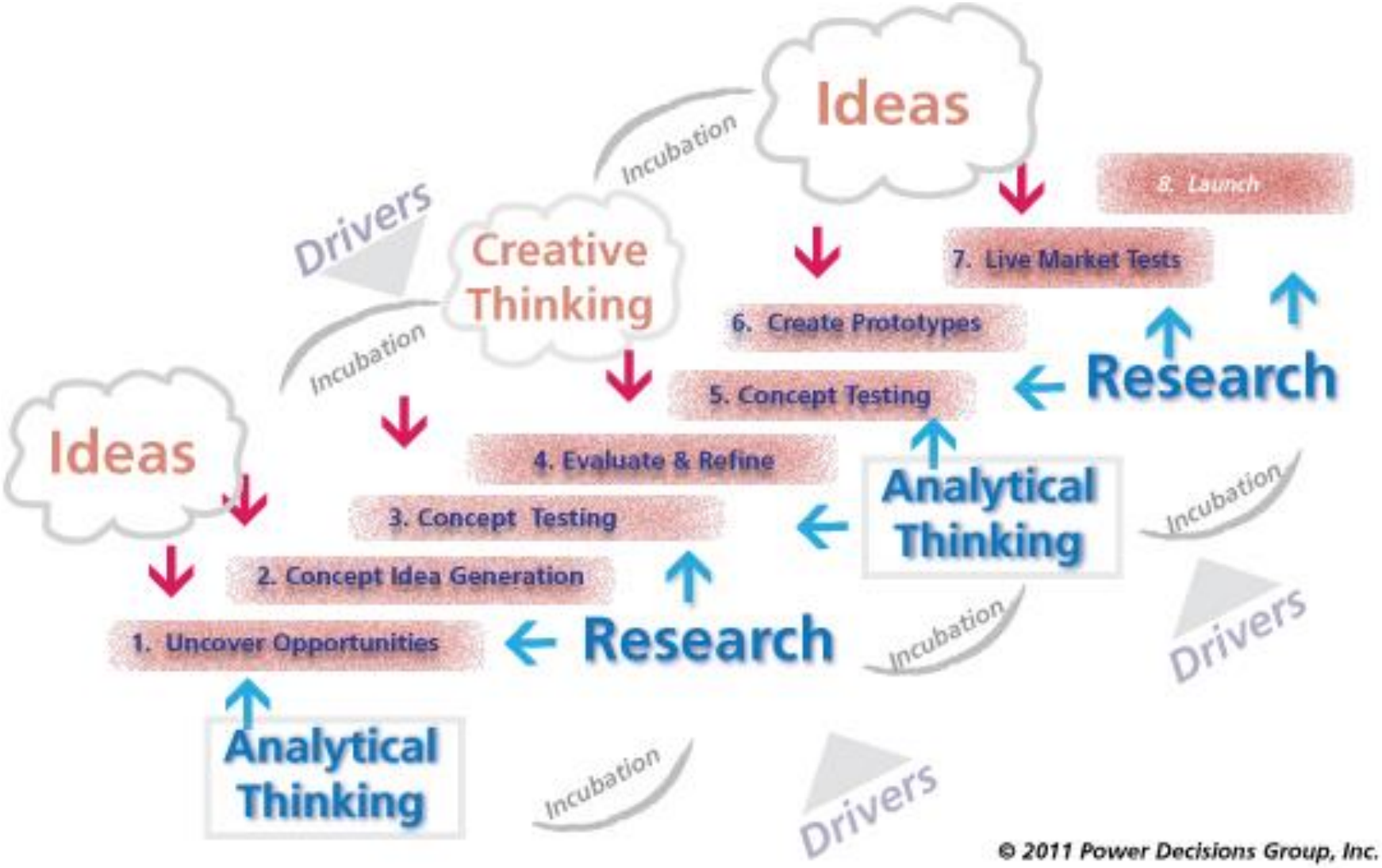
Introduction to Ansys Fluent

- Intoduction to CFD by using ANSYS
- Introduction to Meshing
- CFD: Setting up Domain and Physics
- CFD: Turbulence Modeling and Transient

References

- Hoffmann, A., Chiang, S., Computational Fluid Dynamics for Engineers, Vol. I, 1st ed., Engineering Education System, 1993
- Lecture PowerPoint notes
- Hirsch, C., Numerical Computation of Internal and External Flows, 2nd ed., Butterworth-Heinemann, 2007
- Anderson, J. D., Computational Fluid Dynamics – The Basics with Applications, McGraw-Hill, 1995.
- Pletcher, R. H., Tannehill, J. C., Anderson, D., Computational Fluid Mechanics and Heat Tranfer, 3rd ed., CRC Press, 2011
- Ferziger, J. H., Numerical Methods for Engineering Application, 2nd ed., Wiley, 1998.
- Chung, T.J., Computational Fluid Mechanics, 2nd edition, Cambridge University Press, 2010.

Research and Development



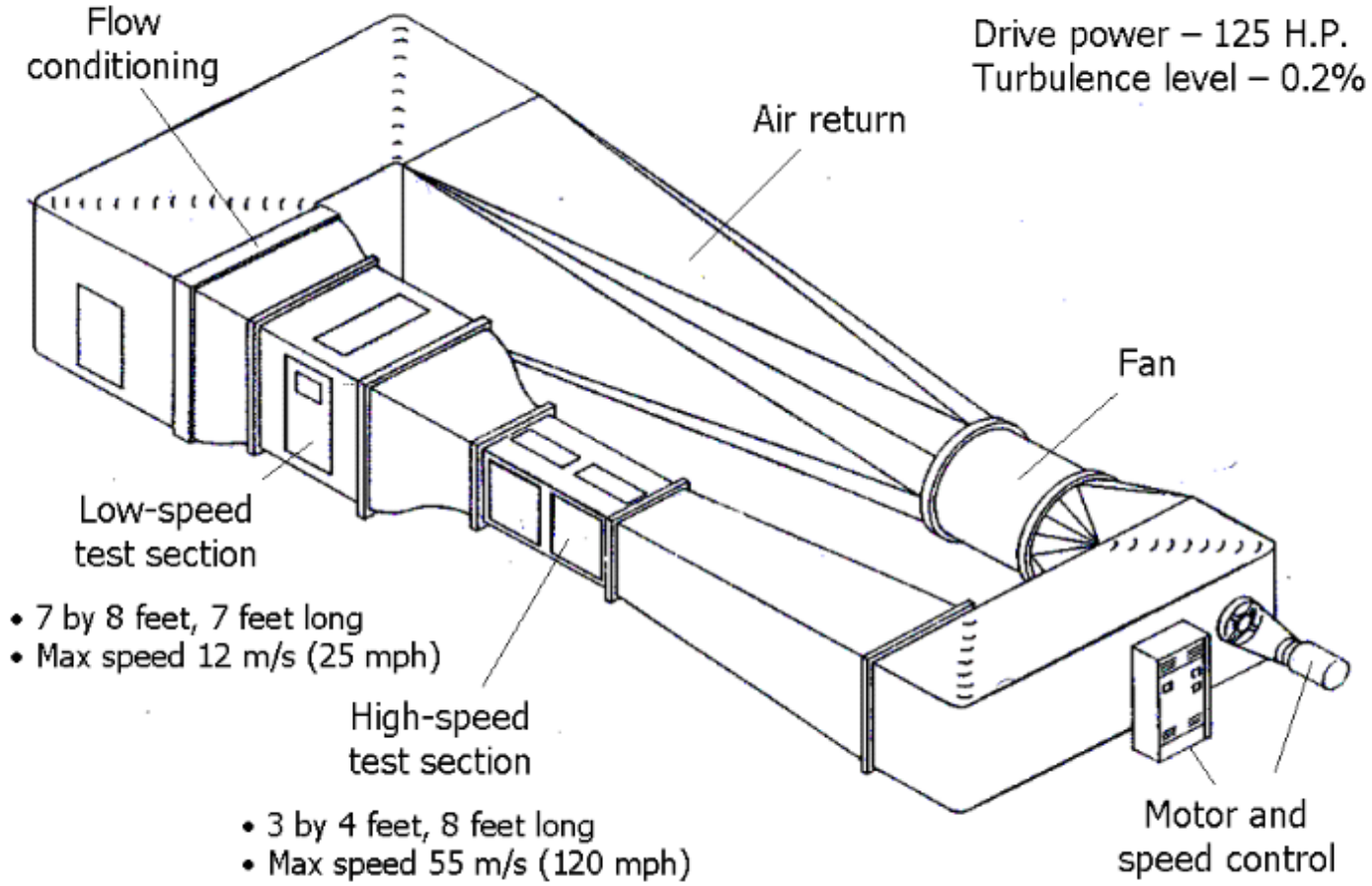
Fluid Dynamics

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 condition problems 3. Computer costs

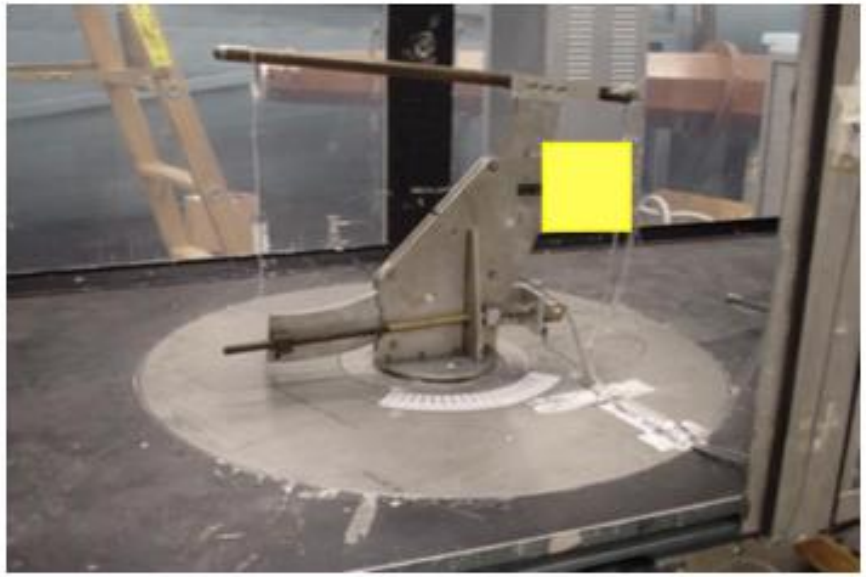
Introduction to Experimental Fluid Dynamics

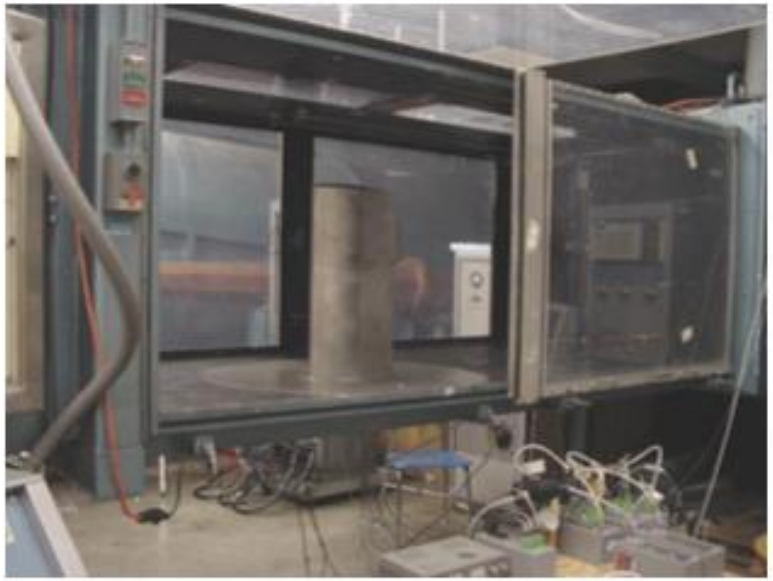
Wind Tunnel

ODU Low Speed Wind Tunnel – KH 143



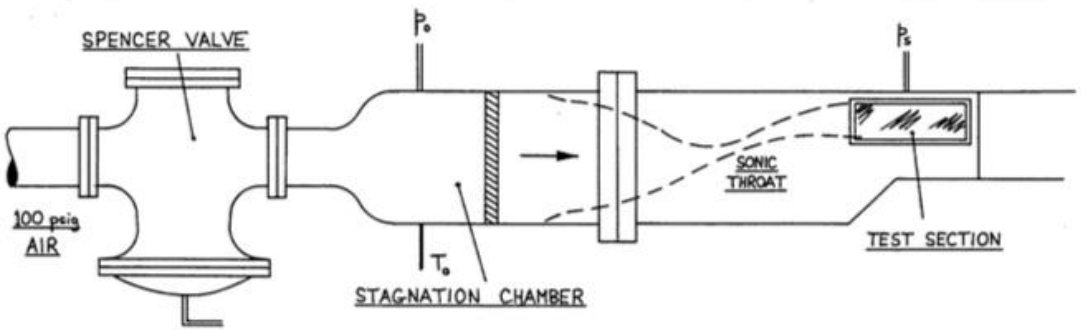
Wind Tunnel





Wind Tunnel

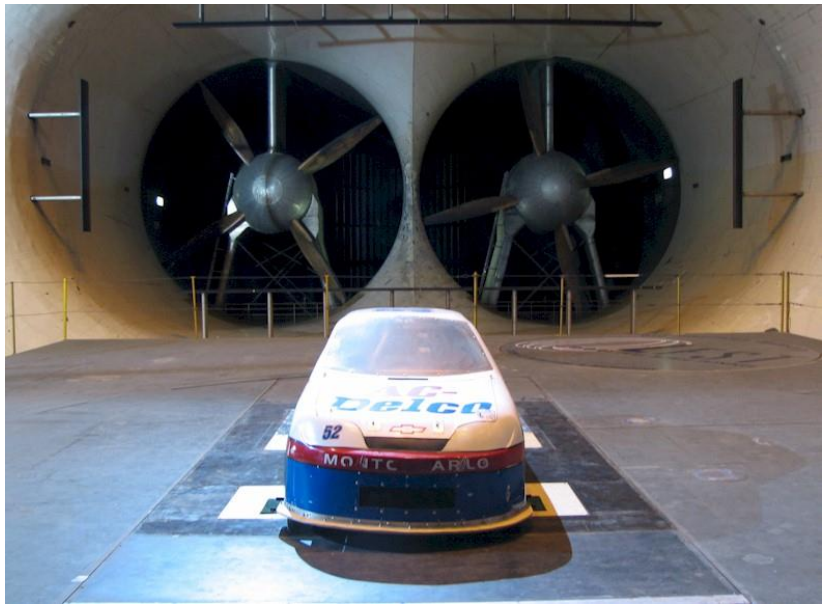
ODU Supersonic Wind Tunnel – KH 143

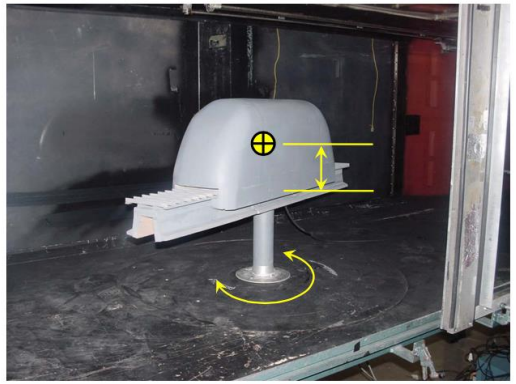


Wind Tunnel

The MIT / NASA Langley Magnetic Suspension/Balance System
NASA Langley Research Center 6/11/1991 Image # EL-1996-00037

Wind Tunnel





Wind Tunnel

Internal Balances

- Below – FF-10 wind tunnel balance

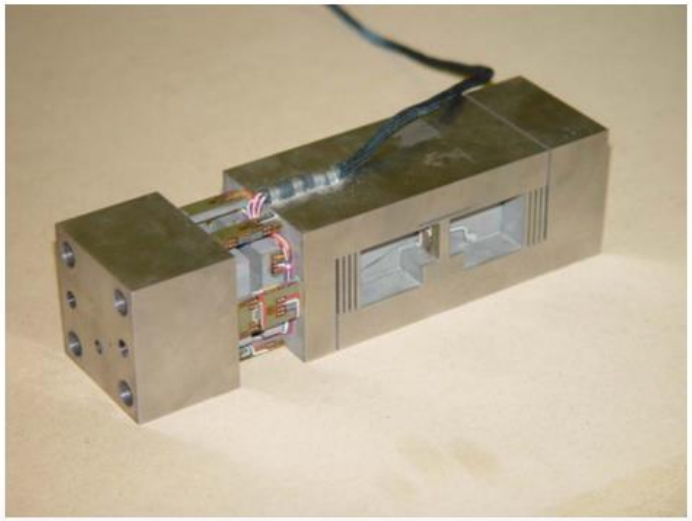
  6 components

- Right – HRC-3 wind tunnel balance

  3-components

- Below right – ATI-Gamma general purpose balance

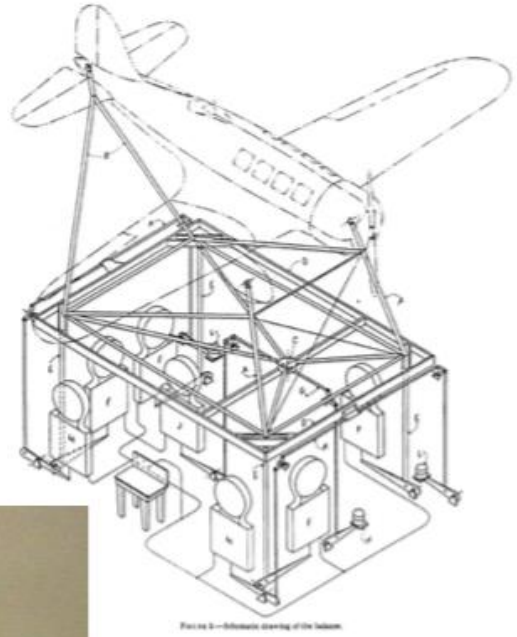
  6 components



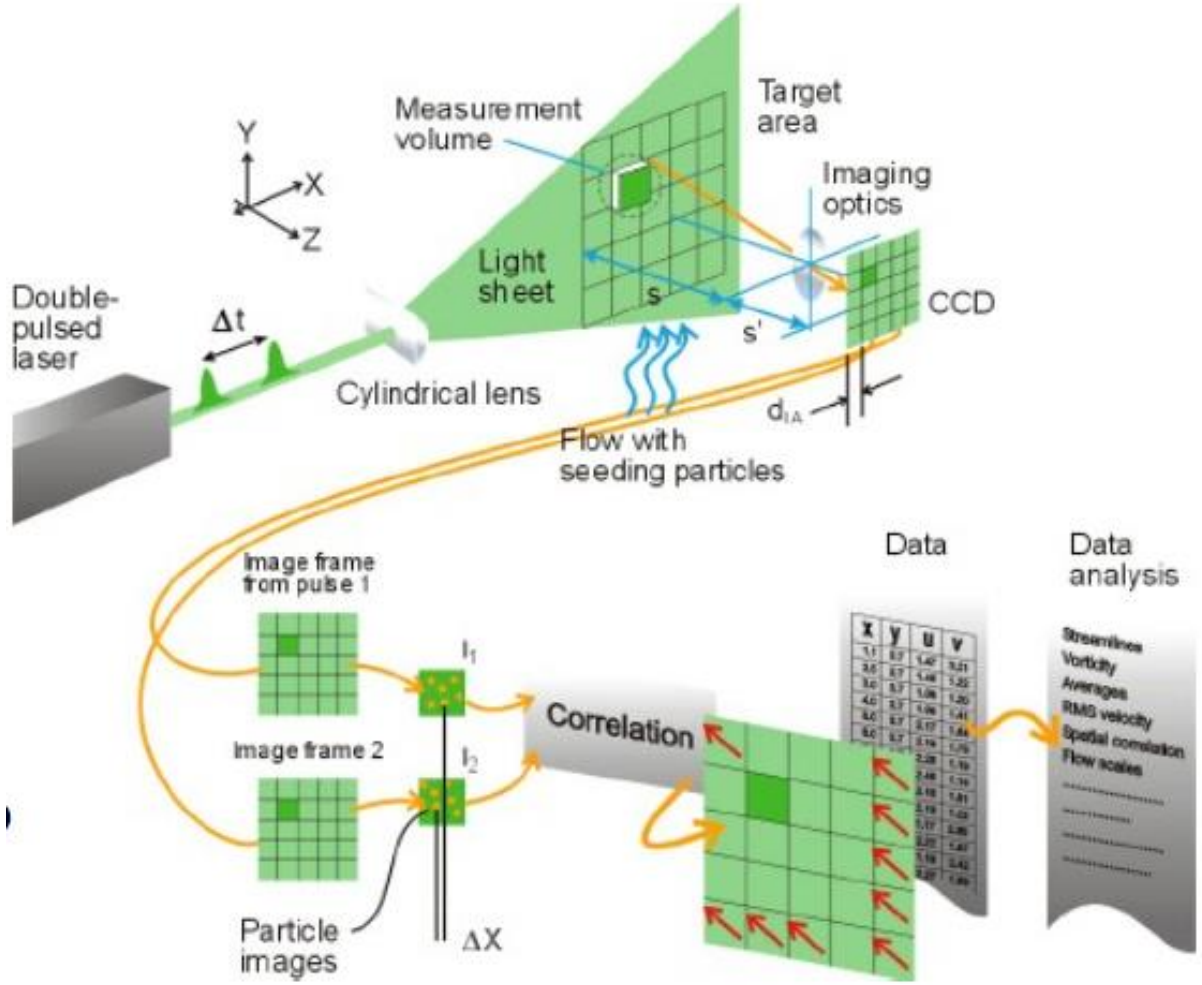
Wind Tunnel

External Balances

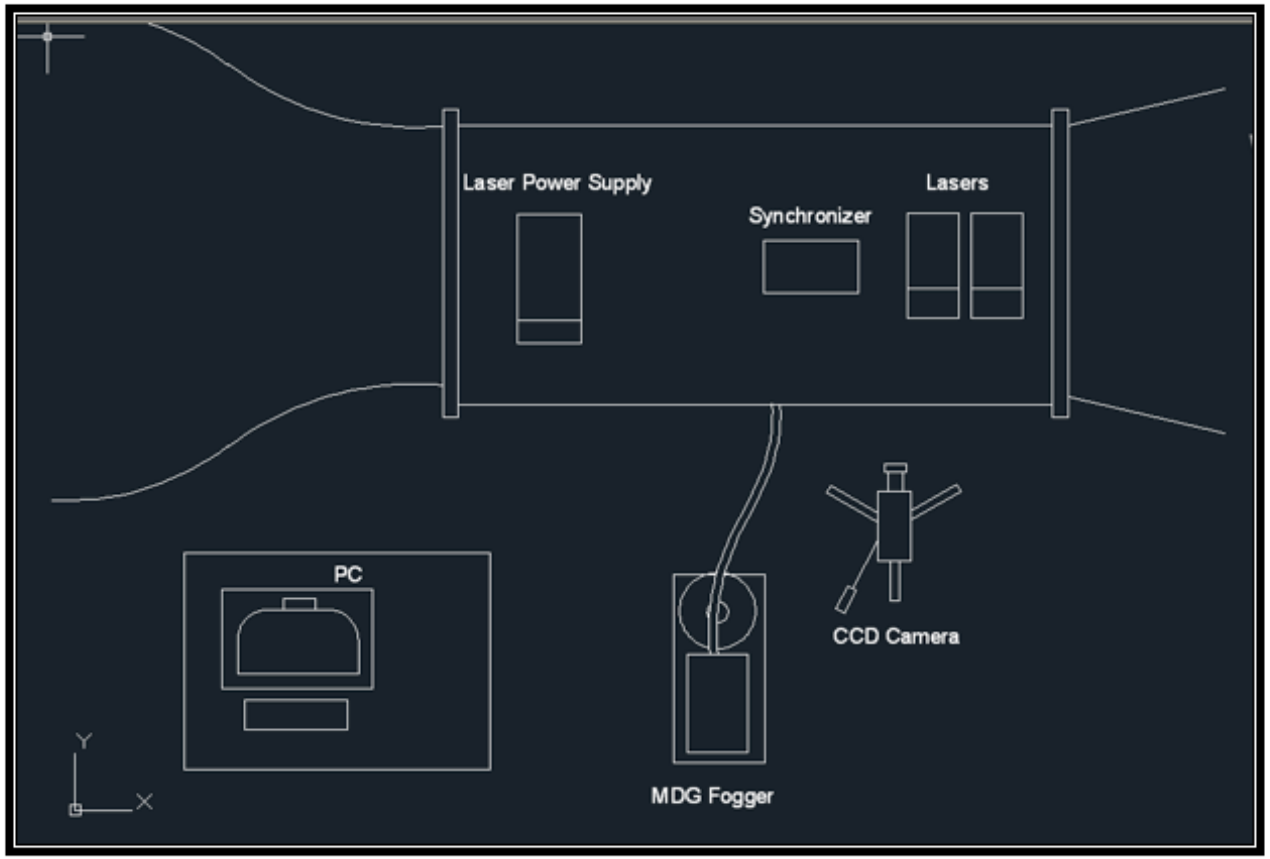
- Right upper – Langley Full-Scale Tunnel
- Right lower – Texas A&M
- Upper center – Aerotech
- Low center – Wright Brothers
- Below – U. Washington



Particle Image Velocimetry (PIV)



Particle Image Velocimetry (PIV)



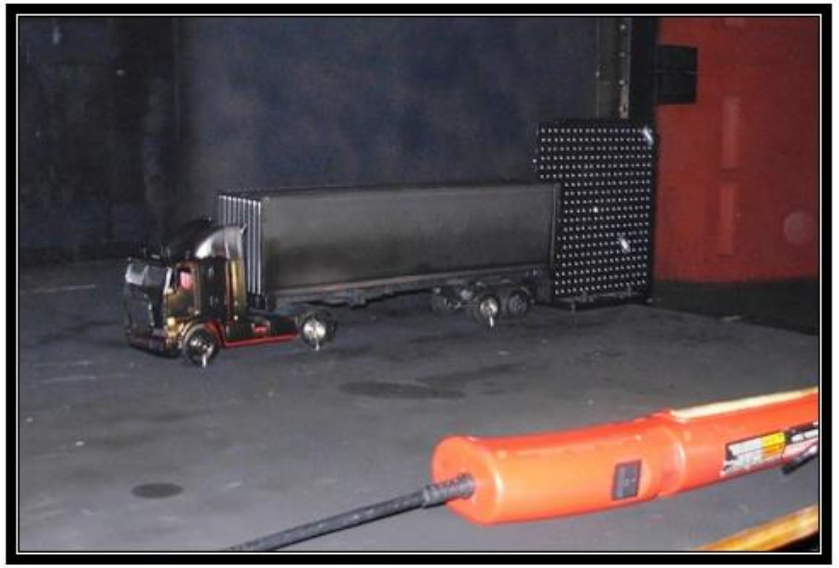
Picture #1 shows the general layout of the experiment. The tested truck is located inside the test section below the lasers and the synchronizer.

Particle Image Velocimetry (PIV)

Picture #2 heavy-duty truck mounted on a plywood.

Picture #3 dual Nd: YAG laser.

Particle Image Velocimetry (PIV)

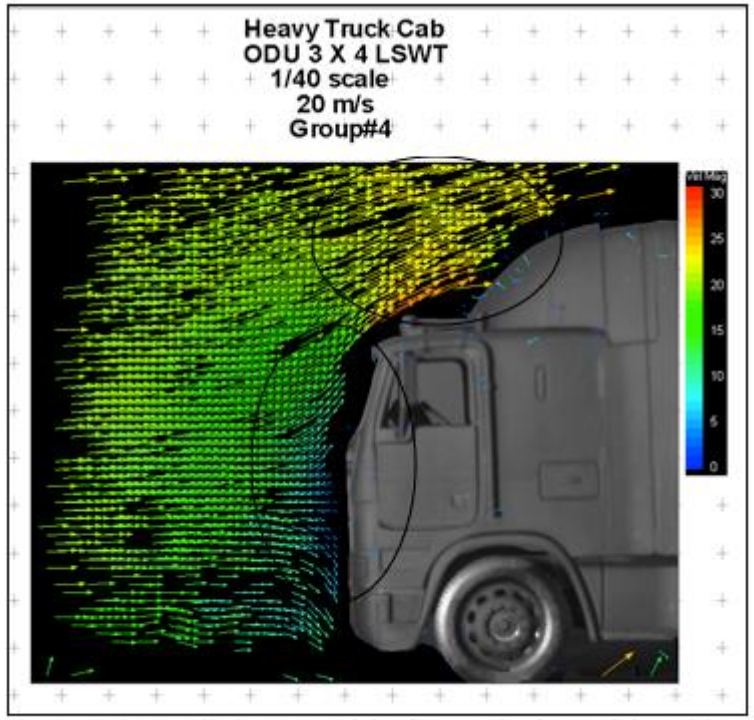


Picture #8 optics alignment target.

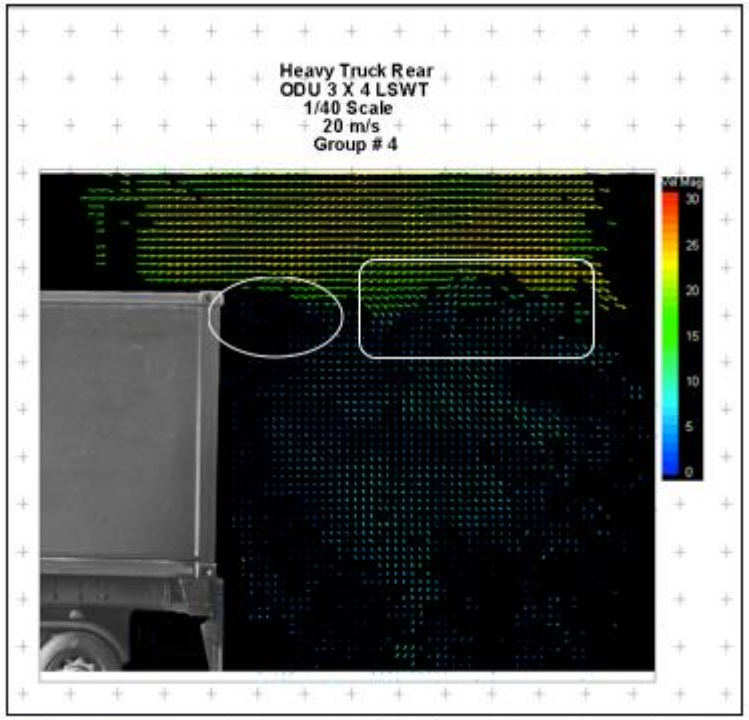
Picture #9 Layout.

Particle Image Velocimetry (PIV)

Particle Image Velocimetry (PIV)

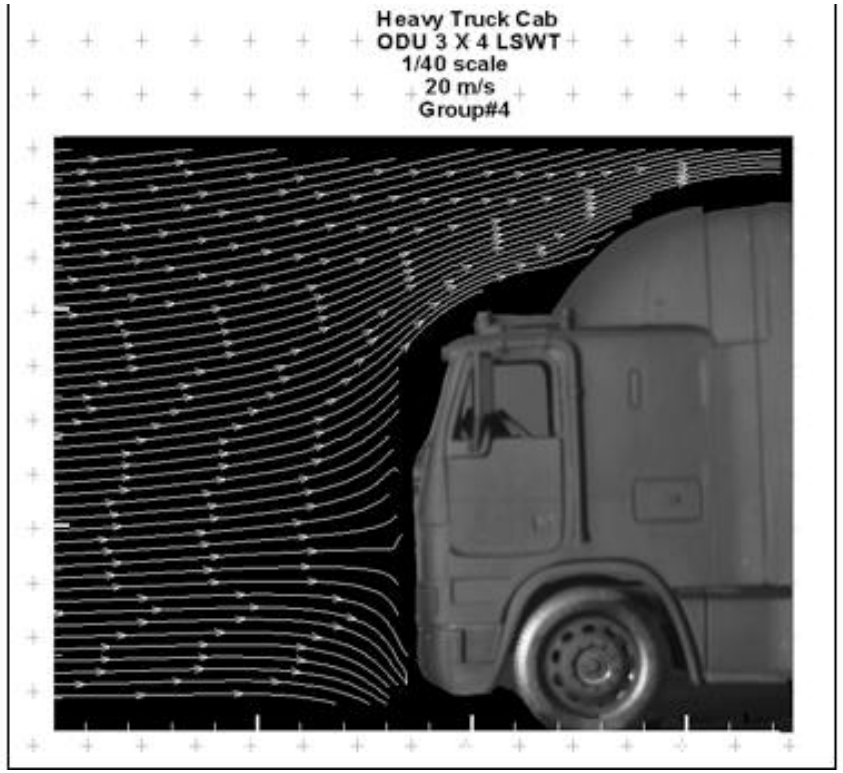


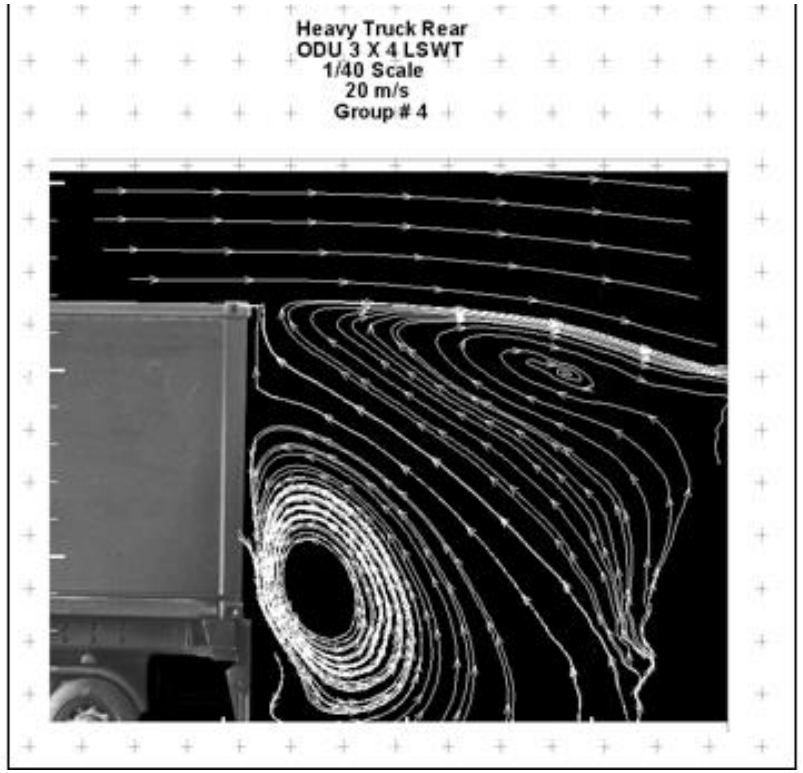
Picture #12 overall character of the flow at the front of the cabin.



Picture #13 overall character of the flow behind the trailer.

Particle Image Velocimetry (PIV)

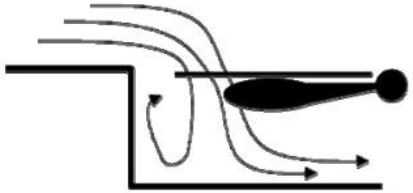




Particle Image Velocimetry (PIV)

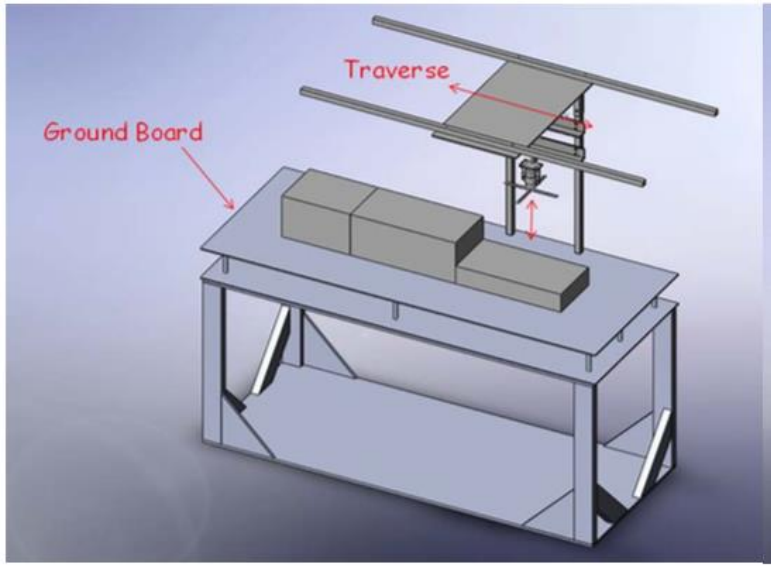
Case Study: PIV Measurements

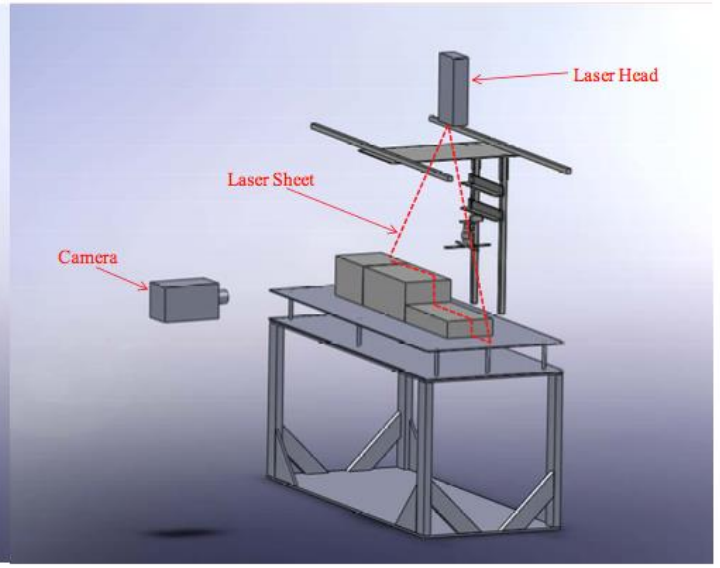
- Interaction of Rotor Downwash and Ships Airwake
  - ODU LSWT – Large test section
  - Simplified frigate model and fixed pitch rotor
- Motivation
  - Landing a helicopter in the "airwake"' of a bluff body ship superstructure
  - Frigate and MH-60 Seahawk Helicopter



Particle Image Velocimetry (PIV)

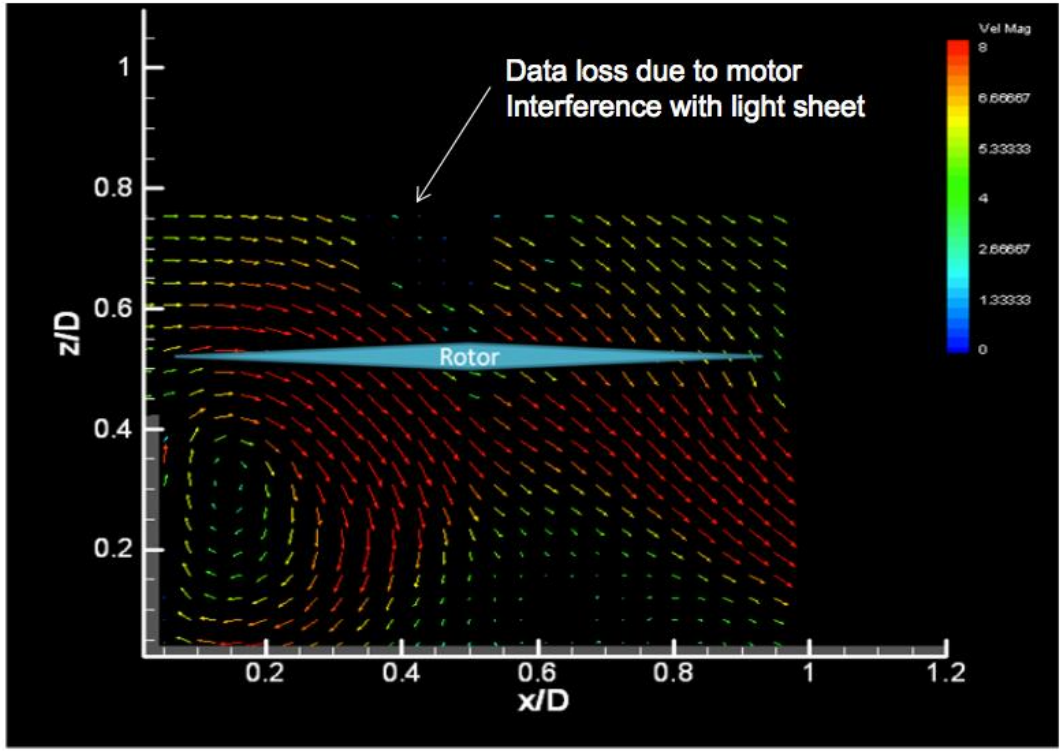
- Electric motor drives rotor, mounted on traverse
- Overhead window with Dual Yag Laser shining through
- Side window allows camera to view laser sheet





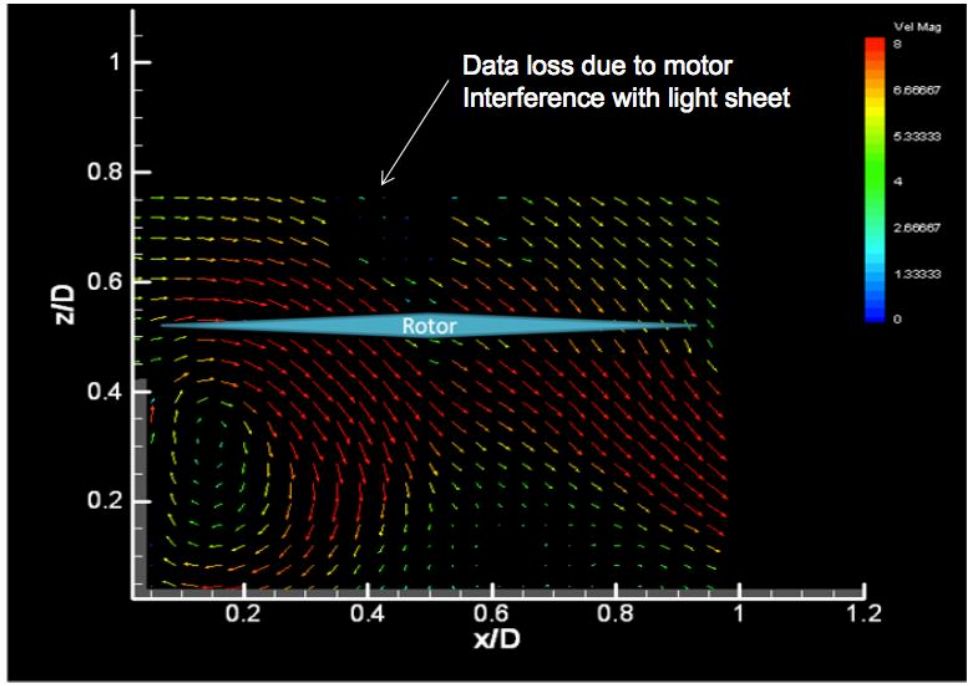
Particle Image Velocimetry (PIV)

- Flow over landing deck with rotor

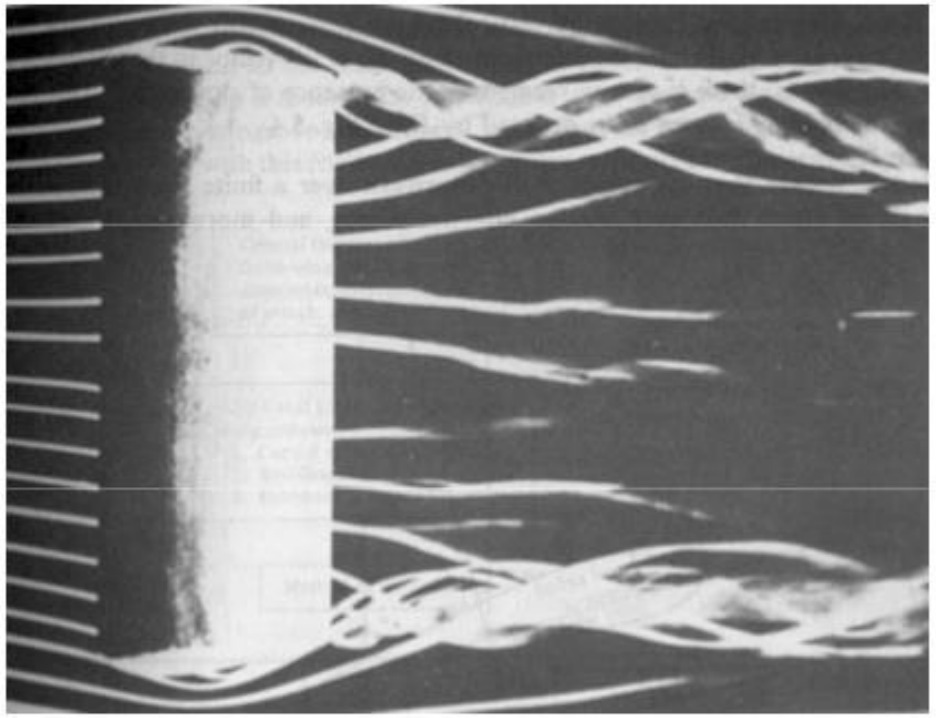


Particle Image Velocimetry (PIV)

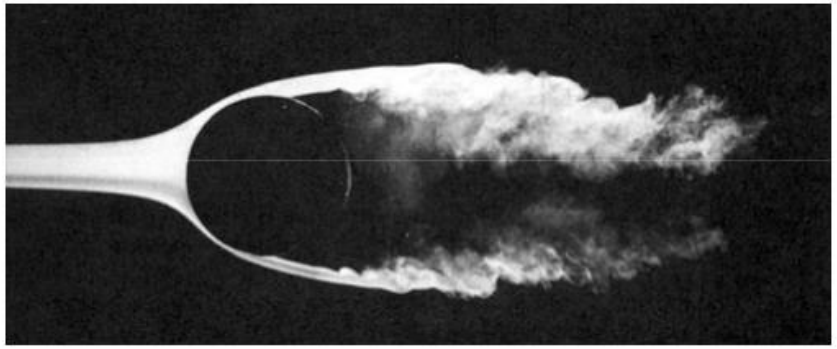
- Flow over landing deck with rotor



Freestream Tracer Injection



Oil dripped on array of wires shows wingtip vortex



Direct injection of smoke shows laminar separation on a cylinder in crossflow

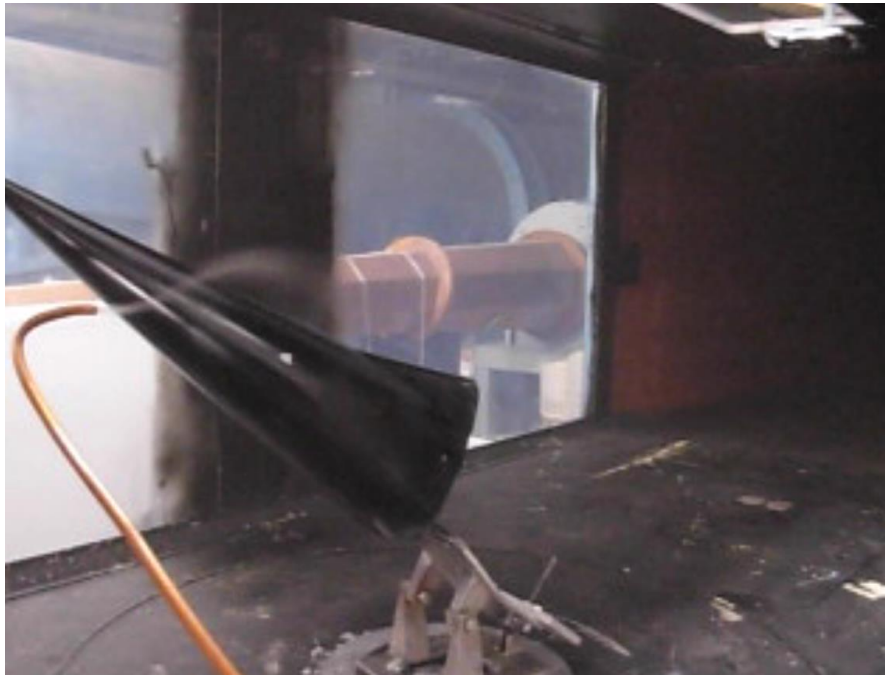
<u>Freestream Tracer Injection</u>

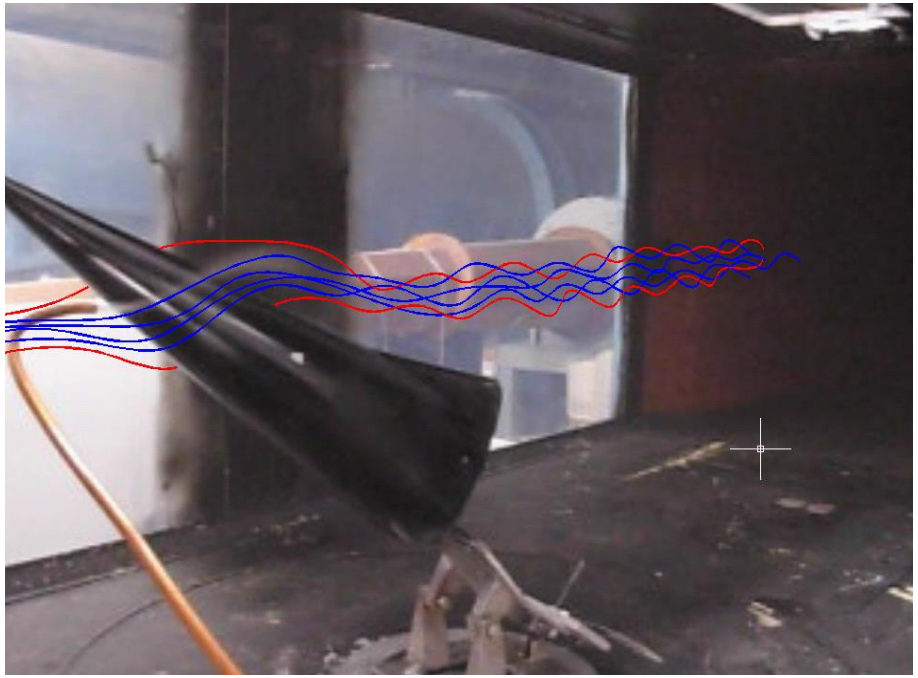
Propylene Glycol “Smoke” Generator

Wand used with full- scale automotive testing

Freestream Tracer Injection

Propylene Glycol “Smoke” Generator

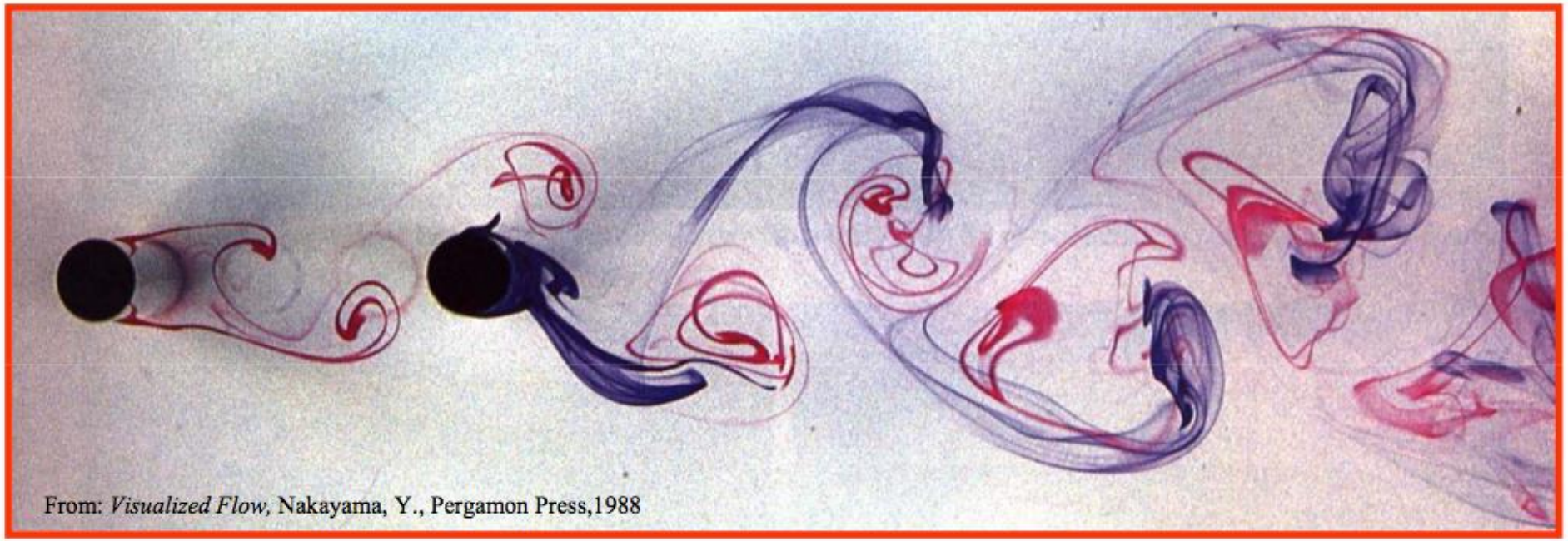




Forebody

Freestream Tracer Injection

Hydrodynamic (Dye)



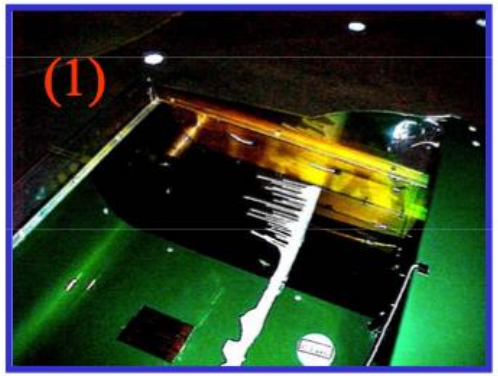
Karman vortex street following two cylinders

Freestream Tracer Injection

Oil FLow

Oil film stripe on automobile body panels

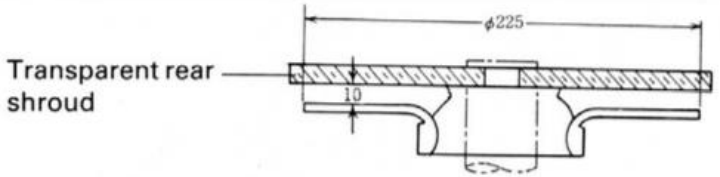
- shows interference of driver compartment on spoiler flow (1)
- interference of support strut on front flow deflector (2)



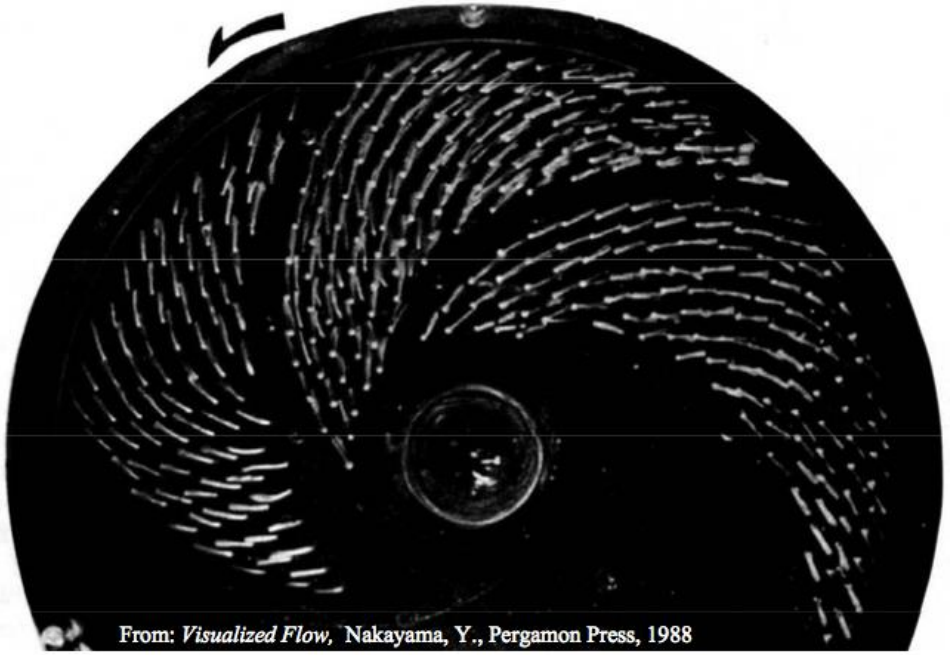
Freestream Tracer

Tufts

Procedure and materials are similar to aerodynamic methods

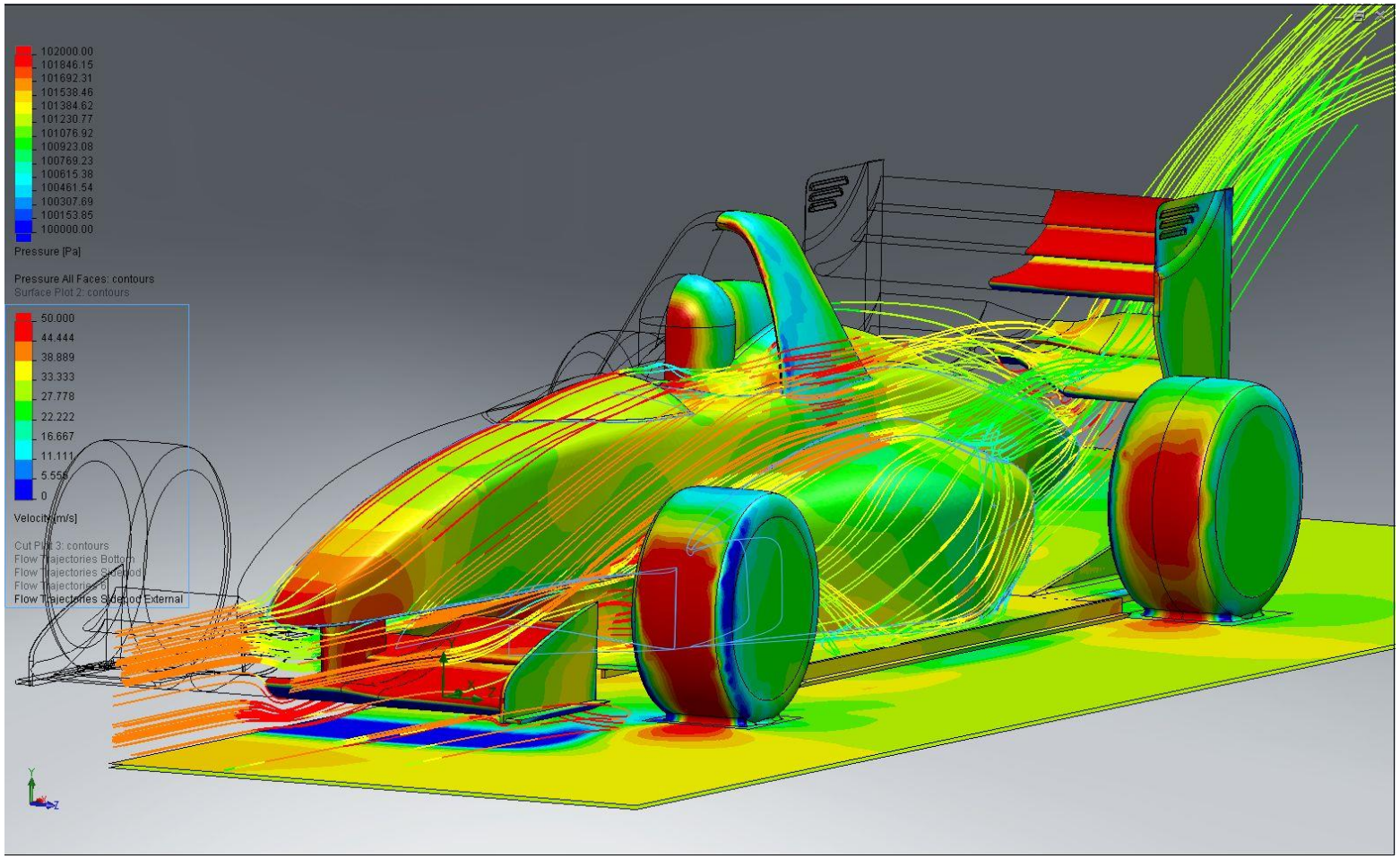


Example:
pump impeller rotating at 1200 rpm in water



From: *Visualized Flow*, Nakayama, Y., Pergamon Press, 1988

CFD Analysis Overview



Commercial CFD codes

- Fluent (UK and US).
- CFX (UK and Canada).
- Openfoam (US)
- Fidap (US).
- Polyflow (Belgium).
- Phoenix (UK).
- Star CD (UK).
- Flow 3d (US).
- ESI/CFDRC (US).
- SCRYU (Japan).
- and more, see www.cfdreview.com.

Introduction

All CFD simulations follow the same key stages. This lecture will explain how to go from the original planning stage to analyzing the end results

Learning Aims:

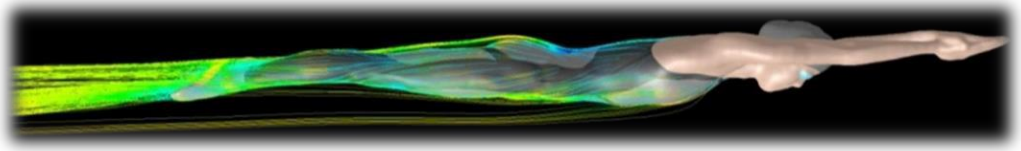
You will learn:

- The basics of what CFD is and how it works
- The different steps involved in a successful CFD project

Learning Objectives:

When you begin your own CFD project, you will know what each of the steps requires and be able to plan accordingly

What is CFD?



Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena.

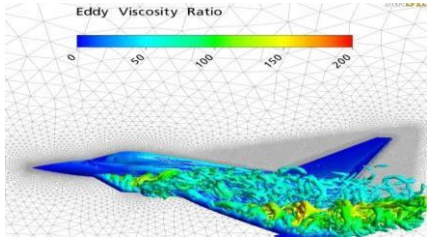
To predict these phenomena, CFD solves equations for conservation of mass, momentum, energy etc..

CFD can provide detailed information on the fluid flow behavior:

- Distribution of pressure, velocity, temperature, etc.
- Forces like Lift, Drag.. (external flows, Aero, Auto..)
- Distribution of multiple phases (gas-liquid, gas-solid..)
- Species composition (reactions, combustion, pollutants..)
- Much more

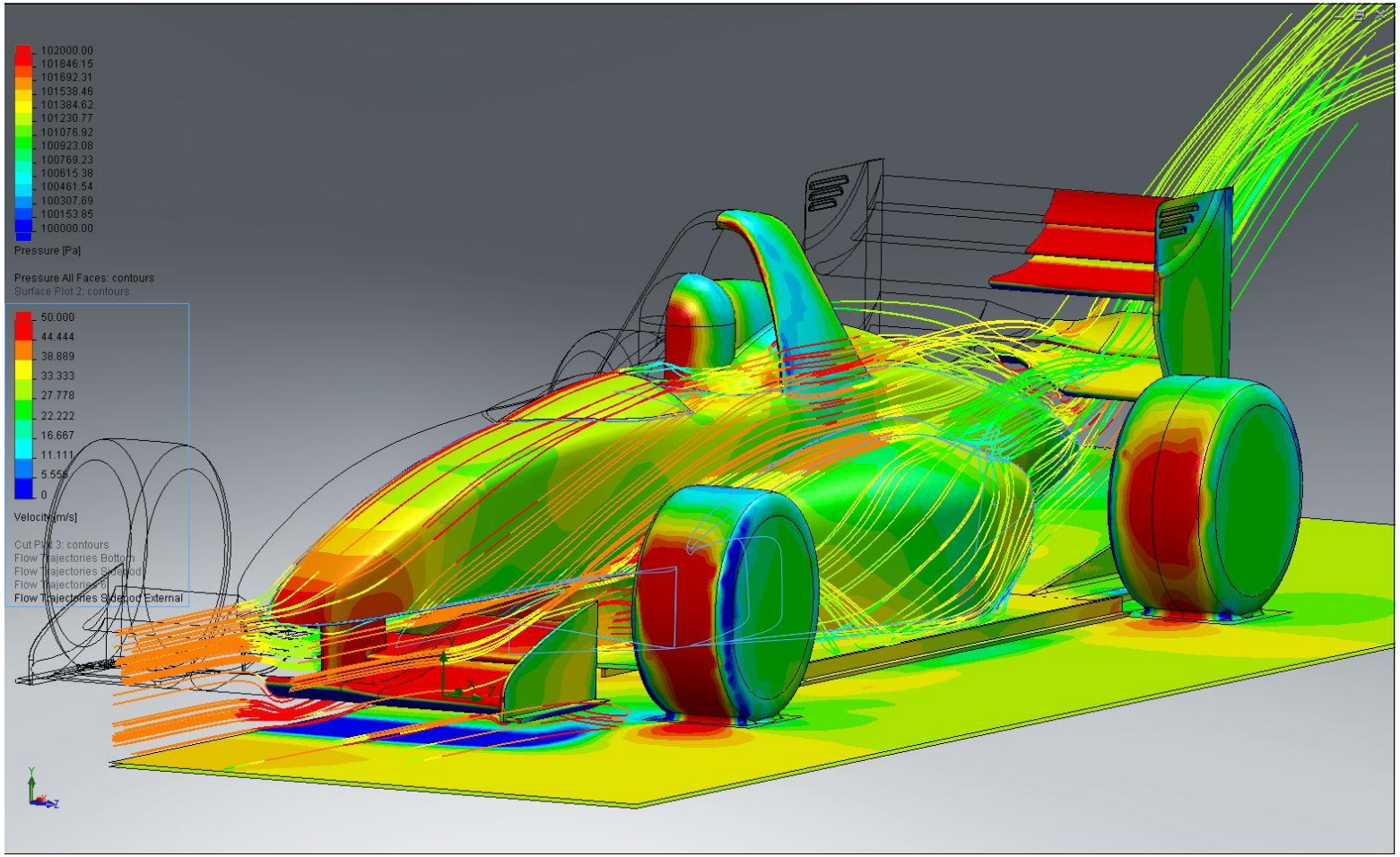
CFD is used in all stages of the engineering process:

- Conceptual studies of new designs
- Detailed product development
- Optimization
- Troubleshooting
- Redesign

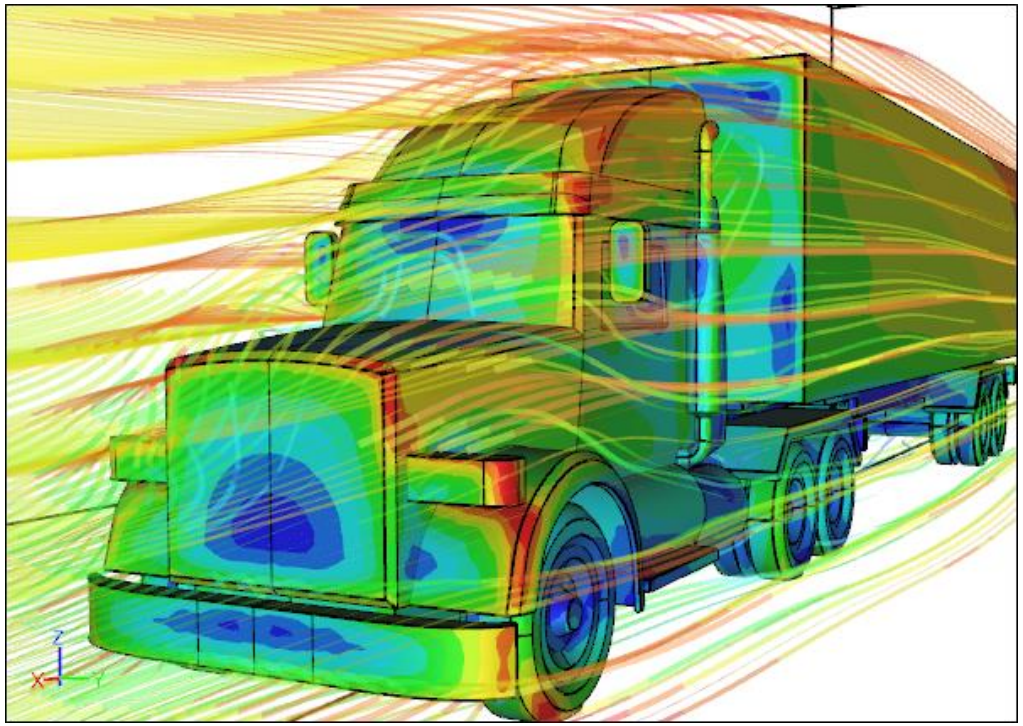


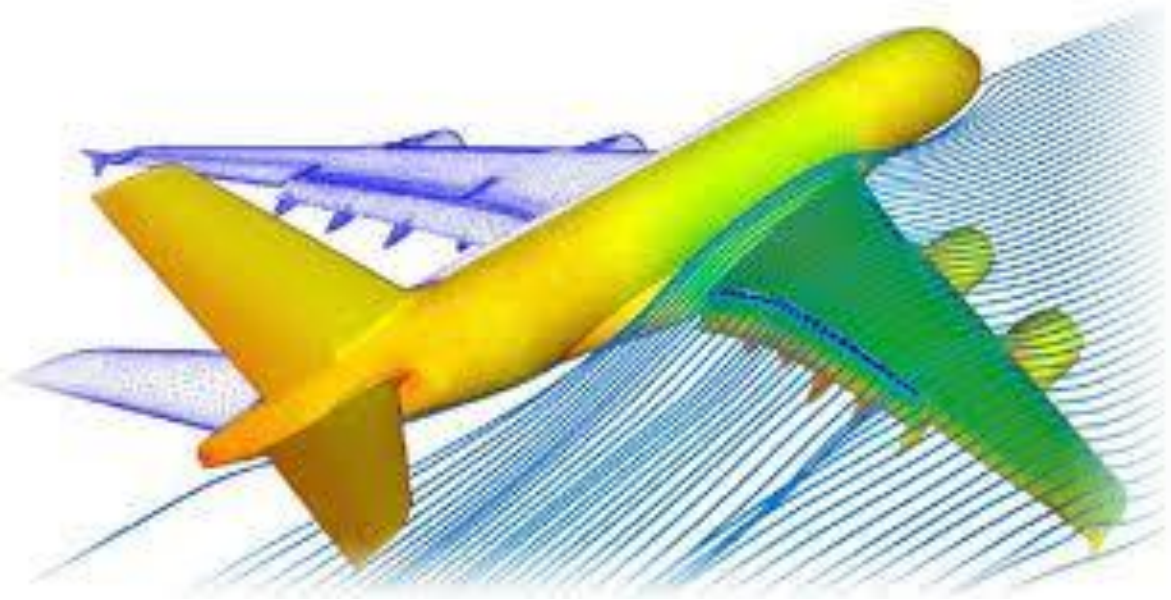
CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition

CFD Applications

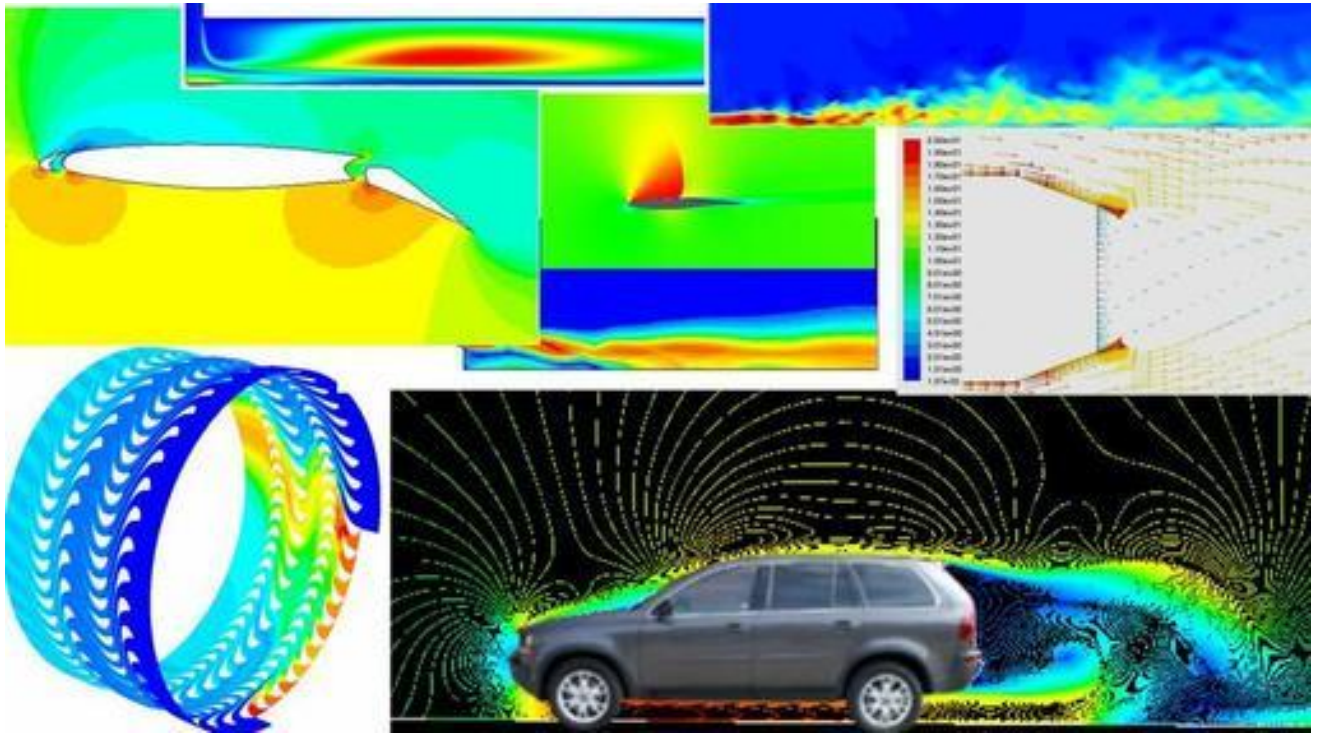


CFD Applications

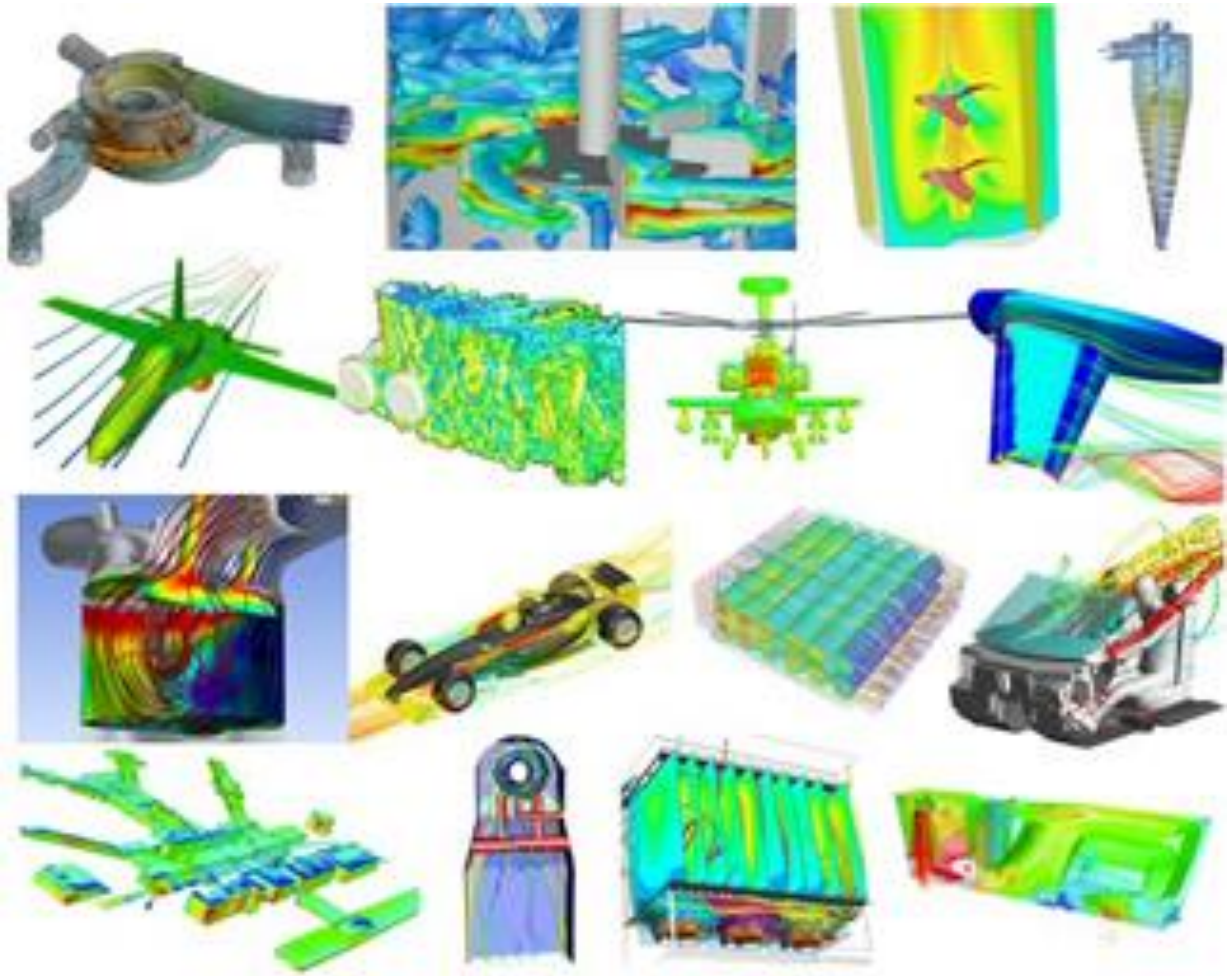




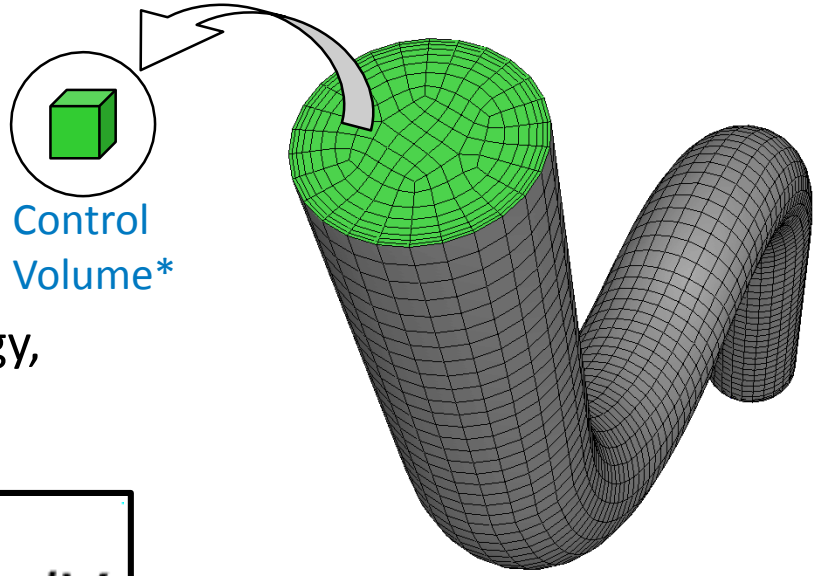
CFD Applications



CFD Applications



How Does CFD Work?



ANSYS CFD solvers are based on the finite volume method

- Domain is discretized into a finite set of control volumes
- General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

Equation	ϕ
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

- Partial differential equations are discretized into a system of algebraic equations
- All algebraic equations are then solved numerically to render the solution field

Step 1. Define Your Modeling Goals

- **What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?**

- **What are your modeling options?**
  - What simplifying assumptions **can you make** (i.e. symmetry, periodicity)?
  - What simplifying assumptions do you **have to make**?
  - What physical models will need to be included in your analysis

- **What degree of accuracy is required?**

- **How quickly do you need the results?**

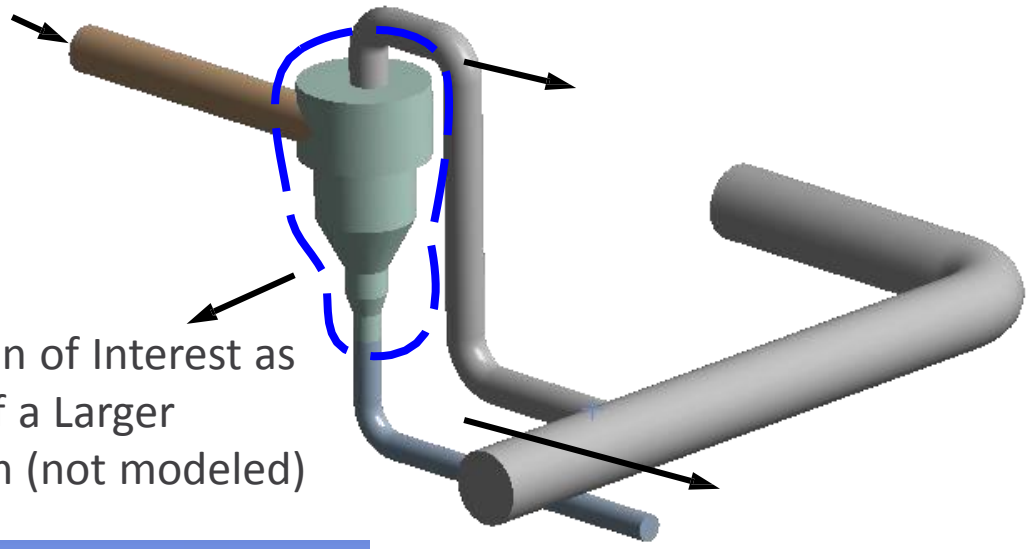
- **Is CFD an appropriate tool?**

Step 2. Identify the Domain You Will Model

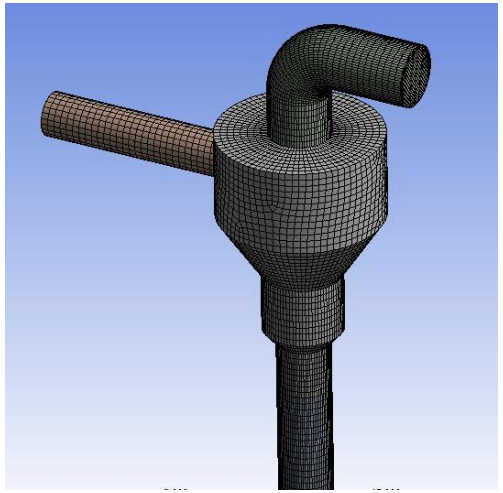
- **How will you isolate a piece of the complete physical system?**

- **Where will the computational domain begin and end?**
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?

- **Can it be simplified or approximated as a 2D or axi-symmetric problem?**



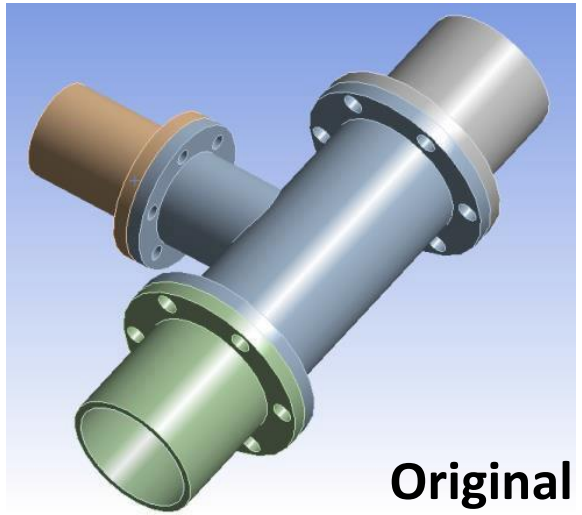
Domain of Interest as Part of a Larger System (not modeled)



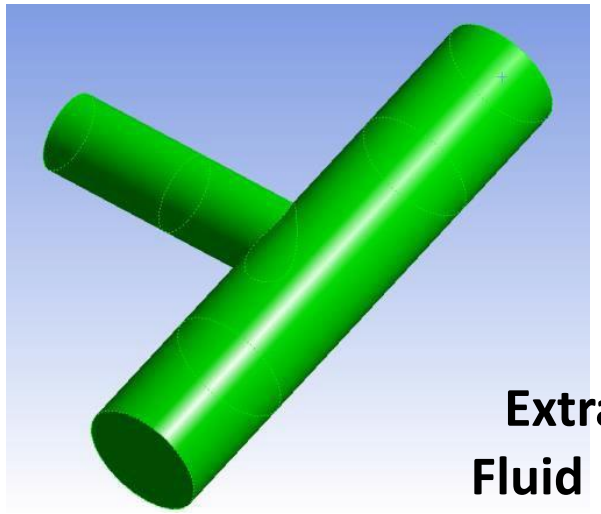
Domain of interest isolated and meshed for CFD simulation.

Step 3. Create a Solid Model of the Domain

- **How will you obtain a model of the *fluid* region?**
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?
- **Can you simplify the geometry?**
  - Remove unnecessary features that would complicate meshing ;fillets, bolts…?
  - Make use of symmetry or periodicity?
    - Are both the flow <u>and</u> boundary conditions symmetric / periodic?
- **Do you need to split the model so that boundary conditions or domains can be created?**



Original CAD Part



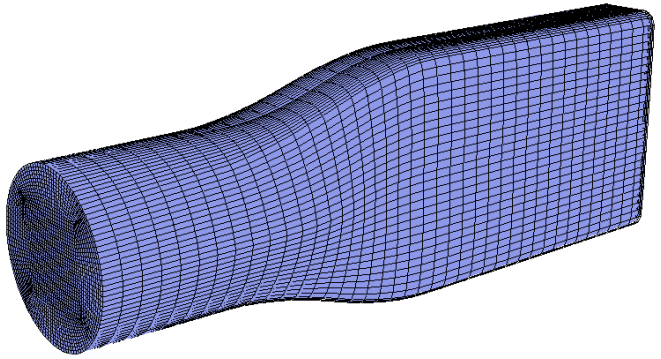
Extracted Fluid Region

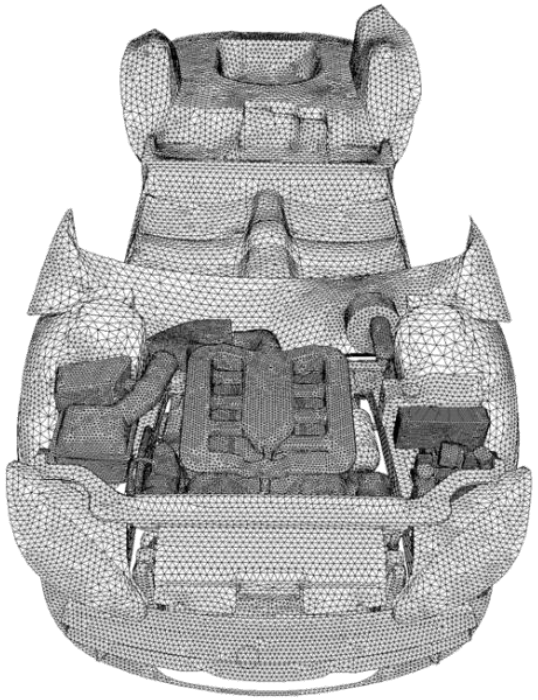
Step 4. Design and Create the Mesh

- **What degree of mesh resolution is required in each region of the domain?**
  - Can you predict regions of high gradients?
    - The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients
  - Will you use adaption to add resolution?

- **What type of mesh is most appropriate?**
  - How complex is the geometry?
  - Can you use a quad/hex mesh or is a tri/tet or hybrid mesh suitable?
  - Are non-conformal interfaces needed?

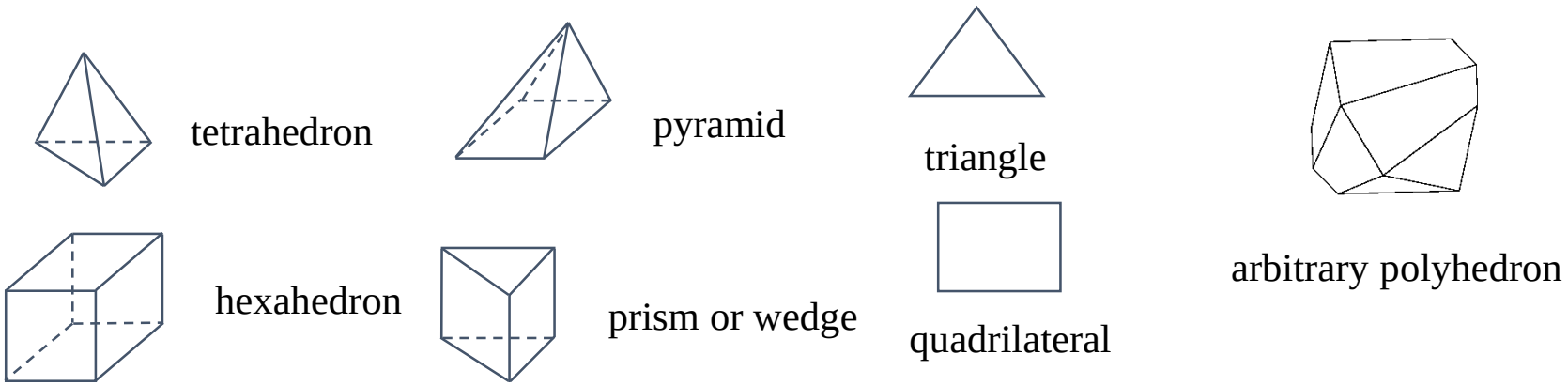
- **Do you have sufficient computer resources?**
  - How many cells/nodes are required?
  - How many physical models will be used?



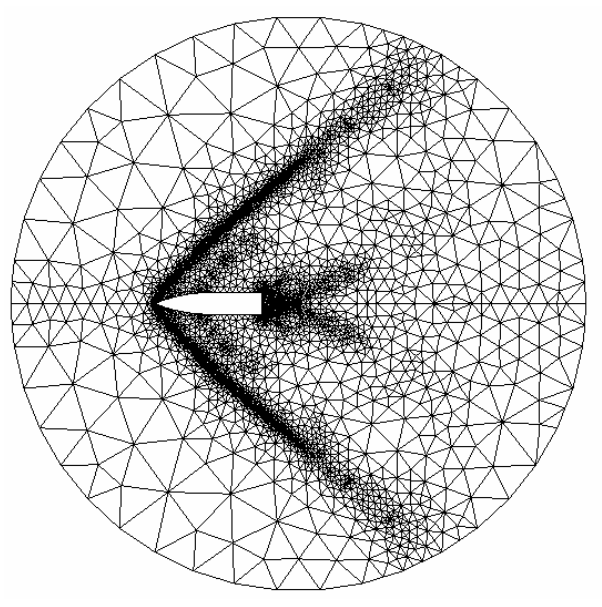


Design and create the grid

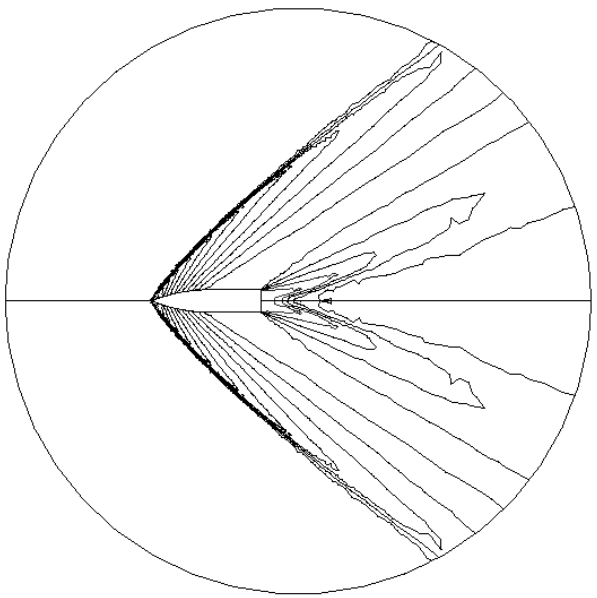
- Should you use a quad/hex grid, a tri/tet grid, a hybrid grid, or a non-conformal grid?
- What degree of grid resolution is required in each region of the domain?
- How many cells are required for the problem?
- Will you use adaption to add resolution?
- Do you have sufficient computer memory?



Adaption example: final grid and solution



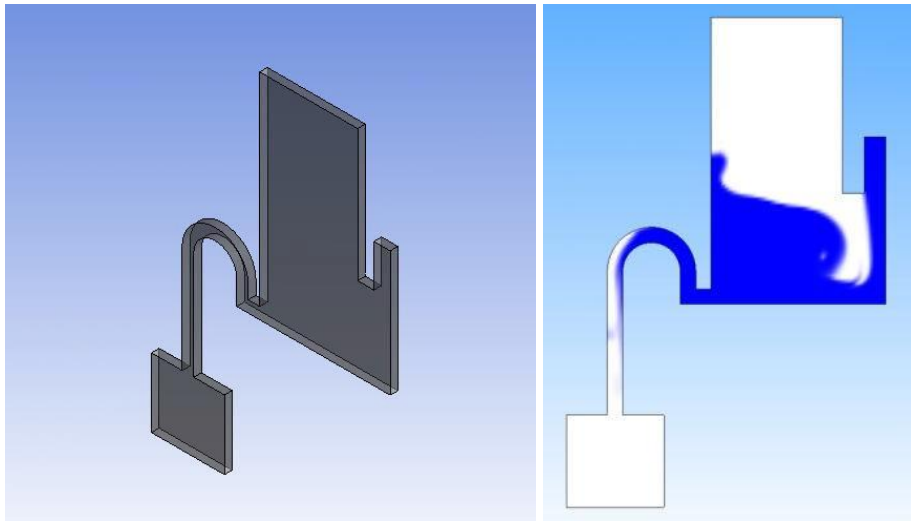
2D planar shell - final grid



2D planar shell - contours of pressure
final grid

Step 5. Set Up the Solver

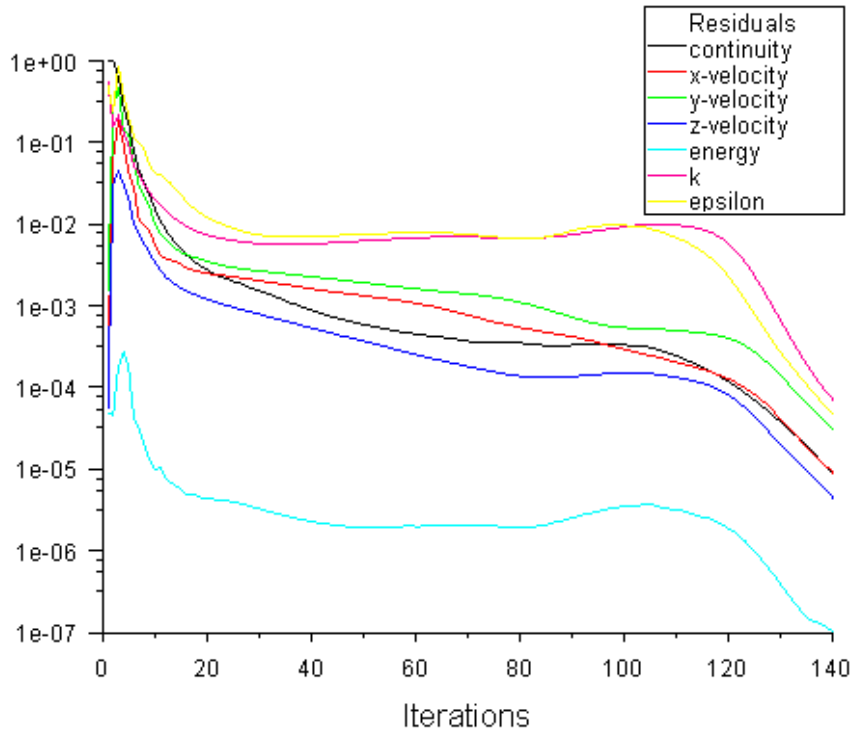
- **For a given problem, you will need to:**
  - Define material properties
    - Fluid
    - Solid
    - Mixture
  - Select appropriate physical models
    - Turbulence, combustion, multiphase, etc.
  - Prescribe operating conditions
  - Prescribe boundary conditions at all boundary zones
  - Provide initial values or a previous solution
  - Set up solver controls
  - Set up convergence monitors



For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time

Step 6. Compute the Solution

- **The discretized conservation equations are solved iteratively until convergence**
- **Convergence is reached when:**
  - Changes in solution variables from one iteration to the next are negligible
    - Residuals provide a mechanism to help monitor this trend
  - Overall property conservation is achieved
    - Imbalances measure global conservation
  - Quantities of interest (e.g. drag, pressure drop) have reached steady values
    - Monitor points track quantities of interest
- **The accuracy of a *converged* solution is dependent upon:**
  - Appropriateness and accuracy of physical models
  - Assumptions made
  - Mesh resolution and independence
  - Numerical errors



A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

Step 7. Examine the Results

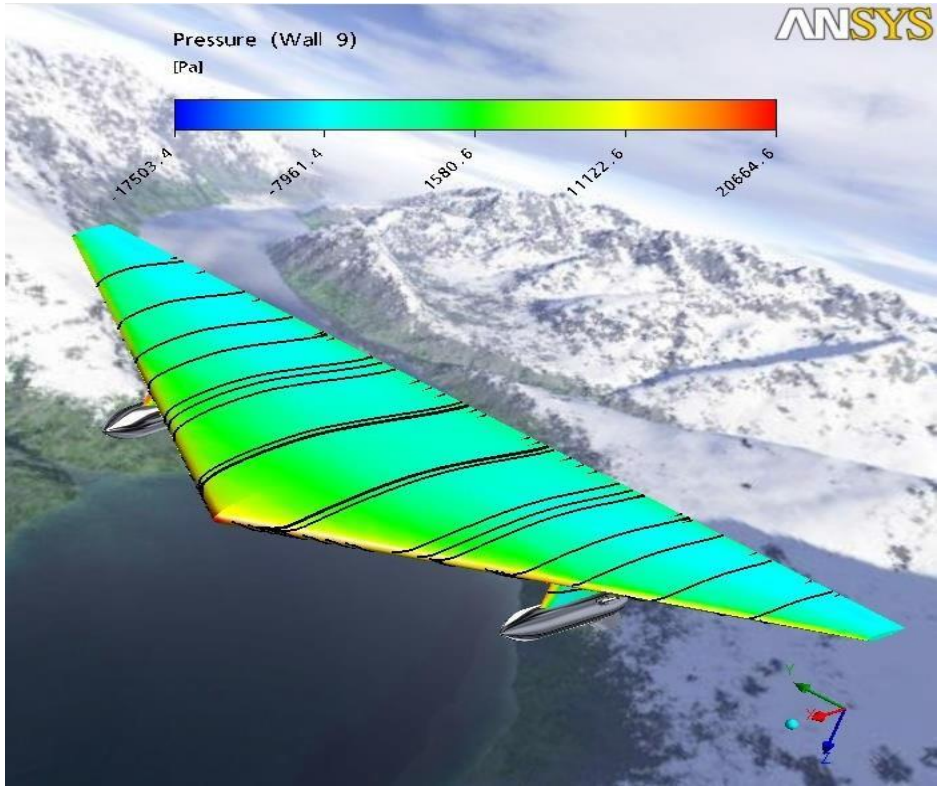
- **Examine the results to review solution and extract useful data**

– Visualization Tools can be used to answer such questions as:

  - What is the overall flow pattern?
  - Is there separation?
  - Where do shocks, shear layers, etc. form?
  - Are key flow features being resolved?

– Numerical Reporting Tools can be used to calculate quantitative results:

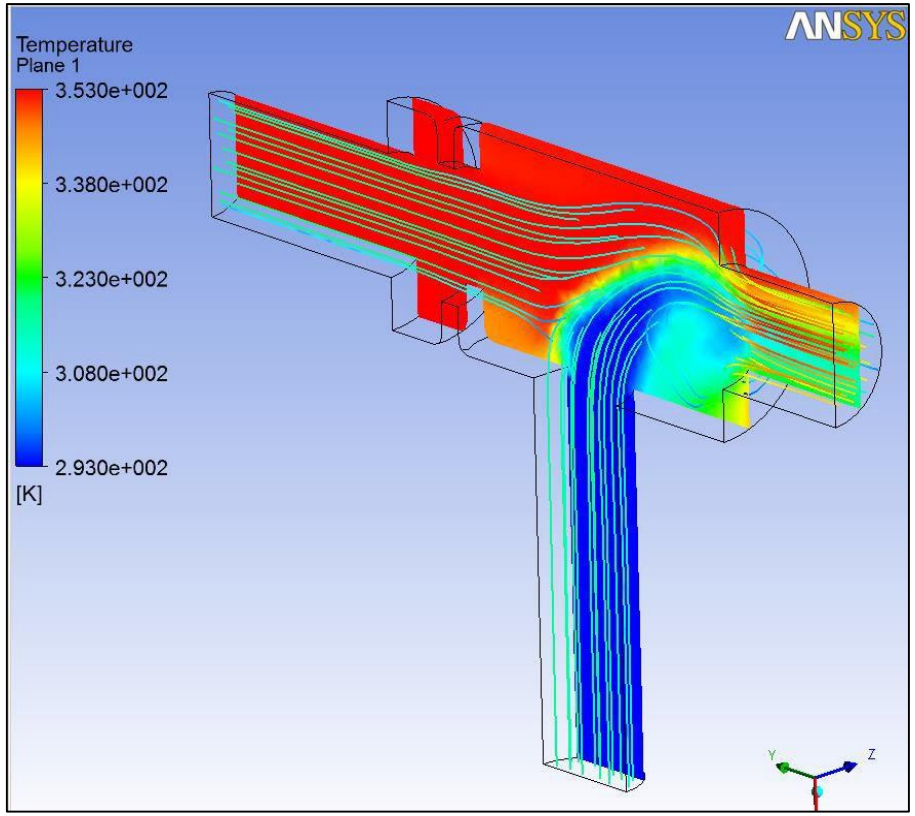
  - Forces and Moments
  - Average heat transfer coefficients
  - Surface and Volume integrated quantities
  - Flux Balances



Examine results to ensure correct physical behavior and conservation of mass energy and other conserved quantities. High residuals may be caused by just a few poor quality cells.

Step 8. Consider Revisions to the Model

- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- **Is the mesh adequate?**
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?
  - Does the model contain poor quality cells?



High residuals may be caused by just a few poor quality cells

Use CFD with Other Tools to Maximize its Effect

Problem Identification

1. Define goals
2. Identify domain

Pre-Processing

3. Geometry
4. Mesh Physics
5. Solver Settings
6.

Solve

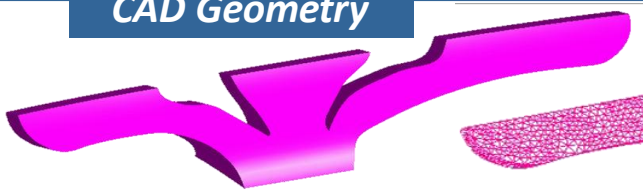
7. Compute solution

Post Processing

8. Examine results

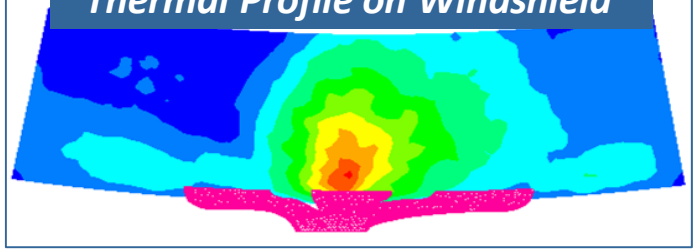
9. Update Model

CAD Geometry



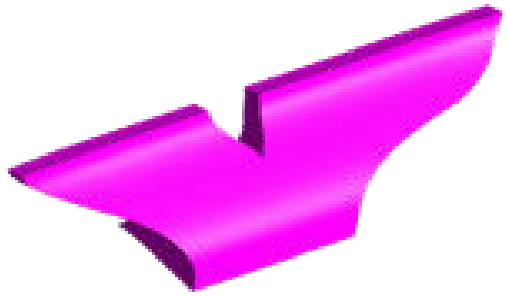
Mesh

Thermal Profile on Windshield



Automated Optimization of Windshield Defroster with ANSYS DesignXplorer

Final Optimized Design



Prototype Testing → *Manufacturing*

Summary and Conclusions

- **Summary:**
  - All CFD simulations (in all mainstream CFD software products) are approached using the steps just described
  - Remember to first think about what the aims of the simulation are prior to creating the geometry and mesh
  - Make sure the appropriate physical models are applied in the solver, and that the simulation is fully converged (more in a later lecture)
  - Scrutinize the results, you may need to rework some of the earlier steps in light of the flow field obtained

1. **Define Your Modeling Goals**
2. Identify the Domain You Will Model
3. **Create a Geometric Model of the Domain**
4. Design and Create the Mesh
5. **Set Up the Solver Settings**
6. Compute the Solution
7. **Examine the Results**
8. Consider Revisions to the Model

Applications of CFD

- Applications of CFD are numerous!
  - Flow and heat transfer in industrial processes (boilers, heat exchangers, combustion equipment, pumps, blowers, piping, etc.).
  - Aerodynamics of ground vehicles, aircraft, missiles.
  - Film coating, thermoforming in material processing applications.
  - Flow and heat transfer in propulsion and power generation systems.
  - Ventilation, heating, and cooling flows in buildings.
  - Chemical vapor deposition (CVD) for integrated circuit manufacturing.
  - Heat transfer for electronics packaging applications.
  - And many, many more!

Advantages of CFD

- Relatively low cost.
  - Using physical experiments and tests to get essential engineering data for design can be expensive.
  - CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.
- Speed.
  - CFD simulations can be executed in a short period of time.
  - Quick turnaround means engineering data can be introduced early in the design process.
- Ability to simulate real conditions.
  - Many flow and heat transfer processes can not be (easily) tested, e.g. hypersonic flow.
  - CFD provides the ability to theoretically simulate any physical condition.

Advantages of CFD (2)

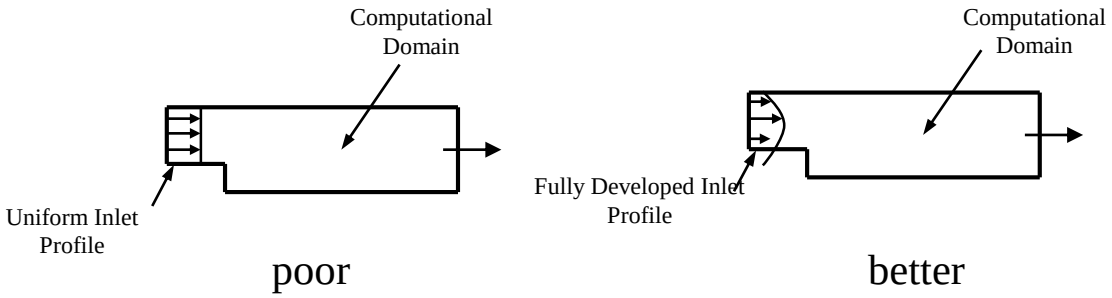
- Ability to simulate ideal conditions.
  - CFD allows great control over the physical process, and provides the ability to isolate specific phenomena for study.
  - Example: a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.
- Comprehensive information.
  - Experiments only permit data to be extracted at a limited number of locations in the system (e.g. pressure and temperature probes, heat flux gauges, LDV, etc.).
  - CFD allows the analyst to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.

Limitations of CFD

- Physical models.
  - CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).
  - The CFD solutions can only be as accurate as the physical models on which they are based.
- Numerical errors.
  - Solving equations on a computer invariably introduces numerical errors.
  - Round-off error: due to finite word size available on the computer. Round-off errors will always exist (though they can be small in most cases).
  - Truncation error: due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.

Limitations of CFD (2)

- Boundary conditions.
  - As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
  - Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.



Summary

- CFD is a method to numerically calculate heat transfer and fluid flow.
- Currently, its main application is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies.
- CFD can also be used for purely scientific studies, e.g. into the fundamentals of turbulence. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem.