A Look at CFD: Applications in Chemical and Other Industries

Presenter:
Hsin-Hua Tsuei, Ph.D.
» CFD and the Chemical Industry
» Industry Challenges
» How can CFD help
» The Guts of CFD
» What is CFD up to These Days
» Summary
Mixing Tank Analysis

- CFD has been applied to mixing tank design and analyses successfully
  - Geometry: impeller design, baffles, single or multiple shafts, tank shapes
  - Materials properties
  - Newtonian or non-Newtonian fluids
  - Temperature dependent viscosity
  - Mixing of multiple materials
Case Study: Solid Particles in Tank

- Experimental setup of Montante et al. (2001)
  - 4 PBT, $D/T = 0.405$
  - $C = T/2, S = T$
  - $T = 0.232 \text{ m}, H/T = 4$
  - Glass Particles, 130 mm to 790 mm, 1–5 g/L
  - Water
  - $N = 1020 \text{ rpm}$

- Experimental data of Micheletti et al. 2003
  - $T = 0.29 \text{ m}, D = 0.098 \text{ m}$
  - $C=0.33T, H = T, 800 \text{ rpm}$
  - Glass beads, 600-710 mm, $C_{av} = 5.5\%$
Numerical Approach

- Eulerian-Eulerian two-phase flow
- Homogeneous two-phase flow model
- Frozen rotor – multiple frame of references
- High order accurate convection scheme
- Solid particle tracking

Streamlines
Normalized particle volume fractions

\[ \nabla \left( \frac{C}{C_{av}} \right) \Delta x \]
Axial Solids Profile

Comparison with data

- Three sets of curves are for nominal location and maximum and minimum values within 5 mm of location
- Measurement location inside the rotating domain, circumferential average

\[ r = 0.25T - 600\text{mm} \quad r = 0.25T - 710\text{mm} \]
Multiple Impeller Vessel

Particle Volume Fraction

Particle Drag Coefficient
Pulverizers and Classifiers

Challenges

► Control product fineness for low carbon loss, minimal NOx
► Balance competing forces of mill
  • Efficiency
  • Throughput
  • Power consumption

• Benefits of CFD
  — Reduce design time by
    • Understanding flow patterns to improve design
    • Predicting performance and erosion behavior for varying designs
    • Predicting dynamic behavior, stress, strain for rotating parts

Contours of erosion rate

Courtesy of Babcock Power, Inc
» Challenges
  ▶ NOx reduction
  ▶ Unburned carbon
  ▶ Fatigue and creep from thermal stresses in coal nozzle

• Benefits of CFD
  — Reduce design effort and field tests by
    • Predicting temperature, NOx, with varying fuel, load, swirl
    • Predicting thermal loads and stresses for different designs

Courtesy Of: Babcock Power and Southern Company Generation
Challenges

- Maintaining stable flame under varying load conditions
- Pollutant formation control
- Maintaining proper radiation and convection properties with retrofitted low NOx burners
- Optimizing retrofitted air staging
- Minimizing water wall corrosion
- Designing optimal spray system for Selective Non-Catalytic Reduction (SNCR)

Benefits of CFD

- Ensuring retrofit success by predicting
  - Flame shape and thermal loads
  - Impact of various air staging methods
  - Corrosion prone regions
  - SNCR local NOx reduction
- Avoiding further downtime in a trial-and-error approach
Fluidized Beds

Challenges
- Erosion
- Maximizing gas-solid contact
- Avoiding channeling
- Maximizing heat transfer to immersed tubes
- Creep and fatigue due to thermal stresses

Benefits of CFD
- Avoid costly problems after manufacture
  - Predict erosion in virtual prototypes
  - Predict channeling problems
  - Predict thermal stresses
- Optimize heat transfer
» Challenges
  ▶ Minimizing erosion and fly-ash build-up
  ▶ Optimizing heat transfer to incoming water

• Benefits of CFD
  — Determine areas of likely erosion and fly-ash build-up early in design phase
  — Make flow distribution modifications that will resolve problems before manufacture
Particulate Control

Challenges
- Short life of systems and components
- High cleaning frequency

Often caused by:
- Uneven flow distribution
- Uneven loading of baghouses, filters, and electrostatic precipitators

Benefits of CFD
- Determine expected loadings prior to field implementation
- Determine stresses on components
- Use results to optimize ducts and turning vanes
- Few shut-downs
- Shorter cleaning frequencies
- Longer bag and plate life

Flow streamlines in an electrostatic precipitator

Particulate flow through a series of baghouses
Sulfur Dioxide Scrubbers

» Challenges
- Poor distribution of spray and/or flue gas
- Designing spray nozzle placement

• Benefits of CFD
  — Improve retrofit and new scrubber performance by using virtual prototyping, predicting:
    • Air and spray droplet flow distribution throughout the scrubber
    • Local sulfur absorption and concentration
    • Droplet-wall interaction

Droplets follow the short-circuiting gas flow without proper residence time

Blue streamlines from upper part of the inlet duct short circuit the quench section below the bowl
Chemical Processing Industry

» Mixing tanks
  ▶ Blenders
  ▶ Polymerizers
  ▶ Hydrogenartors
  ▶ Crystallizers
  ▶ Fermentators

» Heat-exchangers
  ▶ Plate
  ▶ Tube and Shell
  ▶ Jacket Vessels

» Separation
  ▶ Cyclones
  ▶ Hydrocyclones
  ▶ Gravity Separators
  ▶ Electrostatic
  ▶ Magnetic
  ▶ Floatation

» Dryers
  ▶ Rotary
  ▶ Fluidized beds
  ▶ Ovens
  ▶ Spray dryers

» Filtration
  ▶ Centrifugal
  ▶ Granular bed
  ▶ Pressure
  ▶ Vacuum
  ▶ Ulterafine
  ▶ Chromatography

» Powder Handling
  ▶ Dust collection
  ▶ Mixers and blenders
  ▶ Conveyors and elevators
  ▶ Feeders
  ▶ Screening/separation
  ▶ Dryers
  ▶ Size reduction
Power Generation Industry

» Combustion systems
  ▶ Burners
  ▶ Boilers
  ▶ Furnaces
  ▶ Combustors

» Post combustion Systems
  ▶ Scrubbers
  ▶ Chillers
  ▶ Pipes, Ducts, Bends
  ▶ NOx Control
    • Spray Nozzles
    • Ammonia-injection
  ▶ Particulate Control
    • Precipitator
    • Bag houses
    • Classifiers

» Steam generation

» Flow Measurement and Control (Pumps, valves, flow meters, sensors)
» CFD and the Chemical Industry
» Industry Challenges
» How can CFD help
» The Guts of CFD
» What is CFD up to These Days
» Summary
Industry Challenges

- Large part count
- Involves complicated and advanced components and systems
- New product concept has never been tested before
- Develop a new product is time-consuming and expensive, requires a large team, and repeated testing and experiments
- Affordability always an issue
- Reduce development cost is top priority for government agencies or business corporations
- Reduce time to market, increase product competitiveness
- Unknown or challenging underlying physics
- Environmental impact (pollution, noise, etc.)
- Operational hazards (stability, icing, etc.)
- System/Structural integrity
» CFD and the Chemical Industry
» Industry Challenges
» How can CFD Help
» The Guts of CFD
» What is CFD up to These Days
» Summary
How can CFD Help?

- Innovative mixing tank design
  - Use jet momentum instead of the traditional stirring motion to mix
  - Tank consists of an oscillating septum
  - Holes on the septum to create jetting flow

- Design goals
  - Does it work?
  - Does it work better than the traditional tanks?
  - How much power is needed?
  - Multiple septa configuration?
Animations – dye mixing

- Septum speed = 15 cm/s
- Period of one stroke = 2 s
- Physical time step = period / 60
- Dye injected near the bottom of the tank
Animations – 2D Streamlines colored by dye
Animations – 3D streamlines colored by dye
Comparison with data
New Idea: Multiple Septa, does it work?

- Septum geometry remains the same
- Two septa setup
  - One diameter apart
  - Tank height increased by one septum diameter to accommodate additional septum
- Septum speed = 15 cm/s
- Period of one stroke = 2 s
- Dye injected near the bottom surface of the tank
Animations – dye mixing
Animations – 2D Streamlines colored by dye
Animations – dye mixing

dyeflow (Streamline 1)

- 5.000e+000
- 4.500e+000
- 4.000e+000
- 3.500e+000
- 3.000e+000
- 2.500e+000
- 2.000e+000
- 1.500e+000
- 1.000e+000
- 5.000e-001
- 0.000e+000

dyeflow (Contour 1)

- 5.000e+000
- 4.500e+000
- 4.000e+000
- 3.500e+000
- 3.000e+000
- 2.500e+000
- 2.000e+000
- 1.500e+000
- 1.000e+000
- 5.000e-001
- 0.000e+000
Force required to move the septum

Two Septa

Single Septum
Use of CFD in the Product Life Cycle

» Research

» Development
  ▶ Design
    • Requirements
    • General architecture / arrangement
    • How parts are installed and integrated
    • Design iterations
  ▶ Simulate & Troubleshoot Design
    • Discover potential problems
  ▶ Build, Test, Analyze Test
    • Goal is to Demonstrate/Verify/Validate/Certify
    • Final product

» Production/Manufacturing

» Operations
  ▶ Training
  ▶ Maintenance & Support
  ▶ Safety & Security
Challenges
► Understanding the impact of fuel changes
► Slagging, fouling, and corrosion
► Air staging and emissions control
► Grate combustion

Advantages of CFD
► The capability to design furnaces for a variety of fuels
  • Understanding scale-up implications
► Understand air staging implications on emissions, heat transfer, and ash
  • Access to biomass combustion models

Temperature (left) and carbon monoxide (right) contours near secondary air and flue gas inlets in a biomass grate combustion furnace

Courtesy University of Graz, Austria
Temperature contours for a rich-burn/quick-quench/lean-burn (RQL) staged trapped-vortex combustor showing the effect of air/fuel injection into the cavity. The air/fuel injection is changed to drive the main vortex (a) with the main flow, and (b) counter to the main flow.

CFD for Design of Green Buildings

» Award winning entry in U.S. Green Building Council (USGBC) Design Competition modeled using Airpak

» Entries required to meet requirements for LEED Platinum Rating plus target budget of <$100 per square foot.

» Natural ventilation design performance assessed through airflow simulation

Proposed Addition to facility for the Pittsburgh Project
Assessment of Natural Ventilation Design

» Simulation allows exploration of energy efficiency concepts that might otherwise be deemed unrealizable

» Velocity Vectors and Temperature Contours in Proposed Design
» Innovative Decontamination Designs

- Provide estimates of delivery system performance as part of developing specifications for the required system / decontamination operation
  - Can concentrations be raised to X ppm within Y hours
  - Identify areas which need to have enhanced agent penetration devices
Use of CFD to Specify Requirements

» 5 MW under-car-fire
  ► Located below the 2\textsuperscript{nd} car
  ► Fire begins shortly up-track of station
» Train enters station and stops
» No mechanical ventilation systems
» Piston effect due to train pushes air through station
» Buoyant forces due to fire and hot smoke also influence airflow
<table>
<thead>
<tr>
<th><strong>Applications</strong></th>
<th><strong>Benefits</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Ovens</td>
<td>Compute flow and heat transfer to optimize performance and reduce development costs</td>
</tr>
<tr>
<td>Refrigerators</td>
<td></td>
</tr>
<tr>
<td>Mixing equipment</td>
<td></td>
</tr>
<tr>
<td>Washers and dryers</td>
<td></td>
</tr>
</tbody>
</table>

- **CFD to Specify Details**

  » Predict flow and heat transfer within appliances and related components

  - **Applications**
    - Ovens
    - Refrigerators
    - Mixing equipment
    - Washers and dryers

  - **Benefits**
    - Compute flow and heat transfer to optimize performance and reduce development costs
CFD to Simulate & Troubleshoot

» FLUENT widely adopted for subsystem design and installation
  ► Difficult to develop accurate predictive models for flows in complex geometry, especially if complex physics present

Analysis of flow in internal cooling passages of a turbine blade & comparison with exp’t.

*Kawasaki presentation at Japan UGM, 2002*

- Secondary Flow Systems in gas turbine engines
  - CFD used to evaluate & troubleshoot installed performance of cooling systems, etc.

![Graph showing Mass flow rate Gc/Gc_c vs. Blade inlet Pressure P kPaG](image)

Efflux of coolant from turbine blade tip.

Smoke stack redesigned to eliminate offensive smell and eye irritation observed during sea trial.

*Courtesy of Daewoo Shipbuilding and Marine Engineering Co.*
CFD and the Chemical Industry
Industry Challenges
How can CFD help
The Guts of CFD
What is CFD up to These Days
Summary
How CFD Works

» Define domain of interest
  ▶ Flow over an airplane or a car, mixing tanks, blood flow in veins, steam turbine blade passages, etc.
How CFD Works

- Generate computational mesh
  - Subdivide the domain of interest into small cells
  - Each cell is a “control volume” where the flowfield information is stored
How CFD Works

- Select governing equations
  - Applied to control volumes (cells)
- Apply appropriate boundary conditions
- Select numerical parameters
- Start solving until a satisfactory flowfield solution is reached
Physical Models

» Inviscid, laminar or turbulent

» Conjugate Heat transfer
  ▶ Convection, Conduction
  ▶ Radiation

» Porous media and lumped parameter models
  ▶ Fan, heat exchangers, porous jump for zero thickness step changes

» Rotating Flows
  ▶ Multiple reference frames, mixing planes and sliding mesh

» Species transport and combustion suite

» Multiphase suite

» Heat exchanger module

» Icing module

» Acoustics

Contours of static pressure in a 4-stage turbine
Turbulence Modeling Hierarchy

- Direct Numerical Simulation
  - Resolution of largest → smallest eddies
  - 3-D, unsteady
  - Not practical for \( \text{Re} > 1 \times 10^4 \)
- Large Eddy Simulation
  - Resolution of large eddies
  - Simulation of small eddies with sub-grid scale model
  - 3-D, unsteady
  - Uncertainties for calculating wall fluxes
  - High computational demand → research tool
Advanced Turbulence Models

Large Eddy Simulation: advanced, highly promising

DES: a blend of LES and RANS approaches
Averaged Equations

- Averaged momentum equation:

\[
\frac{\partial (\rho \bar{U}_i)}{\partial t} + \frac{\partial (\rho \bar{U}_j \bar{U}_i)}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial (\tau_{ij} + \rho \bar{u}_i \bar{u}_j^\prime)}{\partial x_j} + \rho g_i
\]

- Averaged energy (enthalpy) equation:

\[
\frac{\partial (\rho \bar{H})}{\partial t} + \frac{\partial (\rho \bar{U}_j \bar{H})}{\partial x_j} = -\frac{\partial}{\partial x_j} \left( -q_j + \rho \bar{u}_j h' + \frac{1}{2} \rho \bar{u}_j \bar{u}_i u'_i \right) - \\
\quad -\frac{\partial}{\partial x_j} \left[ \bar{U}_i \left( \tau_{ij} + \rho \bar{u}_i \bar{u}_j^\prime \right) \right] + \frac{\partial \bar{P}}{\partial t} + \rho g_j \bar{U}_j
\]
Reynolds Averaged Navier Stokes (RANS)

- Second moment closures
  - Solve 6 transport equations for Reynolds stresses
  - Solve 3 transport equations for turbulent energy fluxes
  - Solve 1 transport equation for turbulence length scale
  - Expensive
  - Complex equations

- Eddy viscosity models
  - Relate Reynolds stresses to turbulence velocity scale and turbulence length scale
  - Relate turbulent energy fluxes to turbulence velocity scale and turbulence length scale
  - Solve 1 transport equations for turbulence velocity scale
  - Solve 1 transport equation for turbulence length scale
Core Turbulence Models
» Spalart-Allmaras’ one-equation model
» Standard k-e model
» Renormalization-Group (RNG) k-e model
» Realizable k-e model
» Wilcox’ k-w model
» SST k-w model
» Gibson & Launder’s RSM
» Speziale, Sarkar, and Gatzki’s RSM
» Subgrid-scale turbulence models for LES
» v2f model
» Detached Eddy Simulation

Customization
» Turbulent viscosity
» Source terms
» Turbulence transport equations
» Subgrid-scale models

Auxiliary Models
» Kato-Lauder’s modification
» Buoyancy effects
» Compressibility effects
» Low Re effects
» Pressure gradient effects

Near-wall Models
» Standard wall functions
» Non-equilibrium wall functions
» Two-layer zonal model
» Enhanced wall functions
• The famous k - ε model

\[
\rho \frac{Dk}{Dt} = \rho \nabla \cdot \left( v + \frac{v_T}{\sigma_k} \right) \nabla k + \rho P - \rho \varepsilon \\
\]

\[
\rho \frac{D\varepsilon}{Dt} = \rho \nabla \cdot \left( v + \frac{v_T}{\sigma_\varepsilon} \right) \nabla \varepsilon + \rho \beta \frac{\varepsilon}{k} (C_{\varepsilon 1} P - C_{\varepsilon 2} \varepsilon) + \rho C_{\varepsilon 3} (1 - \alpha) 2 \varepsilon
\]

\[
P = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \nabla \cdot \bar{V}
\]
Multiple Frames of Reference

7 million nodes, 2 frames, 2 phases, transient
Multi-Phase Flow

- System includes more than single phase fluid – gas-liquid, liquid-solid, or gas-liquid-solid
- Homogeneous or inhomogeneous equations for two-phase flow modeling
- Surface tension
- Interface resolution
- Inter-phase mass, momentum and energy transfer
Multiphase Flow Examples

Air Blast Atomizer
 Courtesy of UTRC

Liquid Sprays
DPM Model

Free Surface Flows: VOF Model

Liquid/Gas Bubble Column
Eulerian Multiphase
Particle Tracking

- Two-phase (particle-fluid), or multi-component (oil-water)
- Particles groups, sizes, distributions
- Tracking groups of particles through space and time
- Particle mass, momentum and energy exchange with mean flow
Particle Models

» Particle Methods
  ▶ DPM for dilute phase (steady and time dependent)
  ▶ Macroscopic Particle Model (MPM) for large particles (UDF)
  ▶ Particle in Cell method (MP-PIC) for dense flows with large size distributions (under development).

» Continuous Methods
  ▶ Euler-Granular,
  ▶ Euler-Granular with Frictional viscosity for dense phase

» Hybrid Methods
  ▶ Coupled simulations
  ▶ DEM (through partnership with EDEM Solutions)
Contours of temperature and vorticity for a partially premixed GE LM6000 combustor (LES simulation)

Concentration of OH radical for GE LM-1600 Gas Turbine Combustor.

Droplets colored by size for reacting spray flame. Droplet breakup and coalescence models used.

Pathlines of water mole fraction with surface contours of temperatures for a strutjet engine. Courtesy of Aerojet
Numerical Accuracy

1st Order Accurate

2nd Order Accurate
Moving Mesh

User prepares an initial mesh, CFD solver handles the mesh motion according to description or FSI interactions.
Moving Mesh Example: In-Cylinder Analysis

Particle Trajectories Colored by particle-velocity-mag (Time=0.0000e+00)  Apr 15, 2005
Crank Angle=299.00(ddeg)  FLUENT 6.3 (3d, segregated, dynamesh, spe, ske, unsteady)
Flight of the ... Fly?

Fly Flight Simulation
Dynamic Mesh

Dr. Ralf Kröger
Fluent Germany
rkr@fluent.de

Fly Flight Simulation
Pressure View 1 filled

Dr. Ralf Kröger
Fluent Germany
rkr@fluent.de
Solver Validation: Scavenging Analysis

Port configurations and the interface between the burnt gas and fresh charge

Burnt gas mass fraction contours

Honda RS125

Design Variation 1
» CFD and the Chemical Industry
» Industry Challenges
» How can CFD help
» The Guts of CFD
» What is CFD up to These Days
» Summary
Transient Analysis

- More simulations are transient (unsteady)
  - Realistic flow physics
  - inherently transient physical models -- LES/DES/SAS, vortex shedding, shear layer vortex rollups, rotor-stator interactions, fluid-structure interactions, transient particle tracking, etc.
  - Time-derivative term is present in addition to spatial derivatives
Aerospace Industry

AIAA drag workshop

Separated Flow

Separated Flow

ENGGINEERING ADVANTAGE
The Aerospace Industry
The Auto Industry

- Complete Underhood & Underbody analysis
- Sub/System and Component Level analysis
  - Engine Compartment, Fans, Heat Exchangers, Exhaust System, etc…
Conjugate Heat Transfer – Electronic Cooling

Electronics/Avionic Cooling

Temperature contours in motor controller
Courtesy Hamilton Sundstrand

Electronics cooling cabinet simulation
Advanced Electronics Cooling

Swirling Flow on Active Heat Sink

Active heat sink modeling

Microchannel Closed Loop Cooling

Vapor Liquid source

Droplet Evaporation - Phase change cooling

Vapor condensation

heat source

vapor collection

pump

nozzle

nozzle
Allianz Stadium in Munich

Specially built for the World Cup
Holds 70,000 spectators and the largest car park in Europe (10,000 cars) underneath it

» Simulation was used by VTD GmbH to design the stadium with respect to
  • Airflow requirements over the stadium
  • Thermal comfort for the crowd
  • Airflow over the turf for optimal grass growing conditions
  • Fire/smoke management
  • Parking garage ventilation
Turbomachinery – 15 - Stage Axial Compressor

• Prototype first compressor version, before optimization, of the Siemens V84.3A family
• Inlet guide vane, 15 stages
• Tip gaps, mass bleeds, hub leakage flows
• Steady and transient simulation

Courtesy Siemens
- R5-S9, 750 blade passages, 31.2 million nodes, transient
- Instantaneous total pressure after 550 time-steps

Courtesy Siemens
Efficiency, Steady and Transient

Relative difference in efficiency at design point

<table>
<thead>
<tr>
<th>Mesh Size</th>
<th>Simulation Type</th>
<th>Rel. difference to design efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>Steady</td>
<td>-2.0%</td>
</tr>
<tr>
<td></td>
<td>Transient</td>
<td>-1.5%</td>
</tr>
<tr>
<td>Fine</td>
<td>Steady</td>
<td>+0.1%</td>
</tr>
<tr>
<td></td>
<td>Transient</td>
<td>+0.3%</td>
</tr>
</tbody>
</table>

Predicted mass flow compared at design point

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Simulation type</th>
<th>% difference in mass flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>Steady</td>
<td>-1.44</td>
</tr>
<tr>
<td></td>
<td>Transient</td>
<td>+ 0.28</td>
</tr>
<tr>
<td>Fine</td>
<td>Steady</td>
<td>+ 1.04</td>
</tr>
<tr>
<td></td>
<td>Transient</td>
<td>-0.102</td>
</tr>
</tbody>
</table>

Courtesy Siemens
Lobed Pump and Rotary Compressor

Lobed Pump

Pressure Contours in a Rotary Compressor
Wind Power

» Challenges
- Seismic load calculations and assurance
- Fluid-structure interaction of lightweight composite structures
- Maximizing efficiency of turbines and turbine placement

» Benefits of CFD
- Coupled physics for true virtual prototyping
  - Turbulence, Wake, Vortex Shedding
- Optimize turbine output and placement
  - Wind speed prediction over complex terrain

Wind velocity contours showing the wake effect of one turbine on another
Fuel Cells

Challenges

- Channel design that optimizes distribution of oxygen (hydrogen) to the cathode (anode)
- Water management
- Thermal stresses and cooling plate design
- Materials
  - High costs
  - Property variation
- Space limitations

Advantages of CFD

- Probabilistic design in virtual prototypes
- Minimize costly physical prototypes using analysis based on first principle physics (electrochemistry, fluid flow, heat and mass transfer, structural mechanics)

Shape optimization of a stack head, showing where material can be removed (red)

Courtesy of INIA, Spanish Institute for Aerospace Technology
Refrigeration Equipment

» Refrigerated Display Case
A prototype oven design showed poor performance in terms of air flow and temperature uniformity.

Adaptations including cutting off part of the surface, available to air flow through the oven and optimising the size of air deflection plates proved successful.

- Undesired air recirculation was avoided.
- Optimization of the air deflection plate width improved the performance in terms of air flow and temperature uniformity.

Courtesy of Siegfried Denys & Jan G. Pieters Biosystems Engineering Ghent University
Biomedical Industry

- Wall pressure in the human upper airways
- Wall pressure for an abdominal aortic aneurysm
- Particle path in inhaler
- Drug concentration for stent modeling
FSI Example – Pressure Limiting Valve

Ø 4.5 mm
Ø 4.0 mm
55°
Ø 2.4 mm
Ø 10.0 mm
0.25 mm
FSI Examples – Vena Cava Filter

Deformation

Streamlines
» CFD and the Chemical Industry
» Industry Challenges
» How can CFD help
» The Guts of CFD
» What is CFD up to These Days
» Summary
» When working with fluids
  ► Highly non-linear systems
  ► Textbook problems are simplified N-S equations, such as flow over a flat plate, pipe flows, etc. – non-linearity is reduced to a minimum
  ► When heat transfer is involved, the textbook examples are mostly experimental data for simplified situations
  ► When non-linearity (convective) terms dominate, textbook examples no longer apply.
  ► Guesstimate flow solutions based on past experience often leads to over-engineering
  ► When pushing the technology envelop where past experience no longer exists, attempts to extrapolate past data either fail or misleading
  ► Seek advise from CFD or test data to overcome technology bottleneck
  ► CFD has emerged to be an integral part of many companies’ product design cycle -- only at a small fraction of the cost of conducting many expensive experiments