Carnegie Mellon University

Week 14 - Lecture
CFD Analysis

Lecture Topics

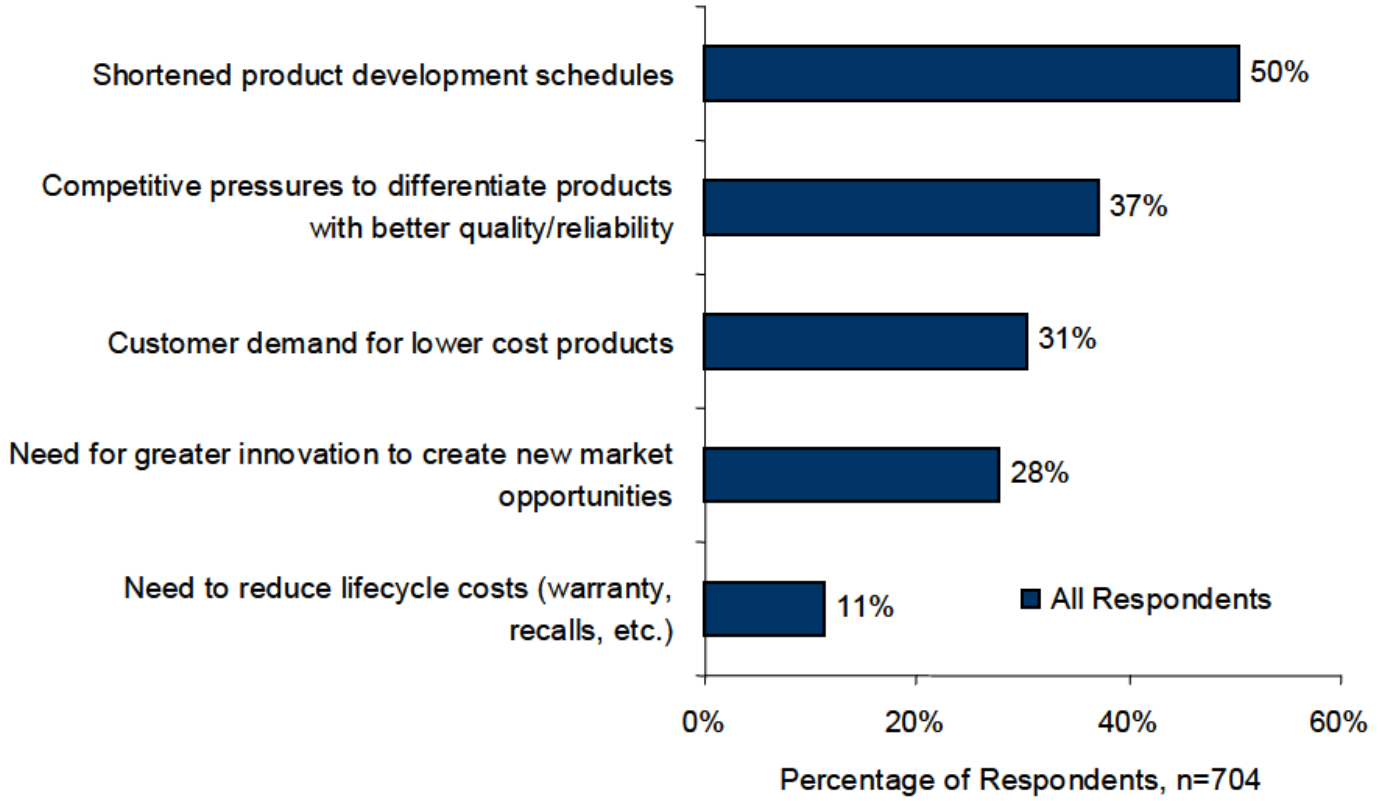
- Team Project 2 Discussion
- Simulation Drivers
- CFD Overview
- CFD Use Cases and Examples
- Common Requirements for CFD
- Introduction to Autodesk CFD Products

Simulation

- Simulation offers the potential to improve products by predicting behavior digitally.

Understanding Product Behavior

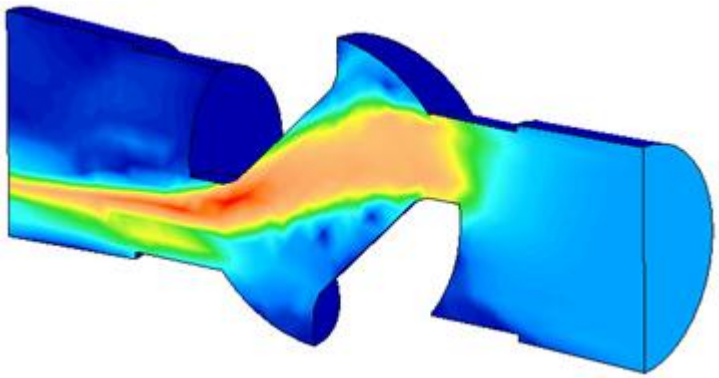
Business needs and challenges of understanding product behavior.



Source: Aberdeen Group: How Best-in-Class Companies Amplify Engineering with CFD, April 2011

CFD Overview

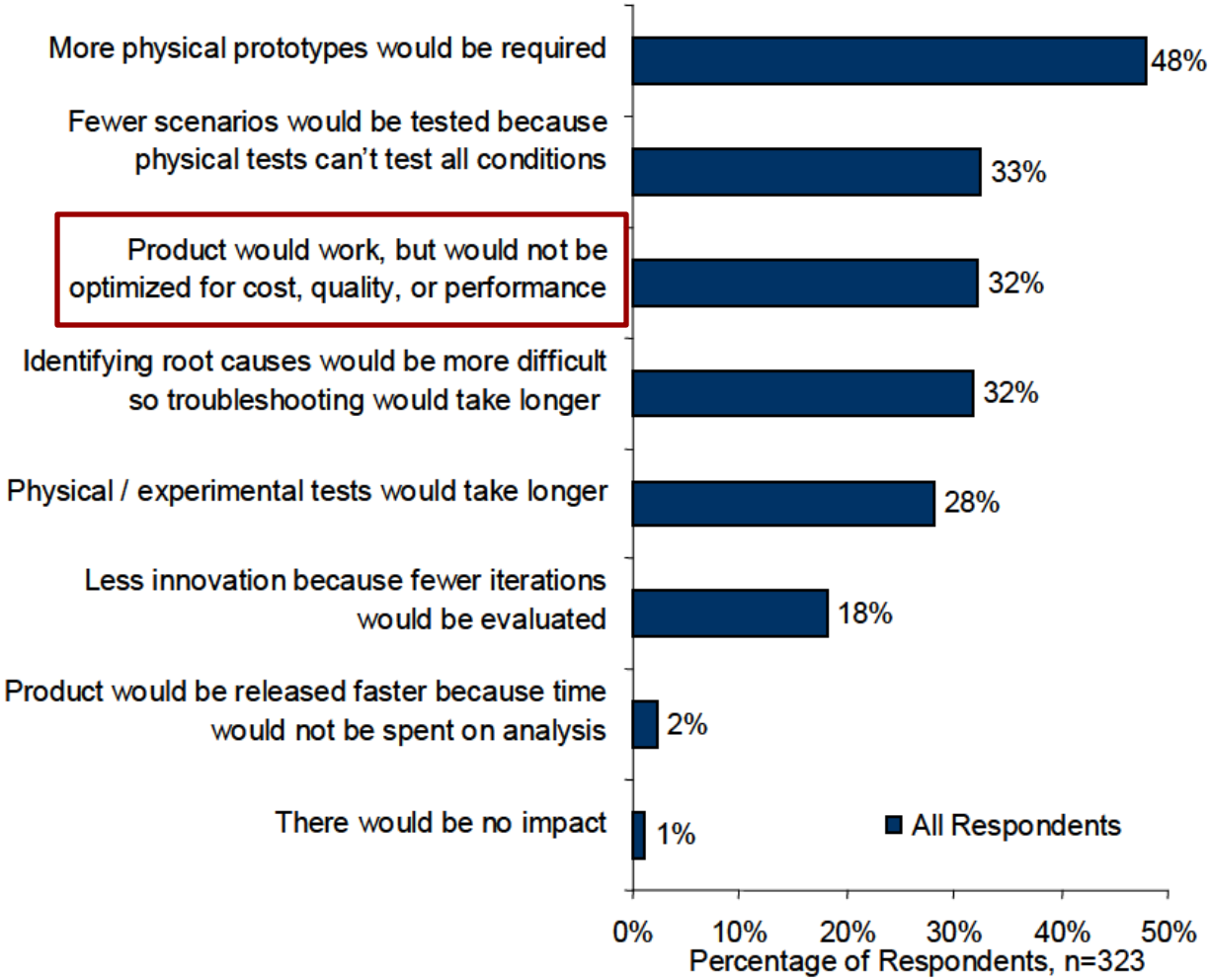
- Computational Fluid Dynamics (CFD) is a specialized simulation that analyzes fluid flow. Used to analysis the interaction of liquids and gases with surfaces defined by boundary conditions.



Products that Benefit from CFD

- Liquid or Gas Flow
- Heating and Cooling
- Chemical Reactions
- Turbulence
- Aerodynamics

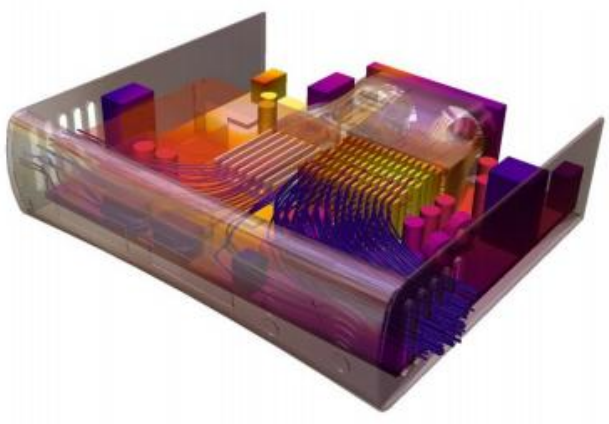
Leading Impact of Not Using CFD

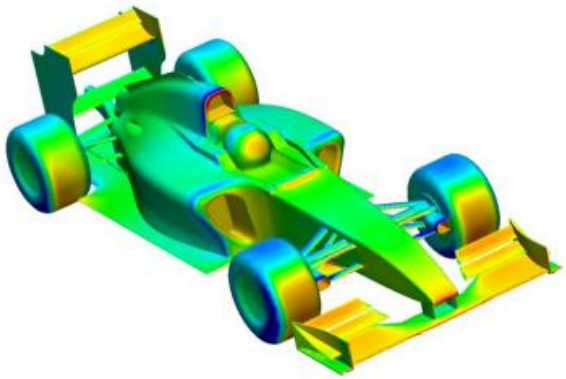


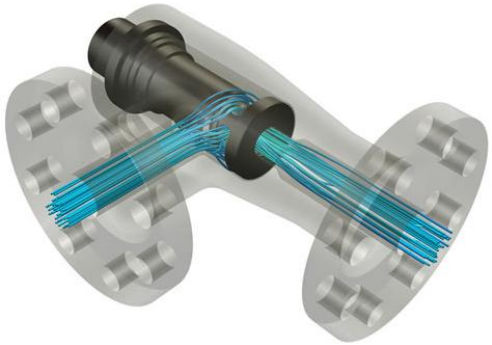
Source: Aberdeen Group: How Best-in-Class Companies Amplify Engineering with CFD, April 2011

CFD Use Case Examples

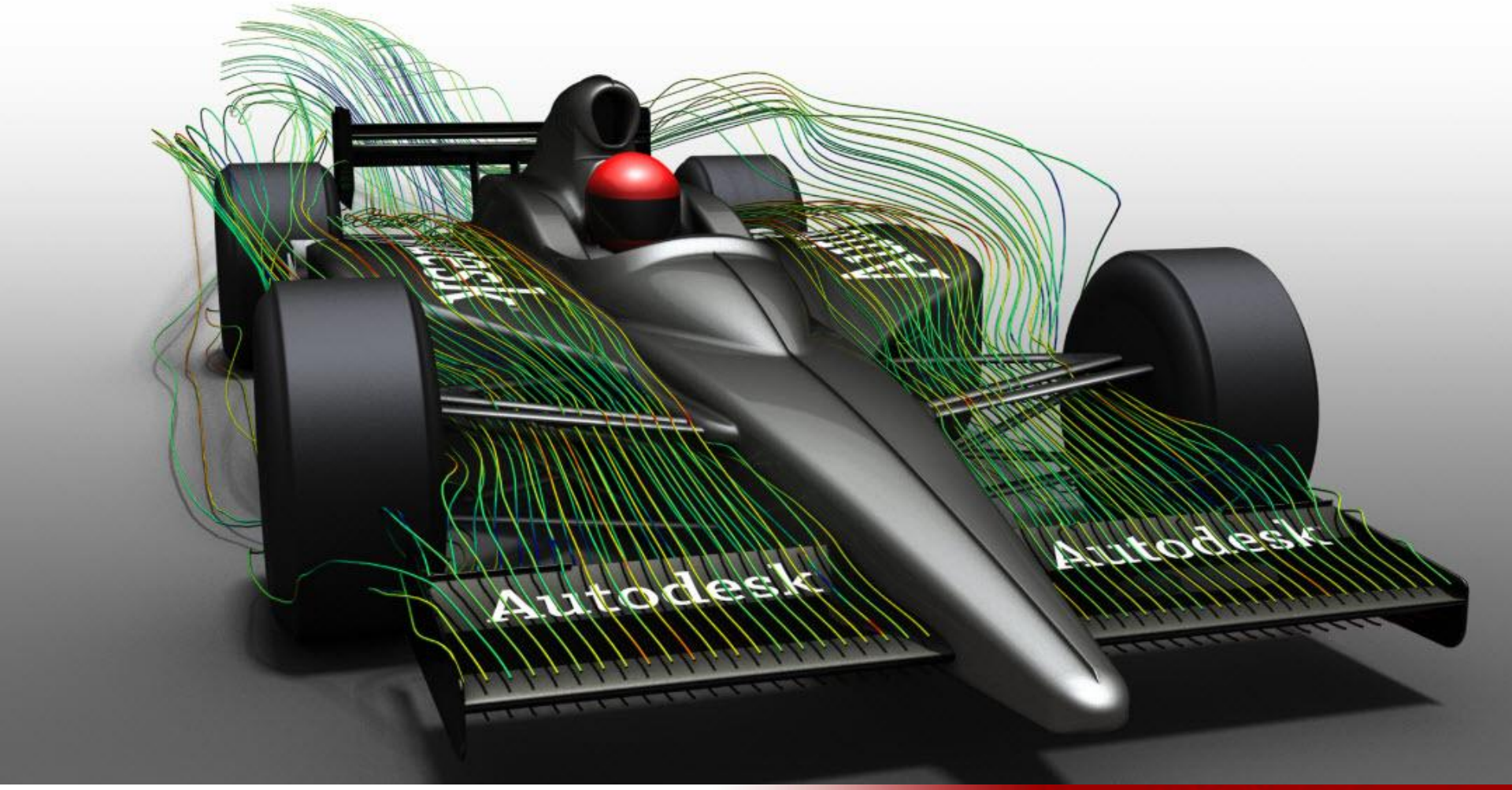
- Airflow cooling of consumer electronics.
- Aerodynamics and down force of a vehicle.
- Fluid flow efficiency through piping valve.



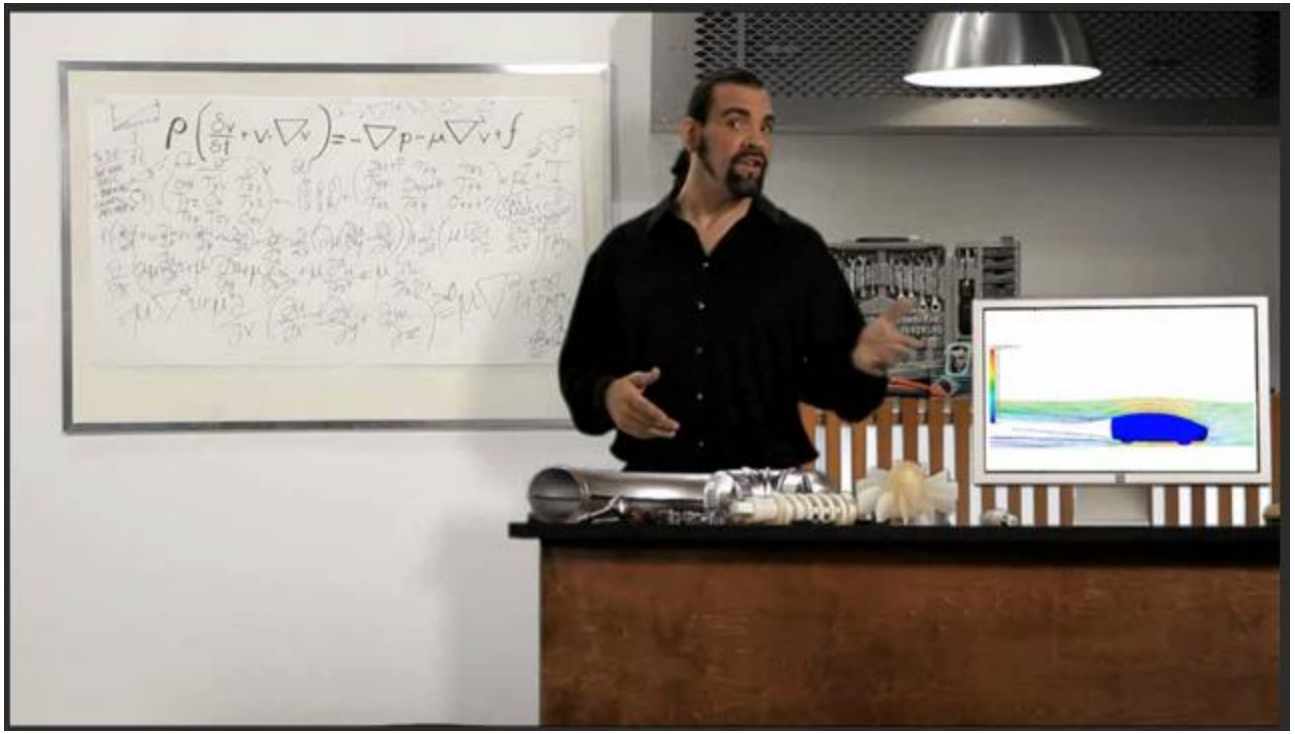




CFD Example



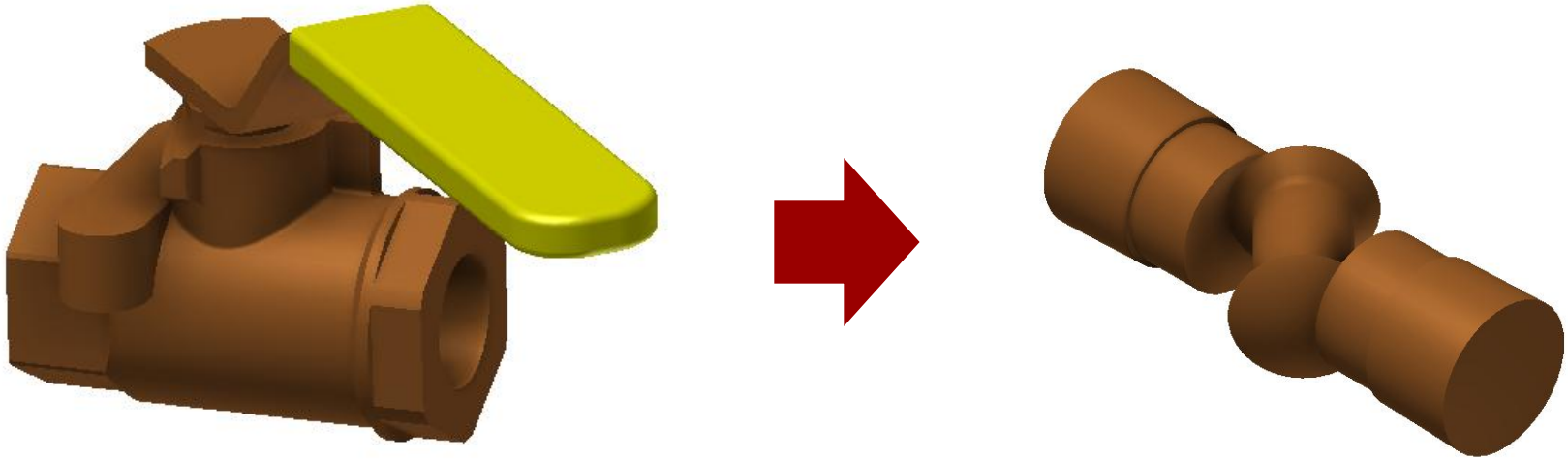
Reducing Energy Loss in Design



http://sustainabilityworkshop.autodesk.com/strategy/fluid-dynamics

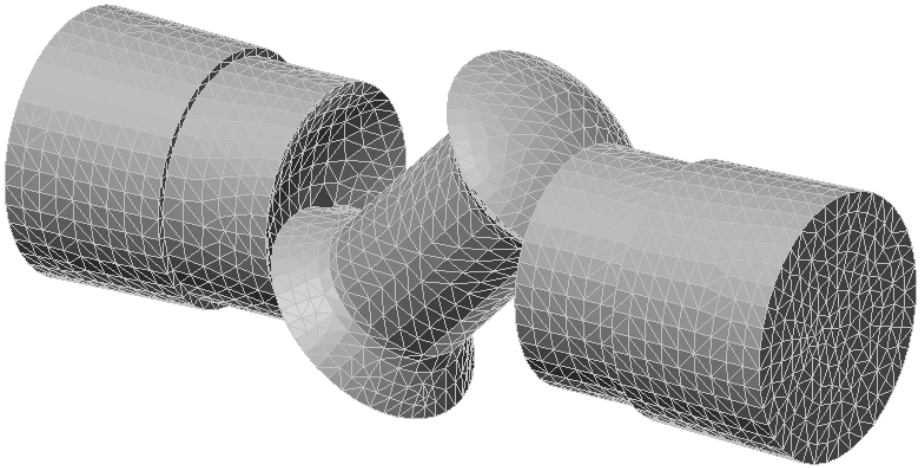
Fluid Model

- There needs to be a model created that represents the fluid volume.



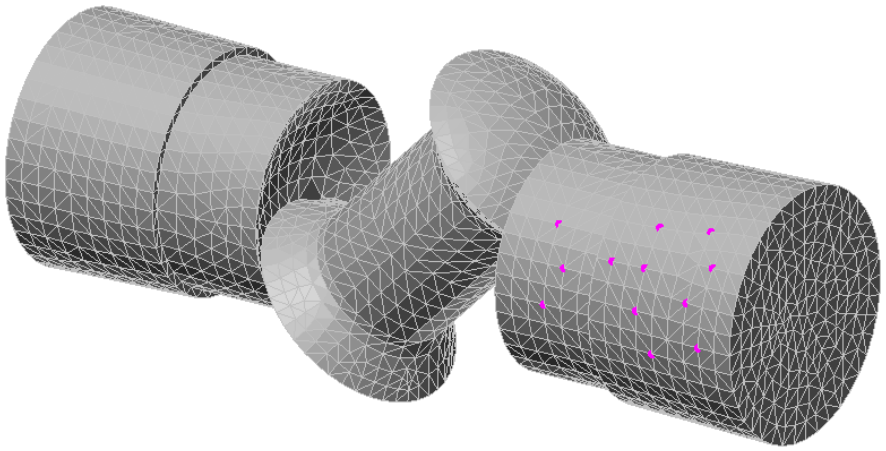
CFD Elements

- Fluid flow analysis generally support 2D and 3D elements. 3D fluid flow elements have 4, 5, 6, and 8 node elements like bricks.



3D Fluid Flow Element Nodes

- Each node on a 3D fluid flow element has four (4) DOFs. These are the velocity components in the X, Y, and Z and the pressure.



Model Mesh (Boundary Layer)

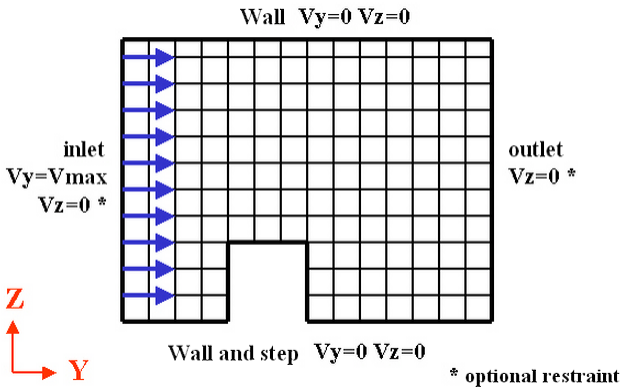
- All Tetrahedra
  - Using all tetrahedra elements on a 3D model ensures high quality interior mesh which is important for most fluid flow analysis. Creating all 4 node elements can sometimes block a small area near the exterior of the model impacting results.
- Tetrahedra and Wedge (Boundary Layer)
  - This option allows for a boundary mesh of just wedge elements to be created at all of the exterior surfaces of the model. Tetrahedra elements will then be used for the rest of the interior. This helps capture the results more accurately around the walls of the model and ensure no small areas are blocked with a single node.

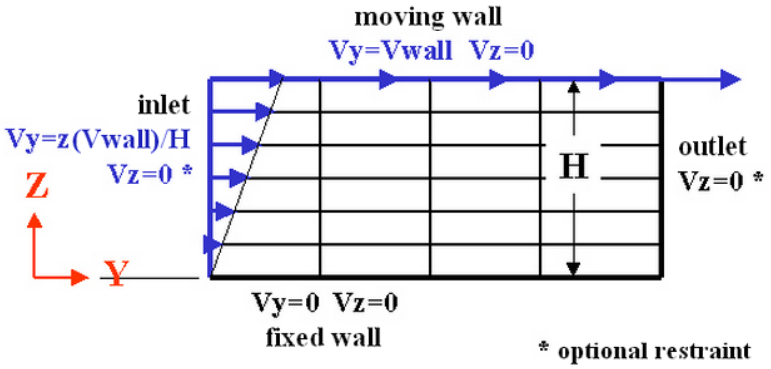
Common CFD Materials

- Liquids
  - Water
  - Oils
  - Ethanol
- Gases
  - Air
  - Nitrogen
  - Oxygen

Common Fluid Flow Loads

- Inlet / Outlet
  - Specifies an input or output surface in most cases for which the velocity is unknown and a zero-traction state is applied.
- Prescribed Velocity
  - Prescribed velocities establish the boundary conditions of a fluid flow problem. Control and set the velocities for the X, Y, or Z DOFs of the selected surface nodes.



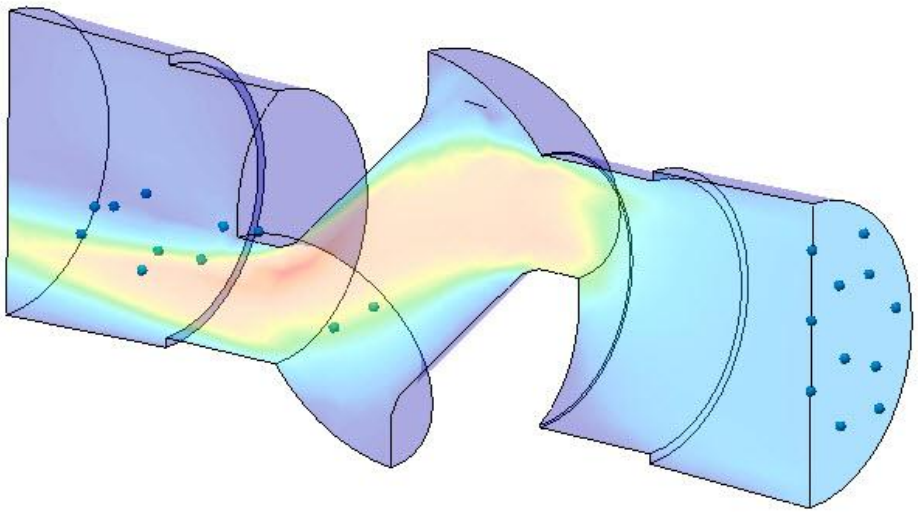


Common Results Types Provided

- Velocity
  - Displays the speed in a given direction of the fluid throughout the model.
- Reaction Forces
  - Displays the reaction forces in the fluid model.
- Pressure
  - Displays the nodal based pressure results of the model.
- Flow Rate
  - Volumetric flow rate results provide positive values for flow into an element and negative rates represents flow out of the element.

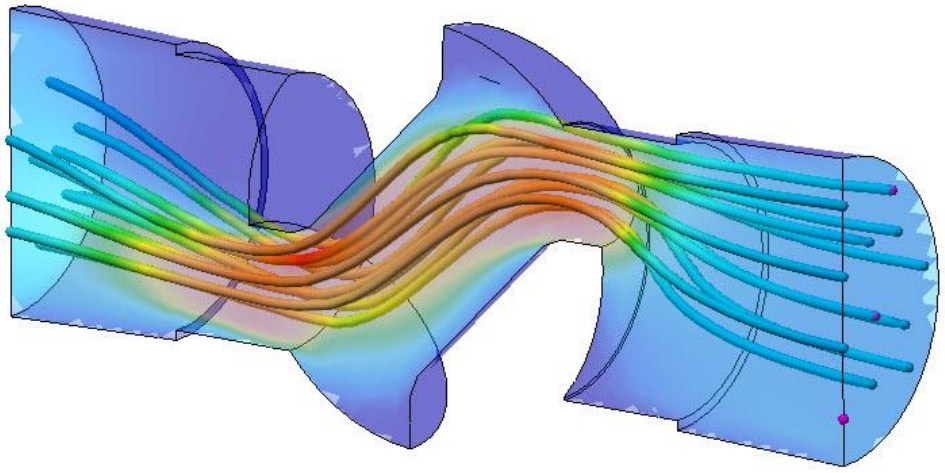
Particle Paths

- Particle paths track the movement of a massless particle in the fluid flow model from select nodes.



Streamlines

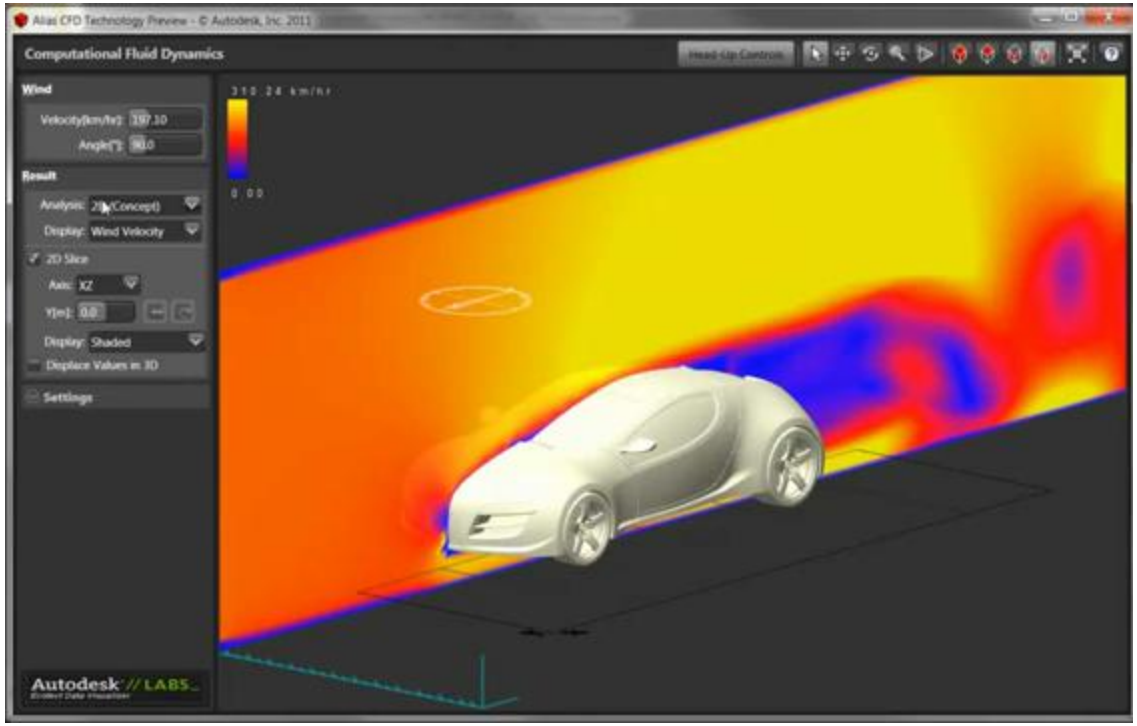
- Streamlines show the flow through a node during a fluid flow analysis from selected nodes.



Autodesk CFD Products

- Autodesk Simulation 
  - Full range of simulation tools including general CFD analysis.
- Autodesk Simulation CFD
  - Comprehensive set of tools for fluid flow and thermal simulation. (formerly known as CFdesign)
- Project Falcon Wind Tunnel Simulation
  - Autodesk Labs (labs.autodesk.com) free technology preview that provide wind tunnel simulation for aerodynamic performance.

Project Falcon



http://www.youtube.com/watch?v=UrrOA0rlaCg

Reminder - Motion in CFD Impact

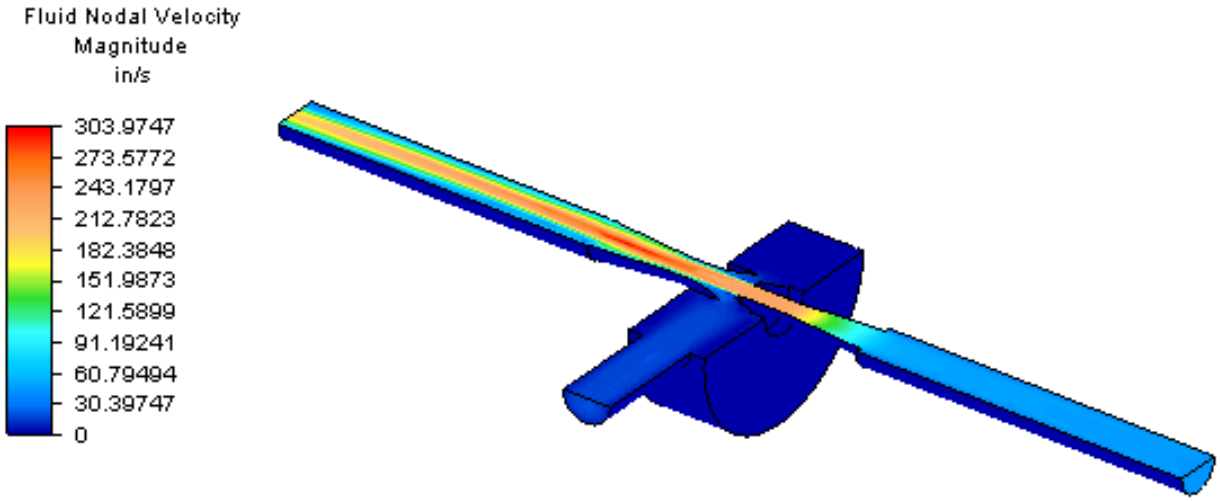
- Flow-Driven Motion
  - Object moves or stops in response to fluid impingement or resistance.
- Mechanical-Driven Motion
  - Object is in motion but does not react to the flow, but instead moves in a completely specified direction over a defined time and direction.

Limitations of CFD

- Separate Fluid Domains
  - No mixture of different fluids
- Viscous Fluids
  - Non-zero friction
- Incompressible Material
  - Constant Density
- Isothermal
  - Material properties are independent of temperature

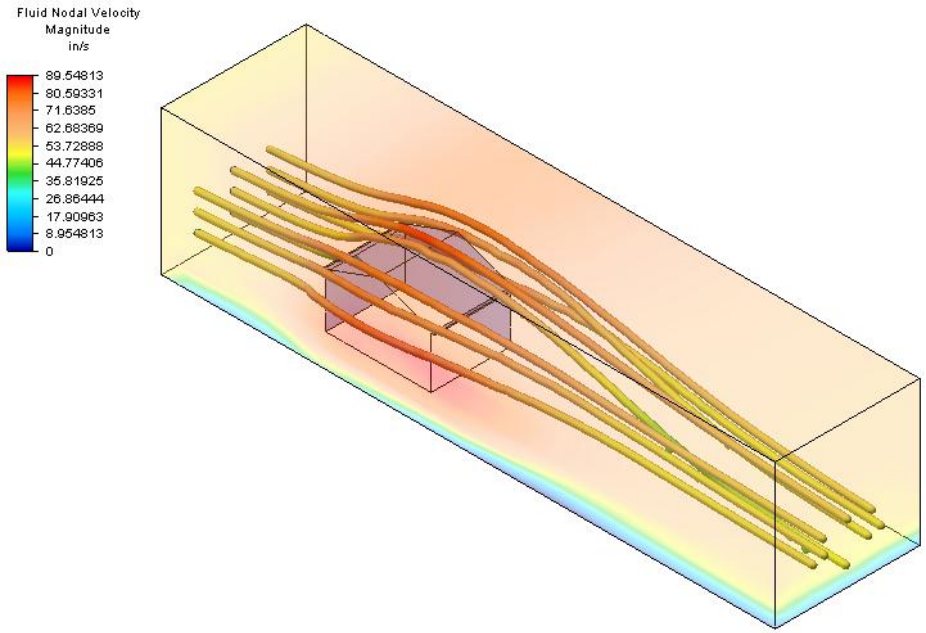
Guided Lab Project 1

- Guides instructions for completing an unsteady internal fluid flow analysis.



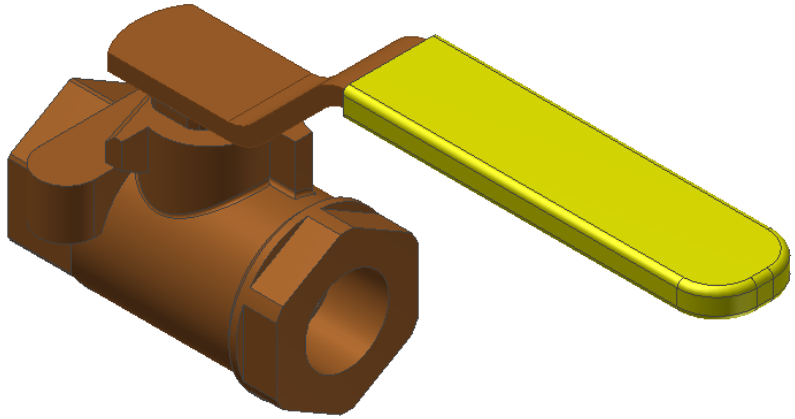
Guided Lab Project 2

- Guided instructions for completing an unsteady external fluid flow analysis.



Guided Lab Project 3

- Guided instructions for completing unsteady fluid flow with additional loading options for a value component.



Problem Set Assignment

- Complete CFD analysis on value assembly to determine impact of performance from improper assembly.