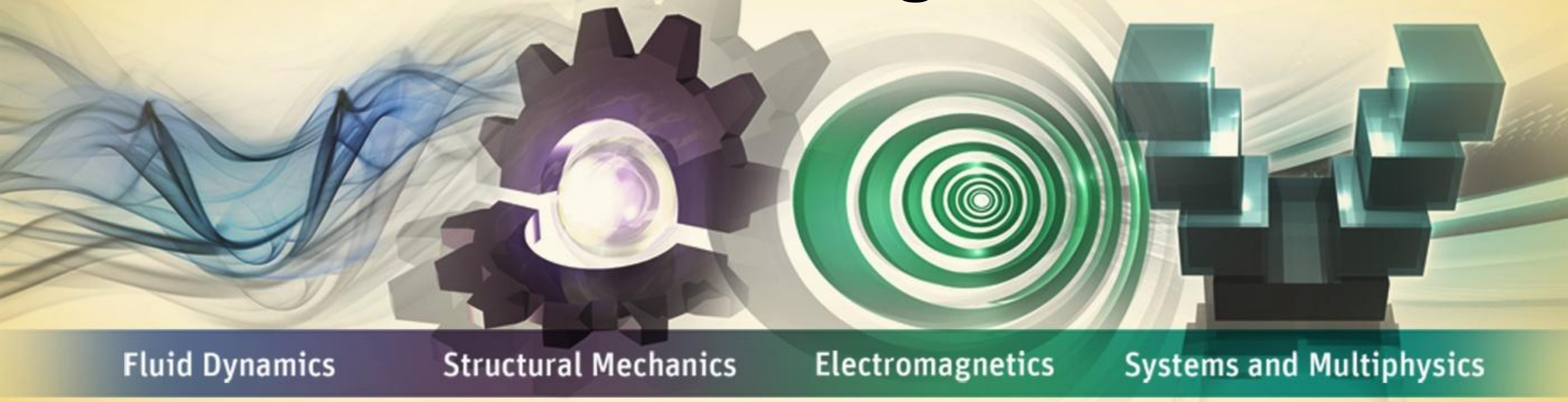
ANSYS

Realize Your Product Promise™

Computational Fluid Dynamics and its Applications in Offshore and Subsea Processing



Madhusuden Agrawal

ANSYS Houston

madhusuden.agrawal@ansys.com

AGENDA

What is CFD?

- Brief Intro to CFD
- When and Why to apply CFD

Why ANSYS?

- Portfolio for broad range of CAE needs
- ANSYS focus for oil & gas industries

Applications of CFD for Offshore and Subsea

- Wide range of Examples including gravity separator, gas dispersion, flow assurance, …

What is CFD?

CFD = Computational Fluid Dynamics

CFD is an engineering tool for virtual experiments on computer before cutting the metal

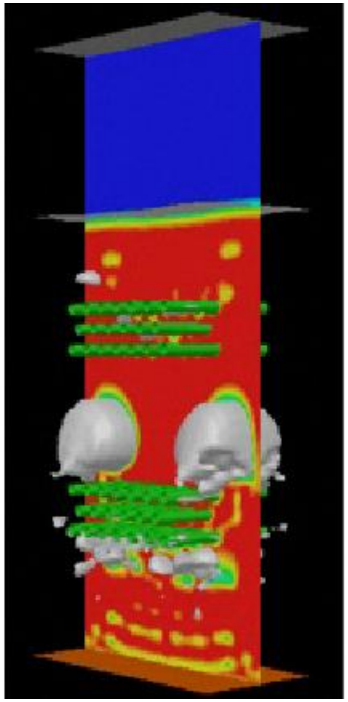
- **CFD is the simulation of fluid flow and heat and mass transfer by solving conservation equations on a computer**
- **CFD results enable visualization inside the "black box"**

Historical CFD:

- **A few exact solutions**
- **1D or 2D approximations**
- **Experiments**

Advances in computer technology

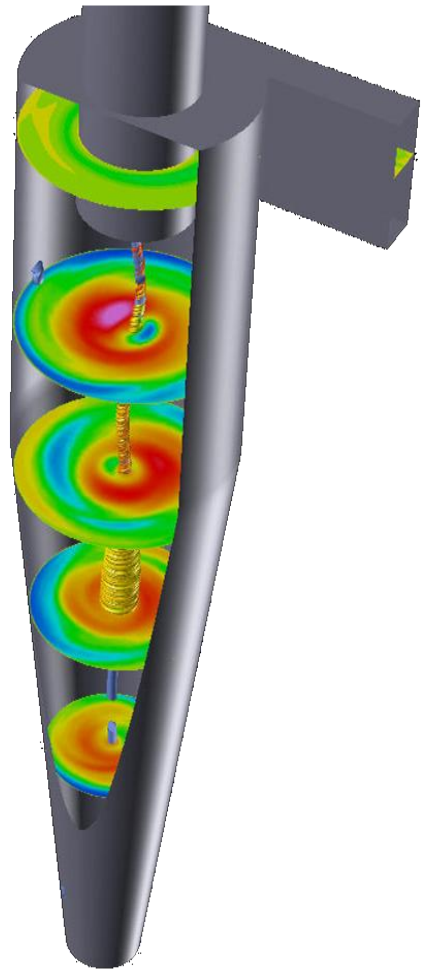
- **CFD is an R&D analysis tool**
- **CFD is a design tool**



Volume fraction of gas bubbles in a fluidized bed with internals (solid volume fraction is in red)

Why to use CFD?

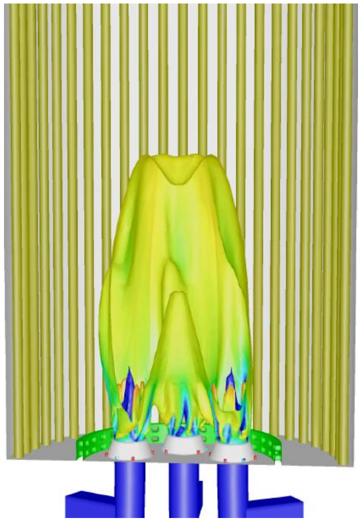
- **CFD is a powerful engineering tool for predicting real process behavior**
  - CFD provides a detailed understanding of flow distribution, pressure losses, heat transfer, particulate separation, collection efficiency, etc.
- **It is typically applied to**
  - Design evaluation, verification and optimization,
  - Scale-up analysis,
  - Performance evaluation, and
  - Problem solving, what-if scenarios
  - Study off-design operating conditions
- **CFD analysis used in complement with testing and field data, when available**



CFD results illustrating the vortex core and flow velocity at various axial planes

When to use CFD?

- **Design correlations or bulk models are not available or give poor results**
- **Scale-up laws are not available**
- **Detailed information on equipment behaviour is needed**
  - Identify root causes of problem, not just the effect
- **Operations involve complex physics (reactions, multiphase flows, non-Newtonian rheology, etc.)**
- **Comparing design alternatives or "what if" scenarios**



Courtesy of John Zink Co.

Lower costs and shorter time to market!

Mathematics of CFD
Conservation Equations

Conservation of Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}\left(\rho u_k\right) = 0$$

Conservation of Momentum: Navier-Stokes Equations

$$\rho \frac{\partial u_j}{\partial t} + \rho u_k \frac{\partial u_j}{\partial x_k} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial u_k}{\partial x_k}\right) + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho B_j$$

Conservation of Energy

$$\frac{\partial\left(\rho E\right)}{\partial t} + \frac{\partial}{\partial x_k}\left(u_k\left(\rho E + p\right)\right) = \frac{\partial}{\partial x_k}\left(k \frac{\partial T}{\partial x_k} - \sum_{j'} h_{j'} J_{j'} + u_j \tau_{ij}\right) + S_h$$

Equation of State

$$\rho = \rho(P, T)$$

Property Relations

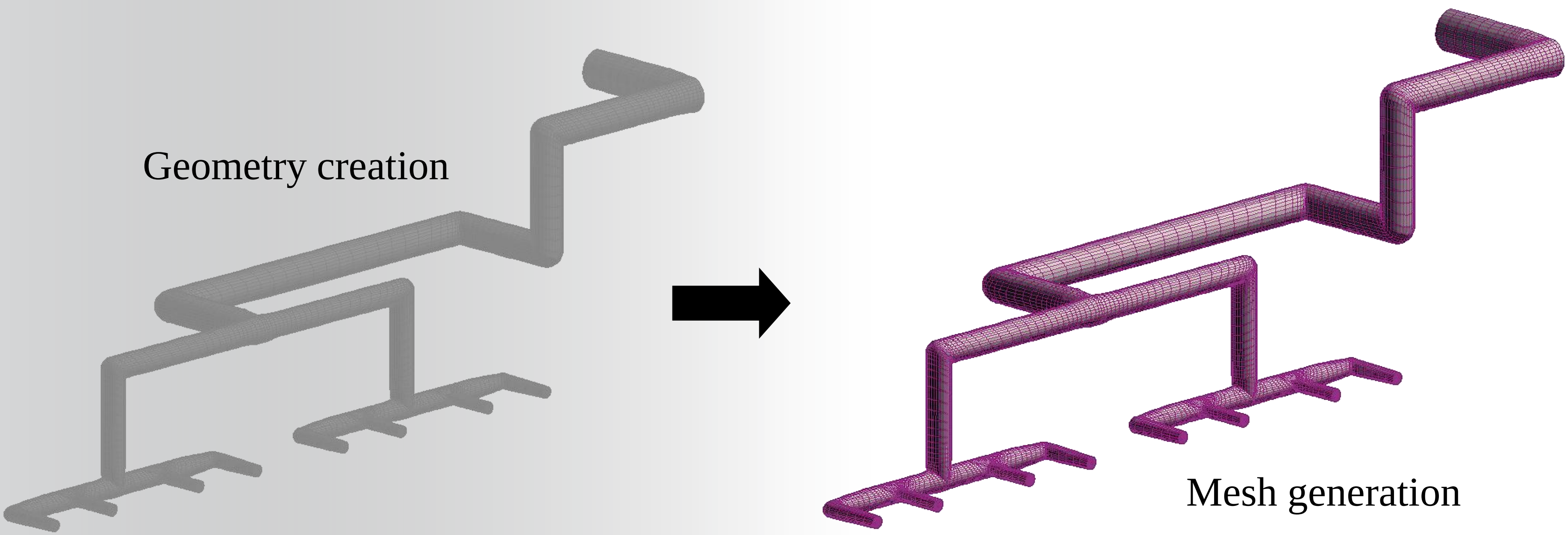
$$\mu = \mu(T, \dot{\gamma})$$

$$k = k\left(T\right)$$

$$C_p = C_p\left(T\right)$$

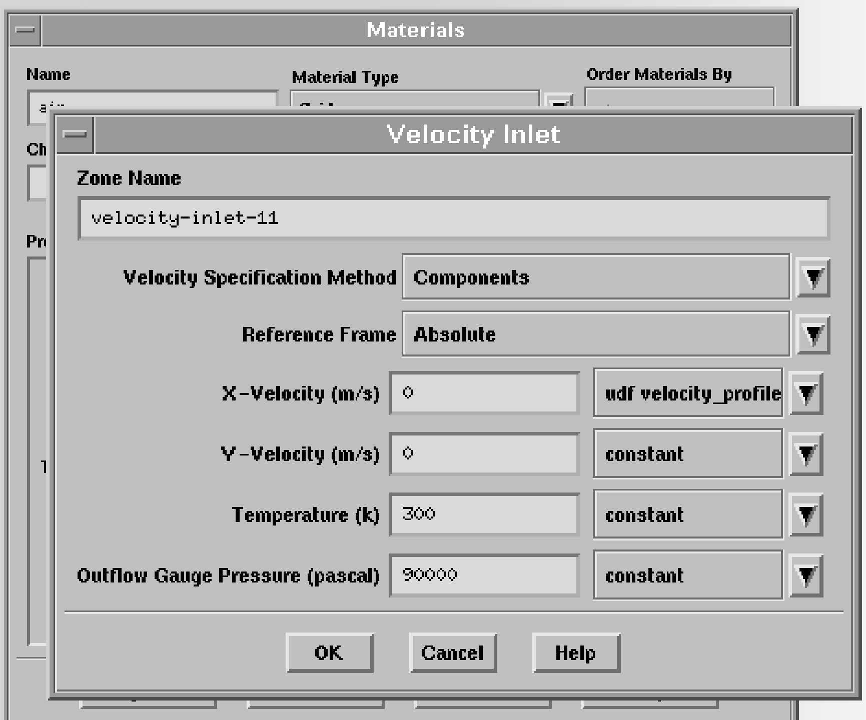
Steps of CFD: Pre-processing

- **Construct/Import/Clean geometry**
- **Form a mesh in which the equations governing the flow physics are solved. This divides the domain into small control volumes**

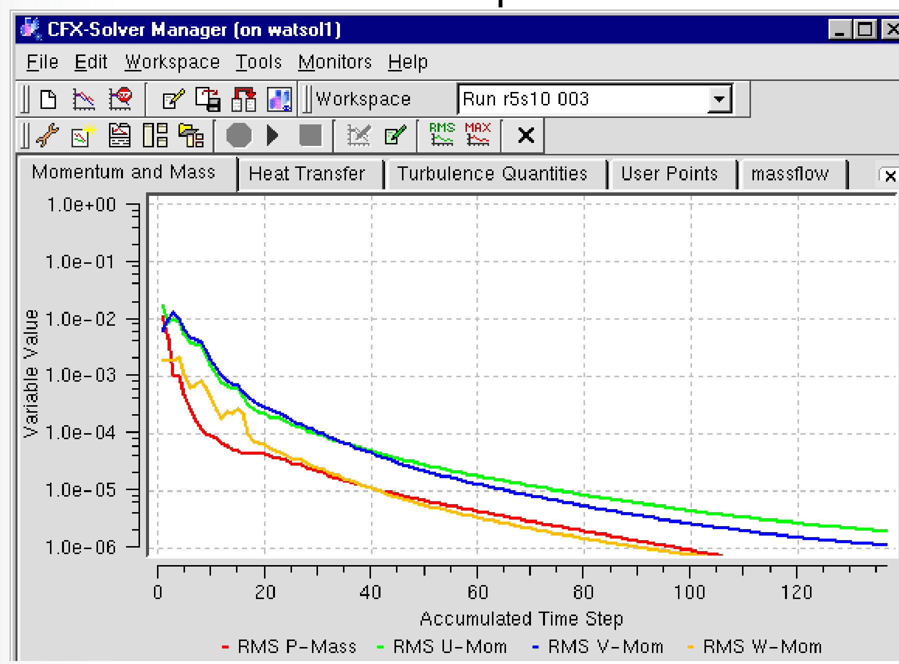


Steps of CFD: Problem Definition and Solving

- Define fluid properties, flow conditions, additional physics, solver settings
- Submit to solver for final solution

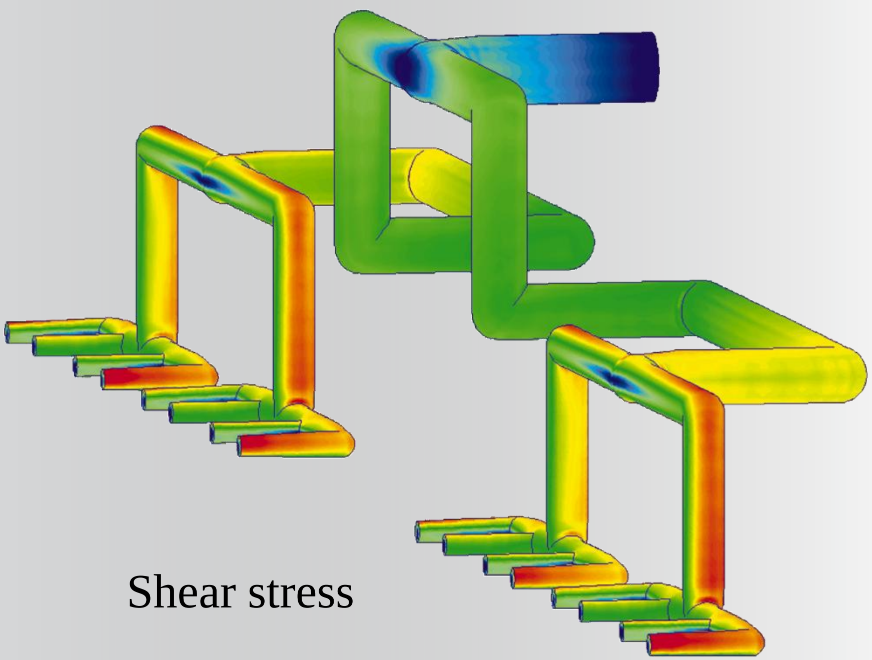


Residual plot

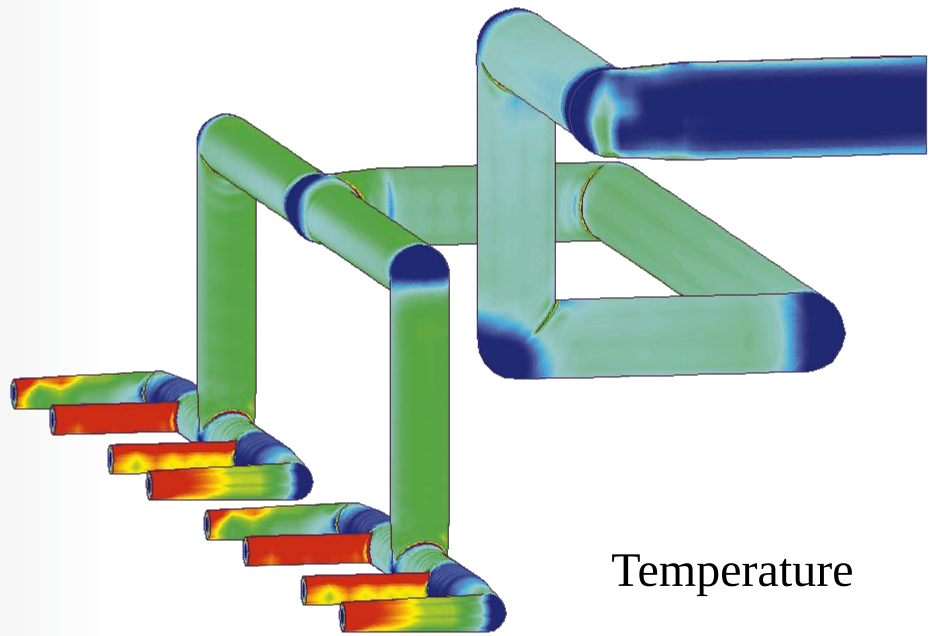


Steps of CFD: Post - Processing

- Viewing the results is called “Post-Processing”
- Generate results graphically & numerically:
    - Numbers, graphs, figures, animations



Shear stress



Temperature

Resources Needed for CFD

Hardware

- PC running Windows XP or Linux
- Fastest CPU affordable, 2GB RAM or more

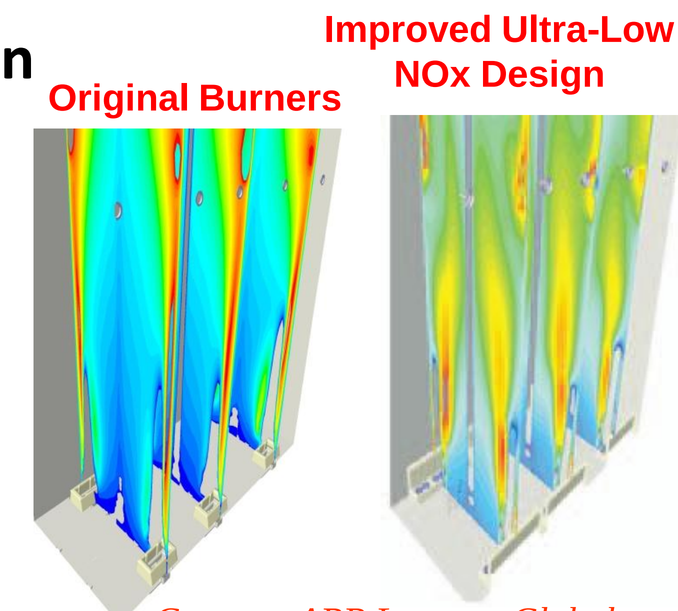
May want to consider parallel computation

- Problem size gets into multi-million cells
- Problems involve complex physics and chemistry

Users

- Routine CFD analysis by Process/design engineers
  - BS in Engineering or related fields
  - Understanding of processes
  - Fluids and CAD background a plus; not required
- Two or more users preferred
- 1-2 months learning curve
- > 30% time

} For long-term expertise cultivation

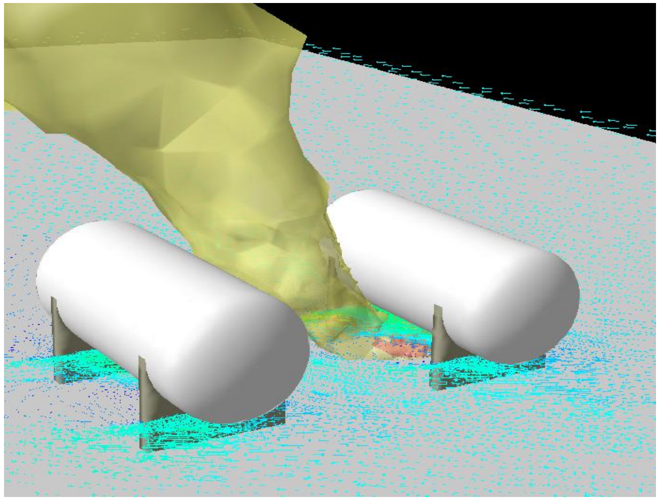


Courtesy ABB Lummus Global

- **ANSYS offers start-up projects to ramp up initial learning**

Testing and Simulation

- Simulation complements but does not replace testing
- Simulation depends upon material and rheological testing for property data
- Simulation depends on field measurements for boundary/initial conditions
- Simulation can guide and reduce testing



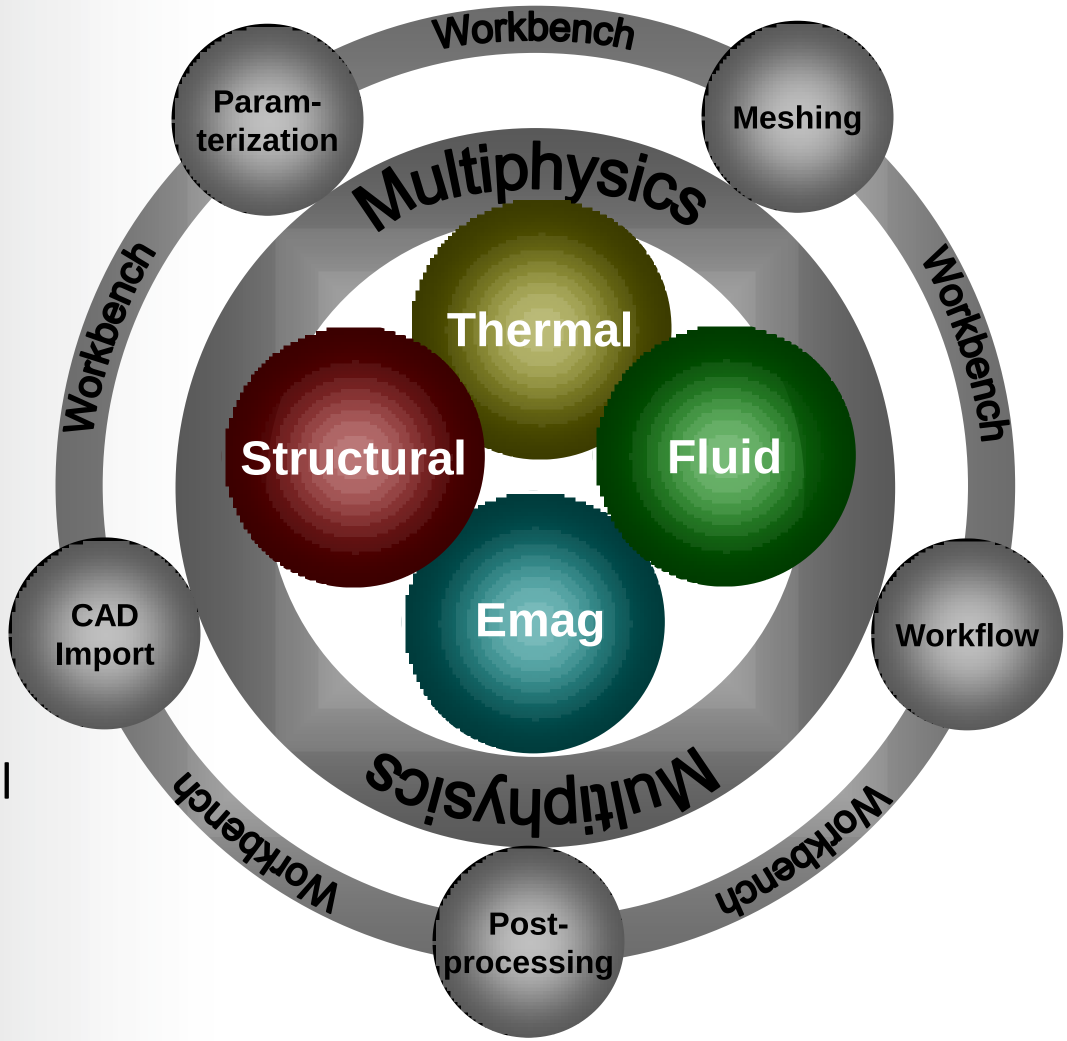
Plumes from evaporating pool of spilled liquid

Everybody believes in experiments except the experimentalists!

Nobody believes in simulations except the analysts!

ANSYS, The Company

- **ANSYS design, develops, markets and globally supports a range of CAE simulation softwares**
- **A suite of multi-purpose software technologies for**
  - Fluid Dynamics
  - Structural Mechanics
    - Implicit
    - Explicit Dynamics
  - Electromagnetics
  - Multiphysics
- **Selection of niche tools**
  - ANSYS ASAS/ANSYS AQWA – offshore specific tools for global structural/hydrodynamics
  - ANSYS ICEPAK (electronics)



ANSYS Simulation Includes…

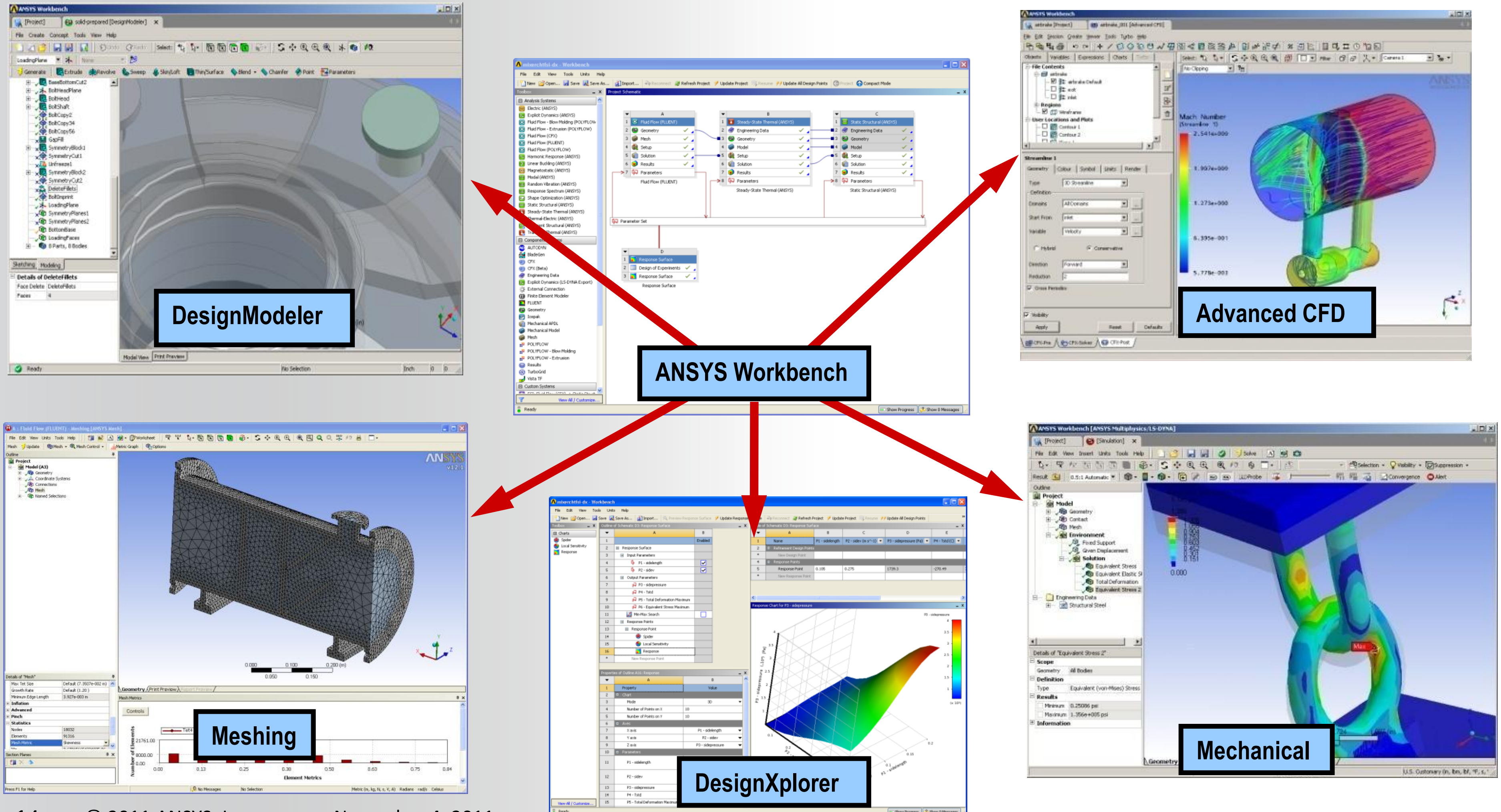
ANSYS Multiphysics Solutions

- **Electromagnetic**
  - **Low Frequency and EM**
    - Maxwell
    - Q3D
    - Simplorer
  - **High Frequency**
    - HFSS
    - Siwave
    - Designer
    - Nexxim
- **Mechanical**
  - **Implicit**
    - ANSYS Mechanical
    - ANSYS Structural
    - ANSYS Professional
    - Post Tools
      - ANSYS Ncode Design Life
      - Design Assessment
  - **Explicit**
    - ANSYS AUTODYN
    - ANSYS LS-Dyna
- **CFD**
  - **Electronics Cooling**
    - ANSYS Icepak
  - **General Purpose**
    - ANSYS CFD Fluent/CFX

Optimization: Design Explorer and Optimetrics

ANSYS Workbench

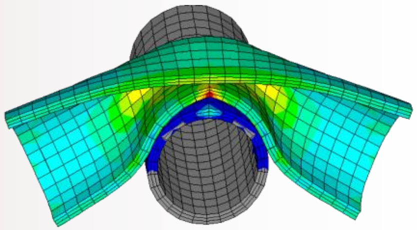
A common environment integrating ANSYS tools for multi-disciplinary CAE simulation

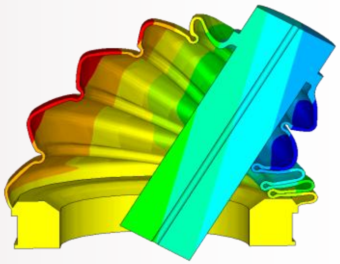


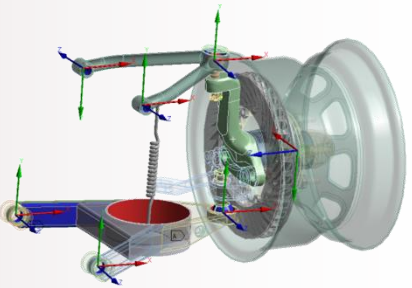
ANSYS Solution: Structural

Structural | Fluids | Thermal | Emag | Meshing

- Advanced Capabilities
  - Nonlinear materials
  - Large deformation
  - 3D frictional contact
  - Structural dynamics
  - Explicit dynamics
  - Large overall motion
- High Performance Computing
  - Parallel processing
  - Highly scalable
- Customizable
  - User elements & materials
  - Scripting





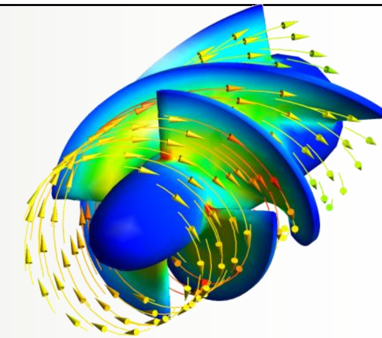


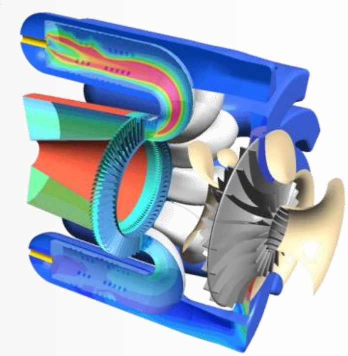
Advanced Technologies

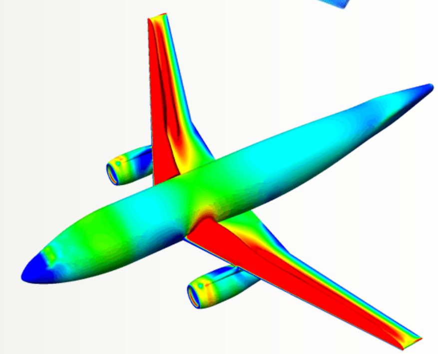
ANSYS Solution: Fluids

Structural
Fluids
Thermal
Emag
Meshing

- Advanced Capabilities
  - Multiphase
  - Multispecies flows
  - Reacting flows
  - Multiple reference frames
  - Latest turbulence models
- High Performance Computing
  - Large models: >1 billion
  - Large clusters: >1024 cores
- Integrated
  - Unified meshing
  - CFX & FLUENT solvers
  - Unified post-processing







Advanced Technologies

ANSYS Solution: Electromagnetics

Structural

Fluids

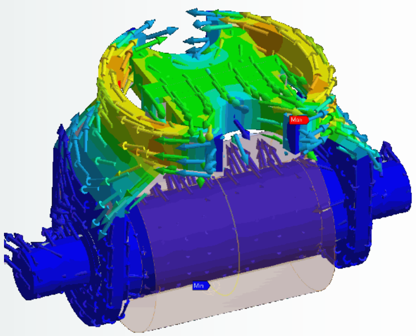
Thermal

Emag

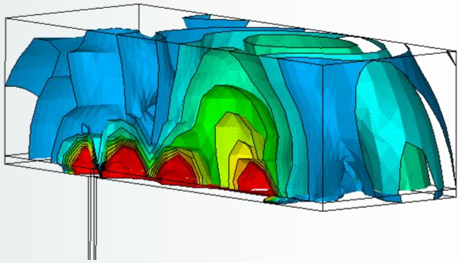
Meshing

Advanced Technologies

- Developed Over 25 Years
- Advanced Capabilities
  - Magnetostatics
  - Electrostatics
  - Low & high frequency electromagnetics
  - Circuit analysis & coupling
  - Joule heating



- Advanced Materials
  - Linear & nonlinear
  - Isotropic & anisotropic

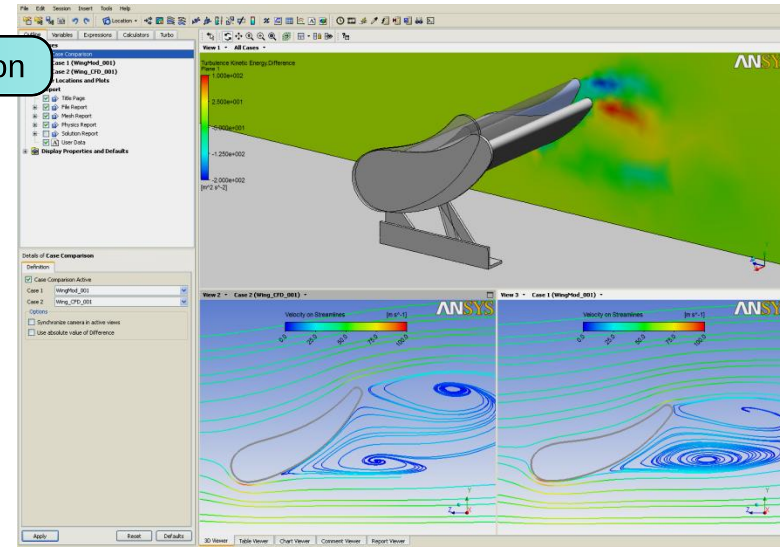


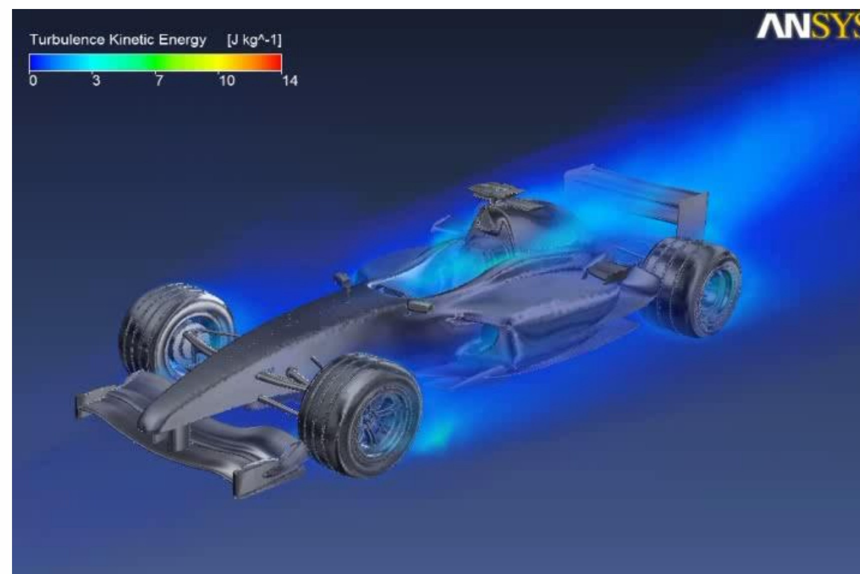
- Automated Post-Processing
  - Field intensity, scattering

ANSYS CFD-Post

- **ANSYS CFD Post Processor**
  - Full featured, modern, rich functionality
  - Common to all ANSYS CFD solvers
- **Advanced case comparisons**
  - Compare analyses
  - Review difference plots
  - Quantitative analysis, reporting
- **Transient analysis support**
  - Efficient data management
  - Support for re-meshing
- **Additional capabilities**
  - Create histograms & FFT's,
  - Flow feature detection
  - Trace particle enhancements

Case Comparison





Gas-Liquid System: Multiphase Models in CFD

Discrete Phase Model

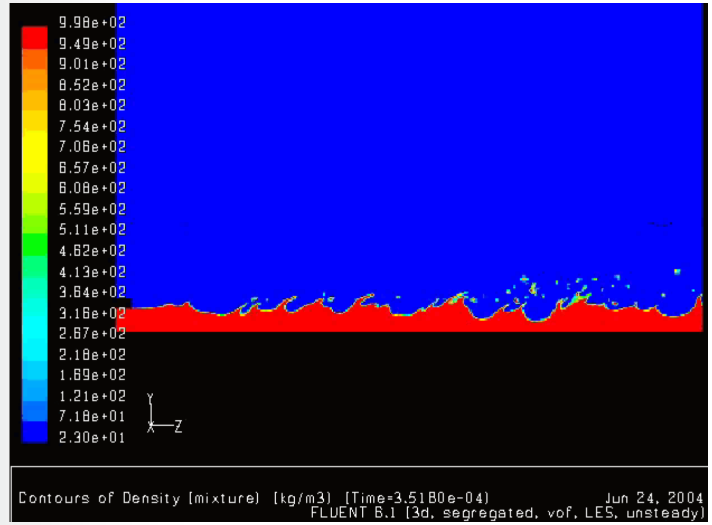
- Liquid droplets/gas bubbles
- Maximum of 12% volume fraction
- Can be used as a post-processing
- Computationally cheap

Volume of Fluid (VOF) Model

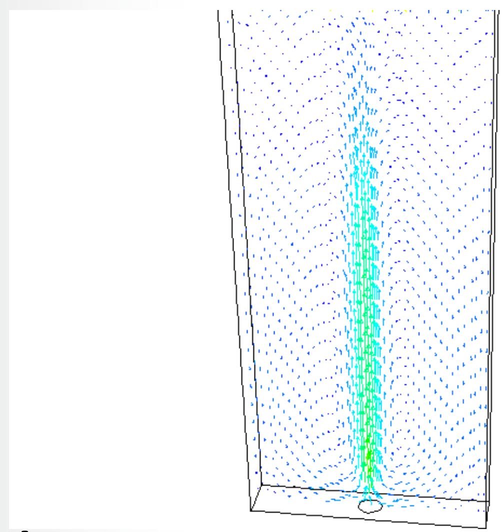
- Tracks fluid interface
- No maximum volume fraction
- Not practical for droplet/bubble modelling at large scale

Eulerian Model

- Liquid droplets/gas bubbles
- No maximum volume fraction
- Expensive if many diameters wanted



Atomization of Liquid



Churn turbulent bubble column

  November 4, 2011

ANSYS CFD Models for Particulate Flows

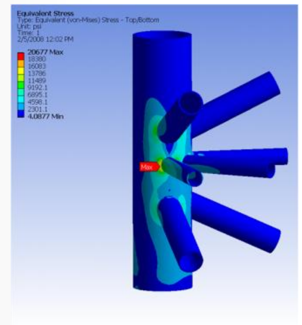
Model	Numerical approach	Particle fluid interaction	Particle-Particle interaction	Particle size distribution
DPM	Eulerian – Lagrangian	Empirical models for sub-grid particles	Particles are treated as point masses	Easy to include PSD as in Lagrangian description
DDPM	Eulerian – Lagrangian	Empirical; sub-grid particles	Approximate by KTG as in granular models	Easy to include PSD as in Lagrangian description
DDPM - DEM	Eulerian – Lagrangian	Empirical; sub-grid particles	Accurate calculations based on soft sphere collisions	Can account for all PSD physics accurately including geometric effects
Euler Granular model	Eulerian – Eulerian	Empirical; sub-grid particles	Approximate by KTG as in granular models	Different phases to account for a PSD; PB models for size change
Macroscopic Particle Model	Eulerian – Lagrangian	Accurate calculations based on local flow, pressure and shear stress distributions	Accurate calculations based on hard sphere collisions	Easy to include PSD as in Lagrangian description

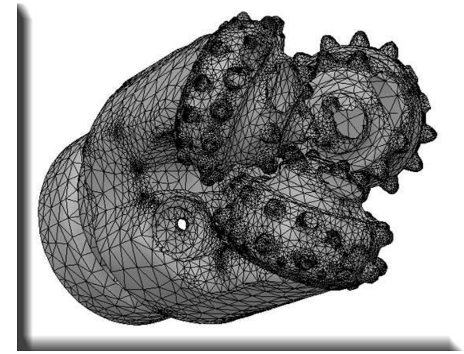
Turbulence Models in ANSYS CFD

- **RANS Models**
  - **One Equation Model**
    - — Spalart-Allmaras
  - **Two Equation Models**
    - — k-ε Family
      - Standard, Realizable, RNG
    - — k-ω Family
      - Wilcox, BSL, SST
  - **Laminar-Turbulent Transition Models**
    - — k-kl-omega (3 eqns)
    - — Transition SST (4 eqns)
  - **Reynolds Stress Model**

- **Scale Resolving Simulation (SRS) Models**
  - — URANS
  - — Large Eddy Simulation (LES)
  - — Hybrid RANS/LES
    - Detached Eddy Simulation (DES)
    - Scale Adaptive Simulation (SAS)
    - Embedded LES (ELES/ZFLES)
    - Wall Modeled LES (WMLES)

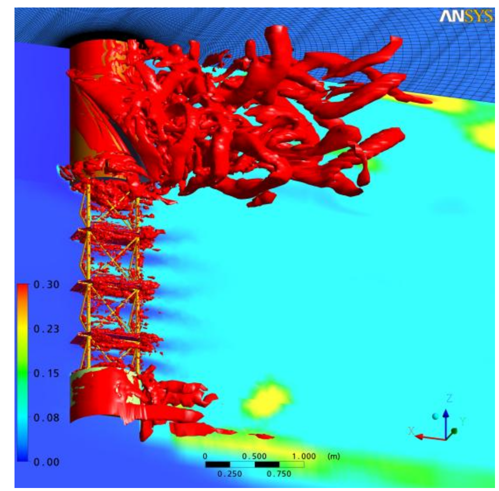
Solutions for Oil and Gas

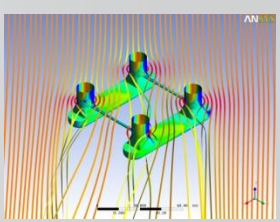


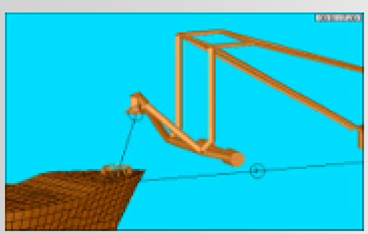


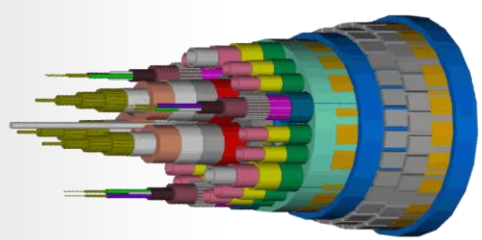
From well to wheel

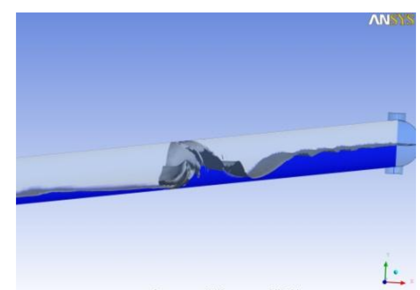
- Drilling and completion
- Enhance oil recovery
- Flow assurance
- Offshore and subsea structures
- Impact, blast, fire and safety
- Piping, transport, storage
- Refining and processing
- Fuel formulation/engine design
- Electronics, sensors and electric machines











ANSYS Solutions used Throughout the Supply Chain

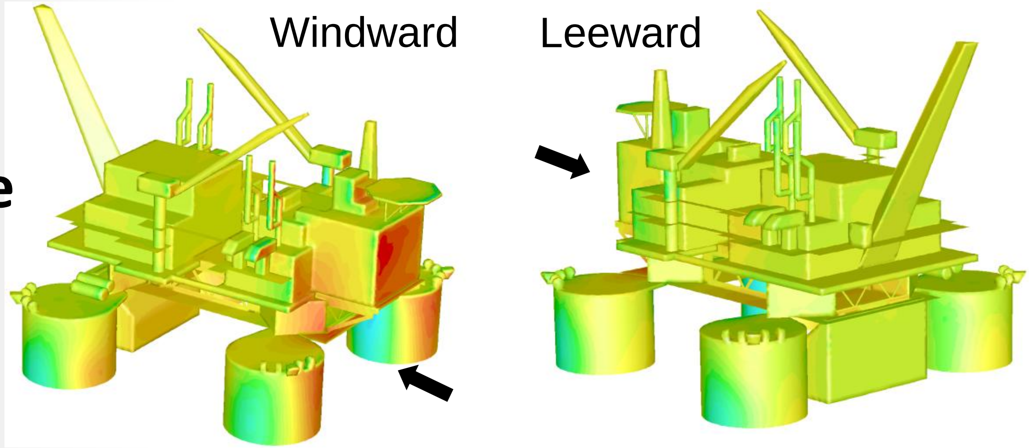
IOC/NOC E&P | Drilling Equip. Services | Oil & Gas Services | Refining | Oil & Gas Processing | Equipment & Machinery | E&C

Shell
Diamond Offshore
Halliburton
Weatherford
Cameron
Oceaneering
bp
Technip
Reliance Petroleum Limited
ExxonMobil
FMC Technologies
ConocoPhillips
Chevron
Eni
GE Oil & Gas
BR Petrobras
Schlumberger
SBM Offshore
Baker Hughes
Repsol YPF
Total
Valero
Statoil
IndianOil
Sinopec
National Oilwell Varco
PetroChina
Syncrude
Oil States International, Inc.
Saudi Aramco
Ecopetrol
KBR
CNOOC
ABS

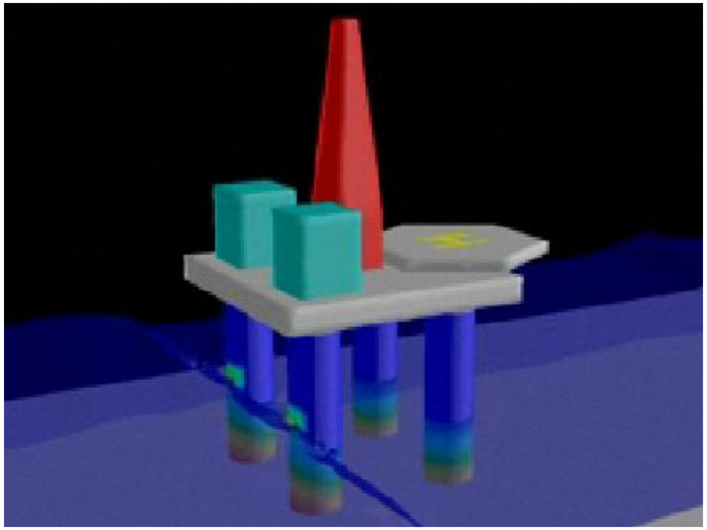
Offshore Structures: Wind and Wave loading

CAE Solutions

- Detail mapping of wind loads on all elements of the structure
- Effect of waves
- Fluid induced motion (VIM) studies
- Account for extreme loads due to storms, effect of wind headings
- Forces and flow details around the helicopter deck
- Visual illustrations of recirculation and low flow areas for smoke and pollutants dispersion



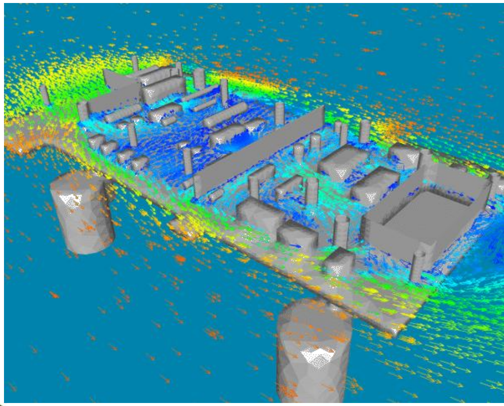
Pressure distribution, Production Troll
Courtesy of Hydro

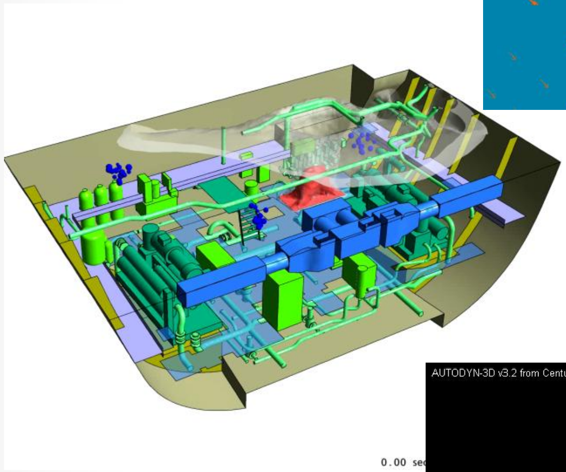


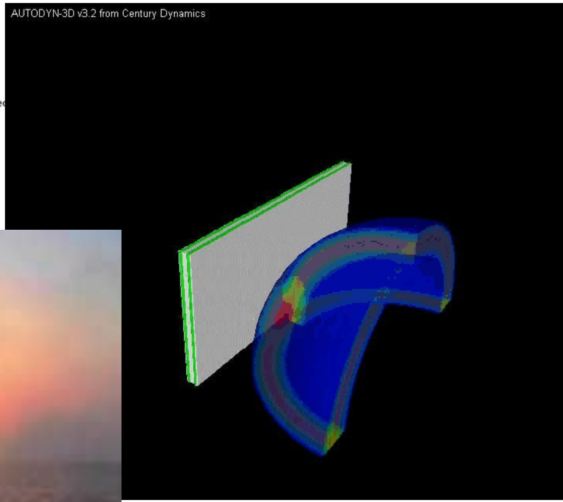
Transient pressure distribution caused by waves on an example shallow water platform

Fire and Safety Related Applications

- **Gas Dispersion and Ventilation**
  - **Formation of combustible gas clouds**
  - **Smoke and plume trajectories**
  - **Gas concentration levels**
  - **Ventilation times**
  - **Helicopter operations**
- **Explosion and Fire Propagation**
  - **Fire Suppression**
  - **Blast wave interaction with structure**



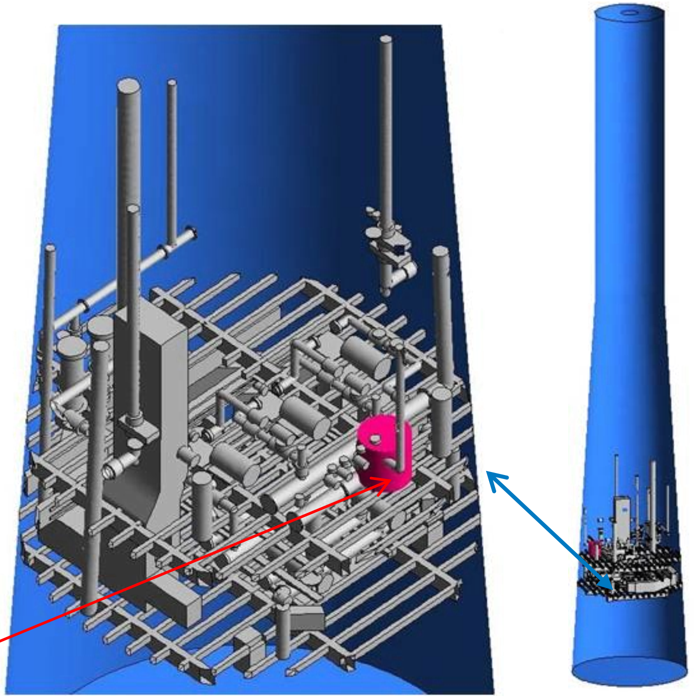




Case Study: Offshore Leg

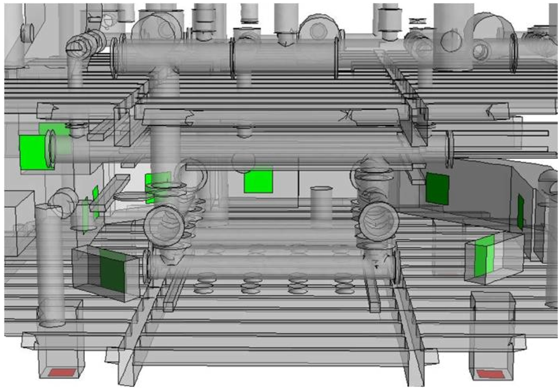
Release of heavy hydrocarbon gas and liquid inside the leg of an offshore platform

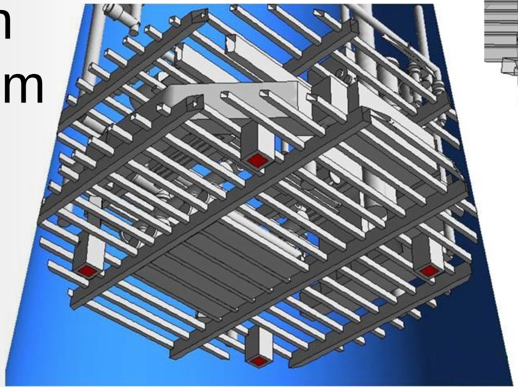
Asses the role of extract ventilation in mitigating against flammable gas concentrations



Potential hazard

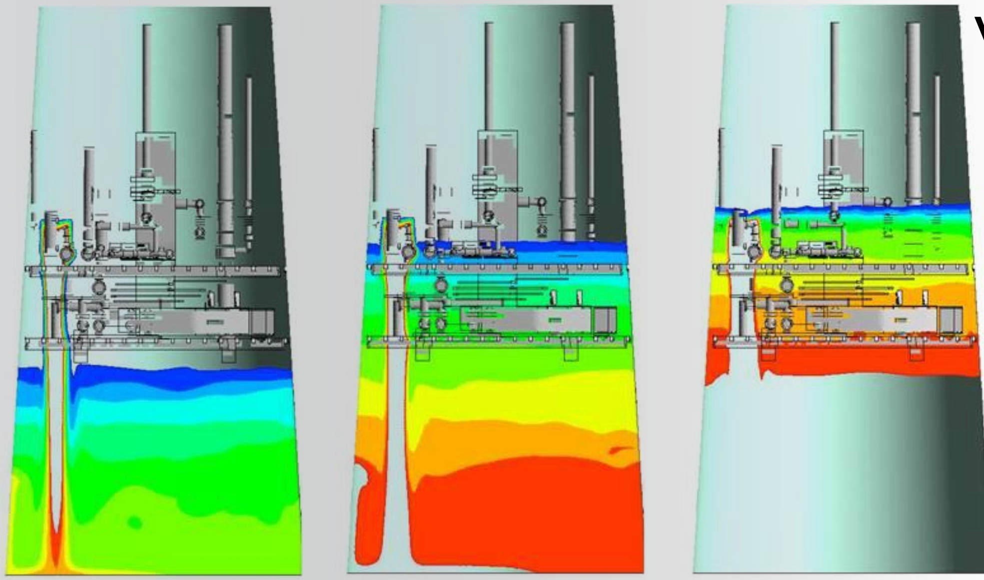
- Heavy gas fills the enclosure from the bottom up
- Flammable mixture forms a band which rises up over time. Potential ignition sources within the structure and from people working with machinery
- Gas is also toxic



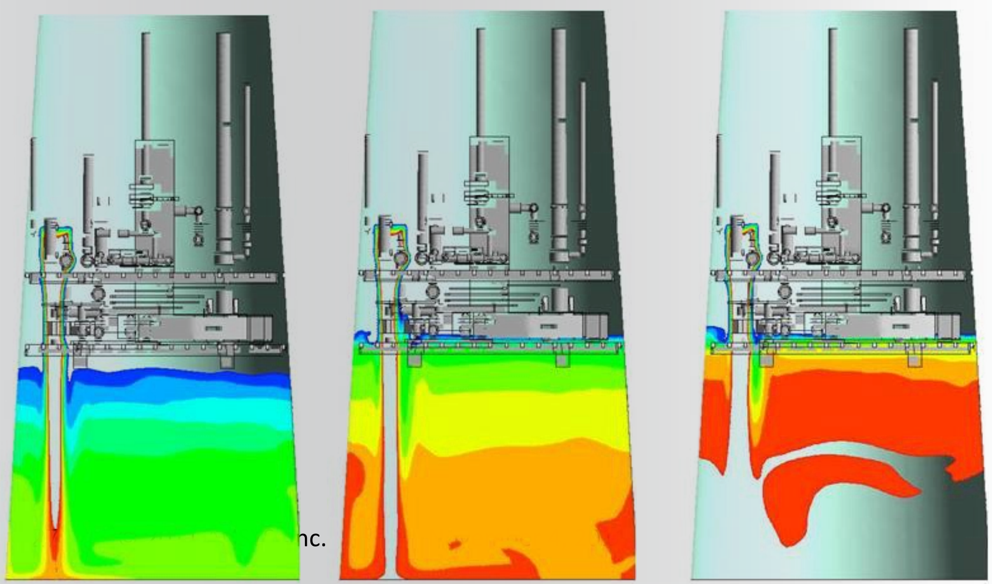


Extract ventilation configuration

Offshore Leg - Results

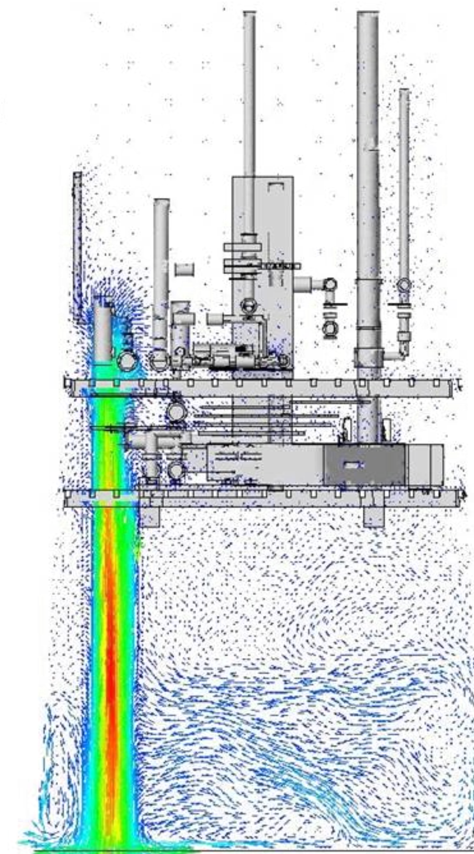


5 minutes 10 minutes 15 minutes



Ventilation turned off

After 10 minutes the flammable volume covers most of the internal structures

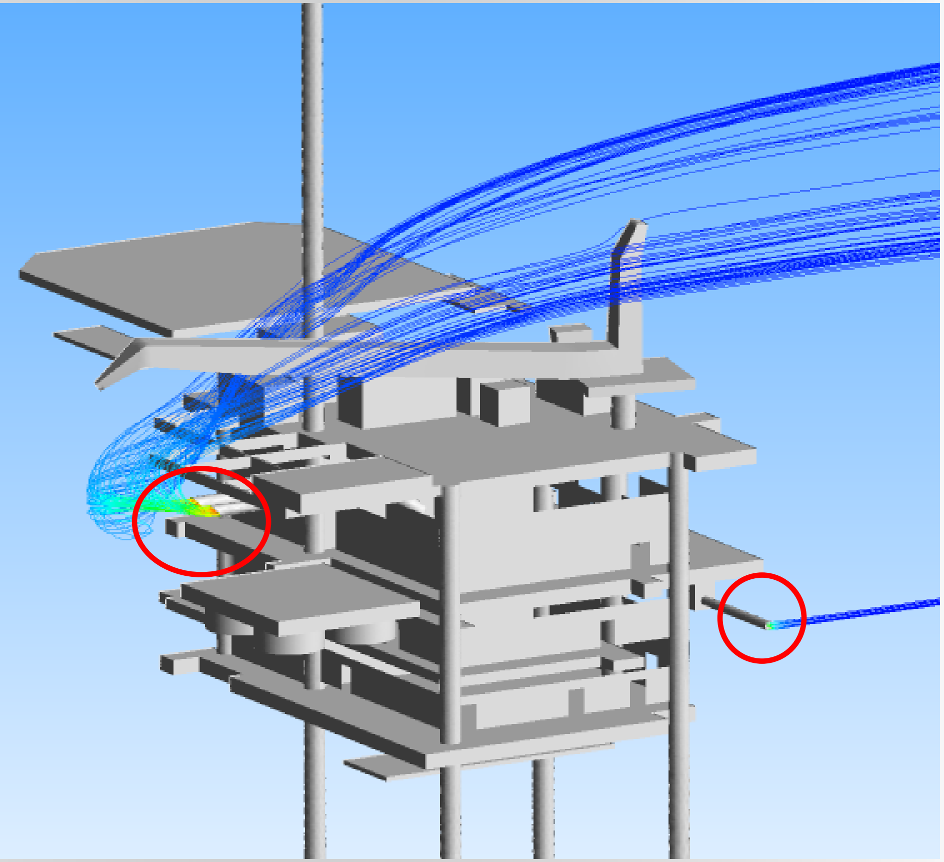


Typical induced flow pattern

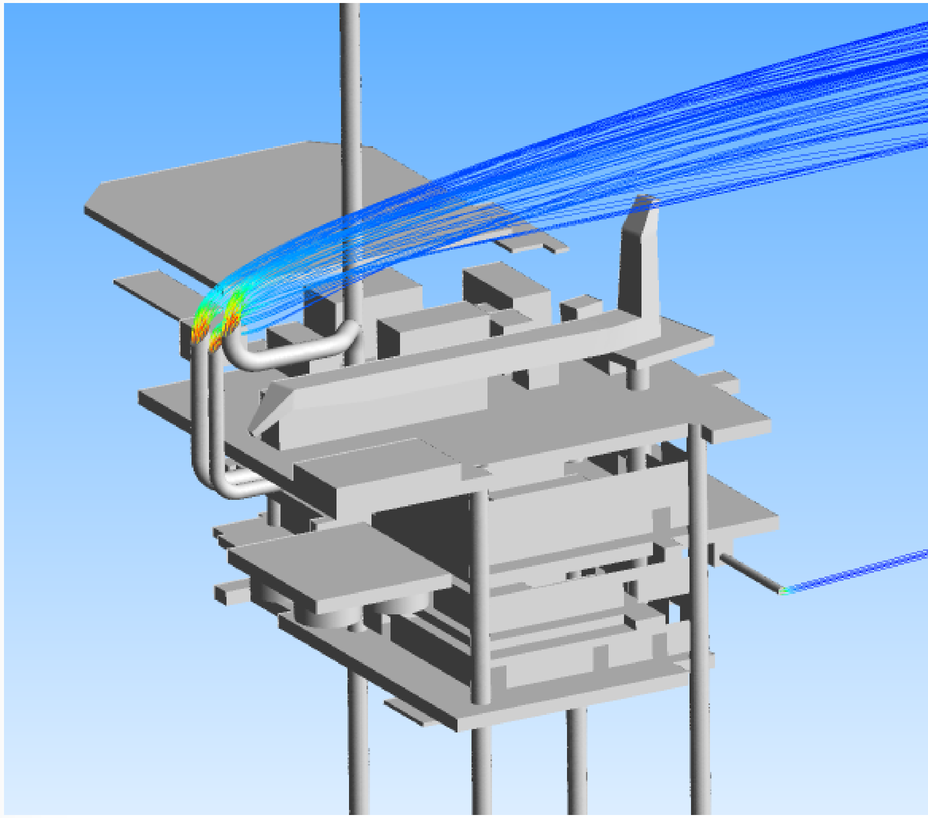
Ventilation turned ON

The flammable volume stays below the internal structures

Example: Platform Exhaust Plume



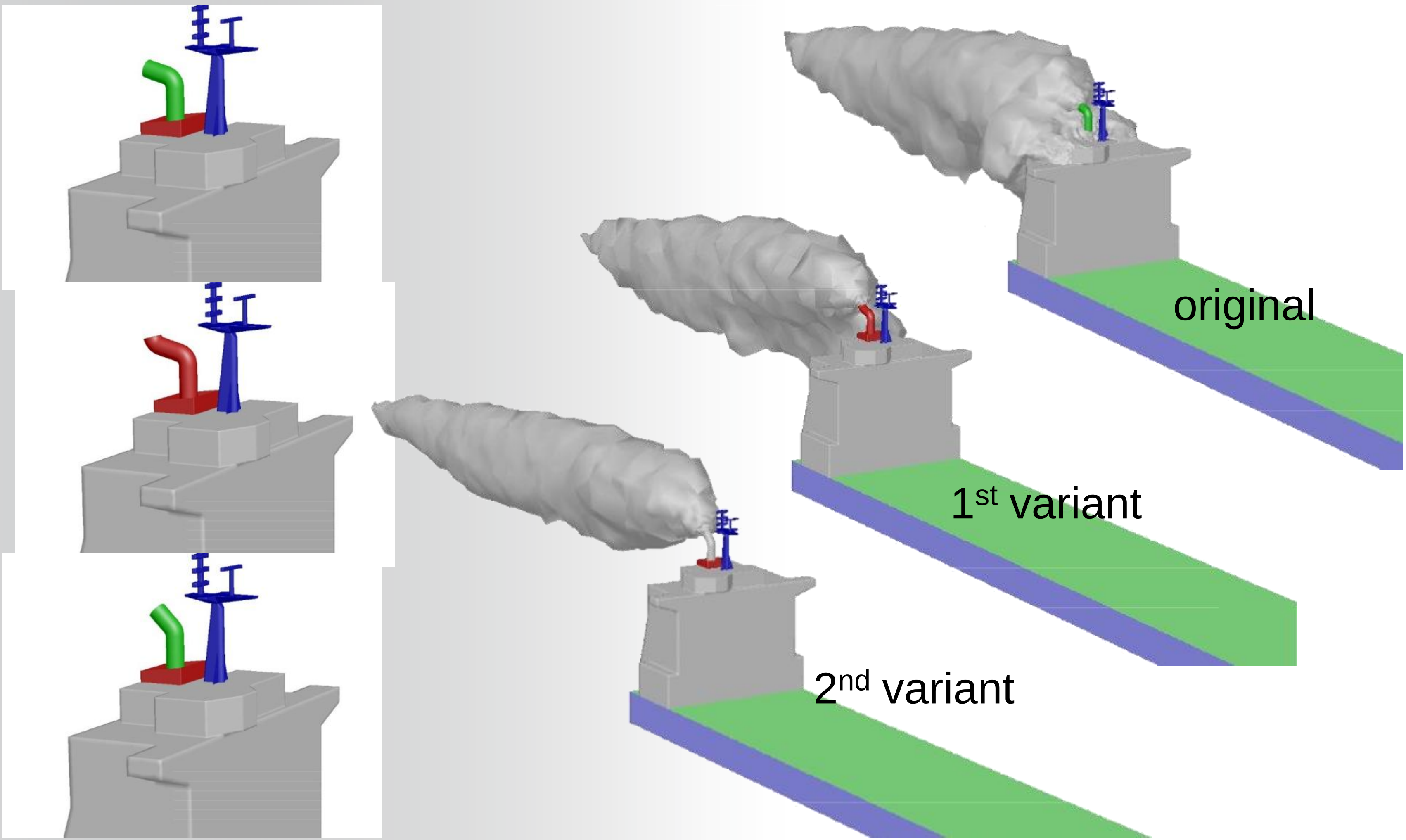
Original platform design



Modified design

The streamlines no longer interact with the platform and essentially merge quickly into the atmospheric wind flow.

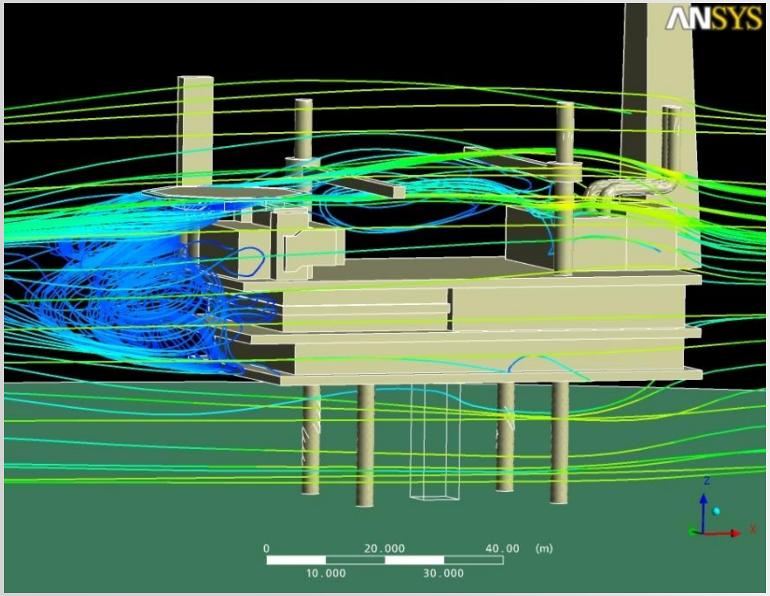
Optimization of a Funnel

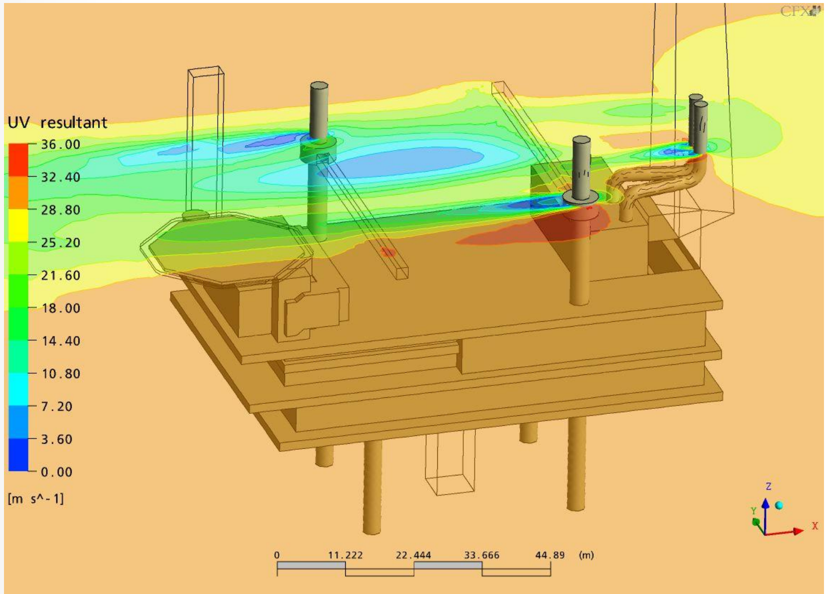


Courtesy of Daewoo Shipbuilding and Marine Engineering, Co.

Helideck Wind Environment

- Bending of streamlines creates shear and generates turbulence – hence it is important for the wind profile to be as “clean” as possible over the helideck (*fig. bottom left - streamlines*)
- GT stacks, crane arms, etc have wakes, creating more shear layers and turbulence (*fig. bottom right – resultant in-plane velocity*)
- Usual to consider 8 or 16 wind directions and a number of critical wind speeds.



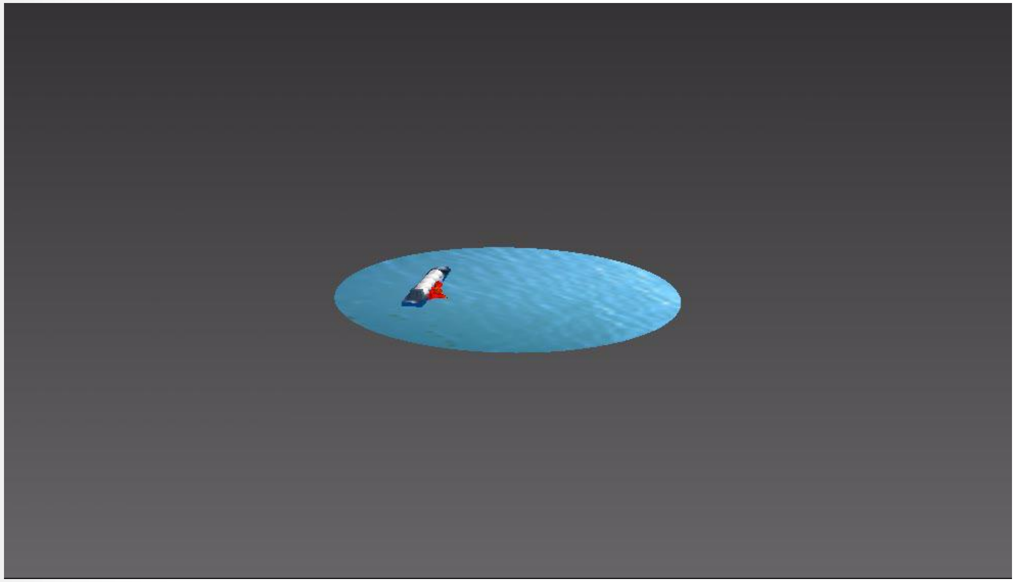


Courtesy of MMI Engineering

LNG Spill and Fire

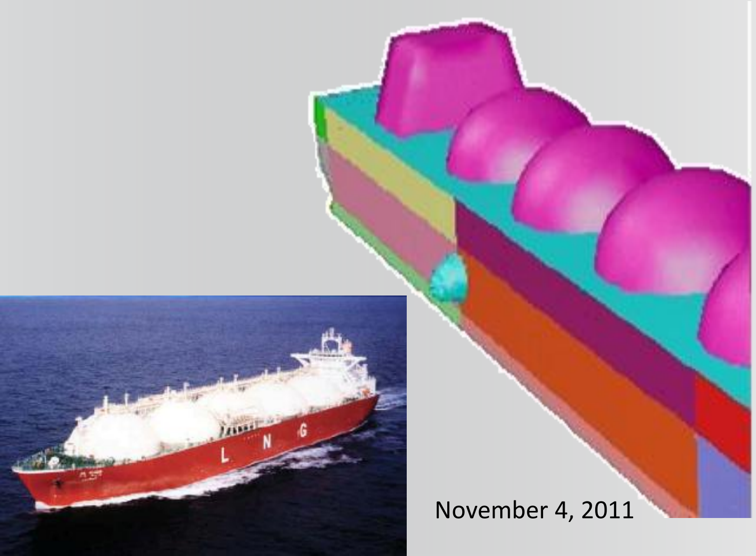
Challenges

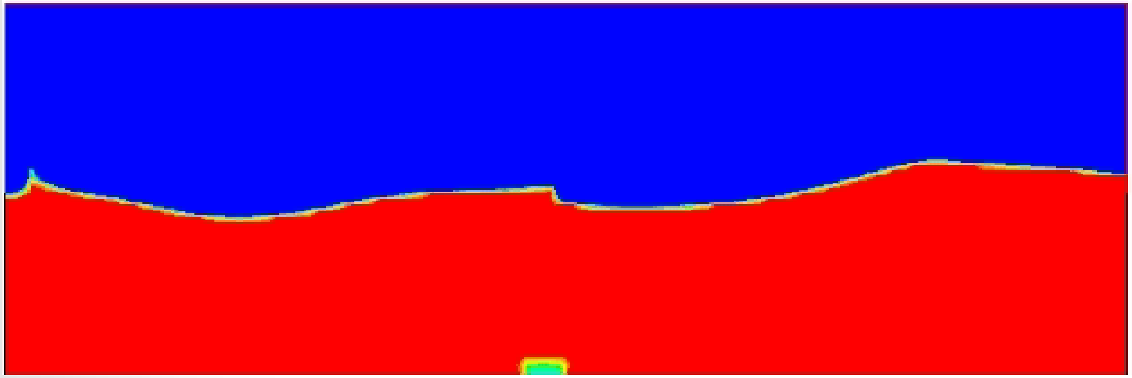
- **Accident scenario and safety concerns**
- **LNG leaks and the dispersion in the liquid**
- **LNG evaporation and tracking of the dispersed gas**
- **Vapor cloud formation and ignition as the vapor cloud reaches the water surface**
- **Spreading of the pool fire**



Pool fire caused by spread, evaporation and ignition of an LNG leak

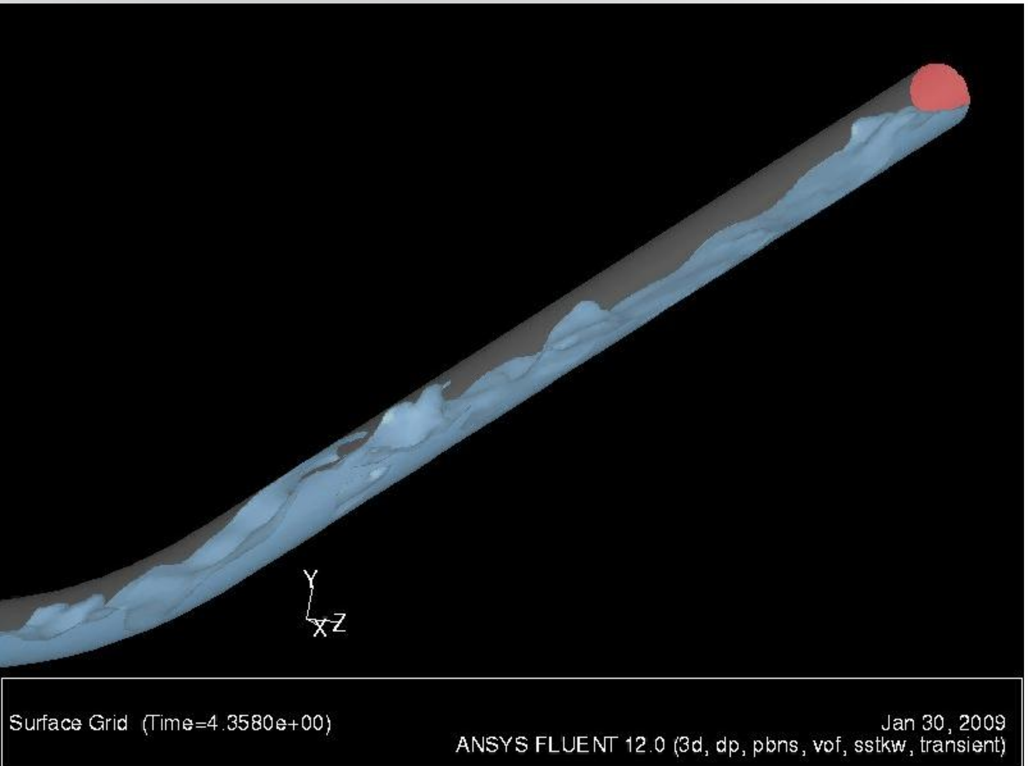
Animation Courtesy of Ensight

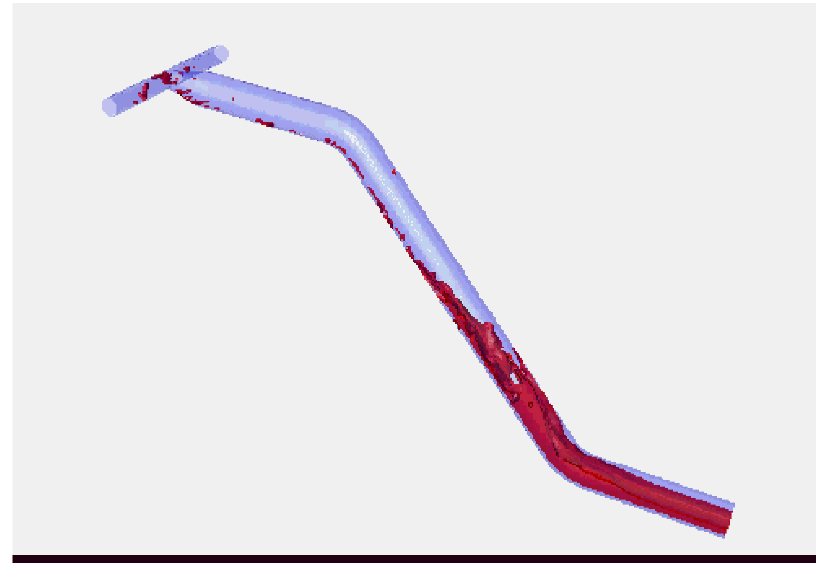


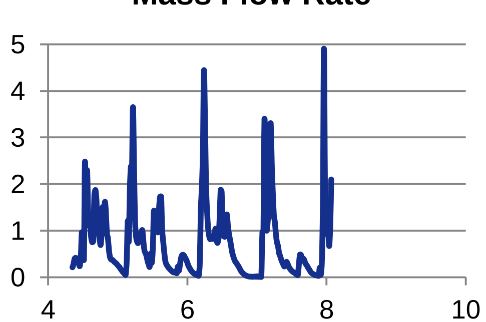


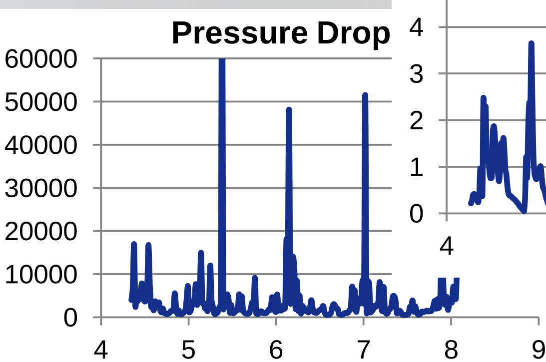
Oil leak

Terrain Induced Slugging



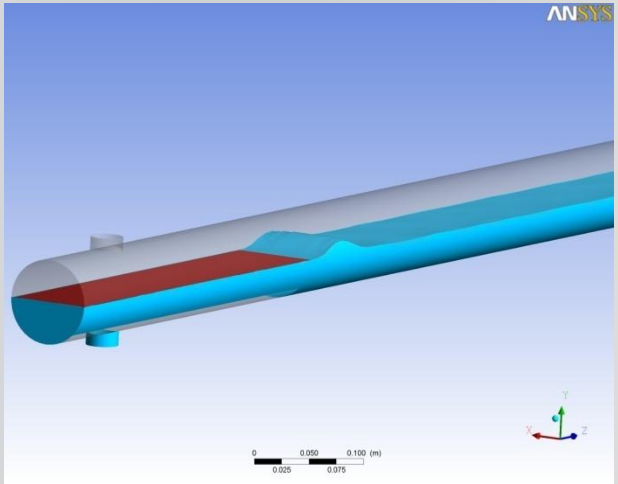


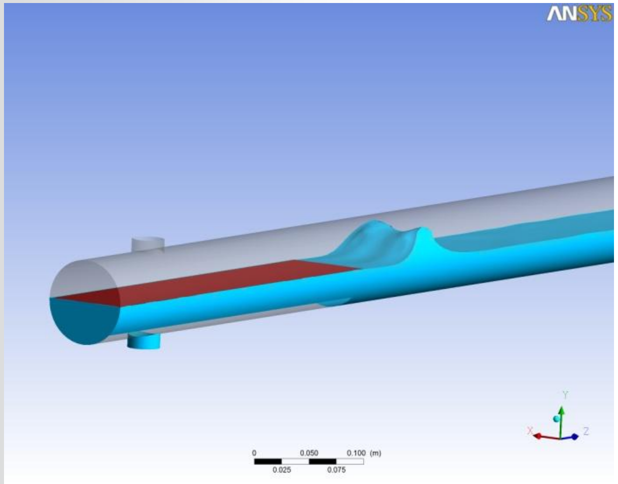


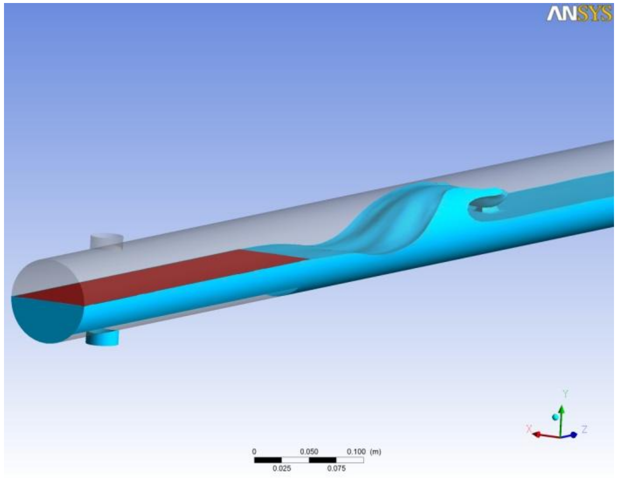


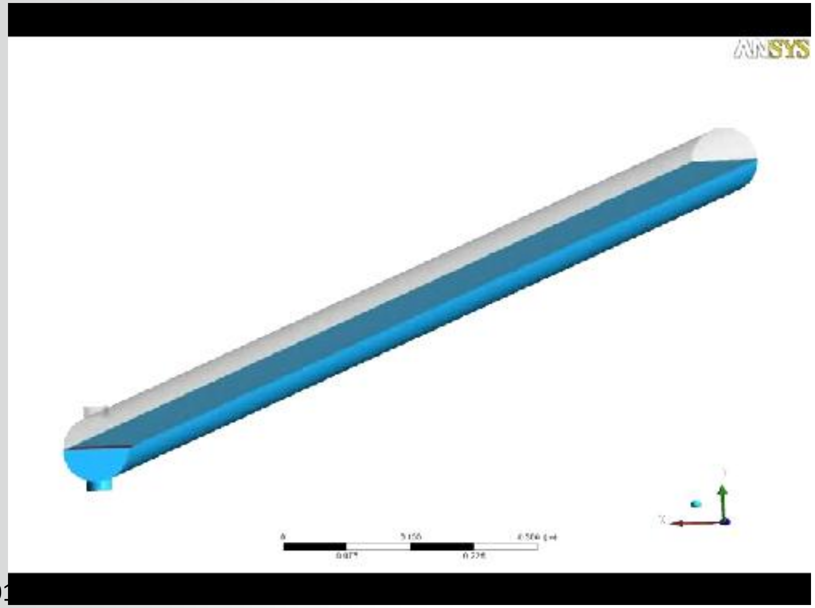
4, 2011

Hydrodynamic Slug Formation







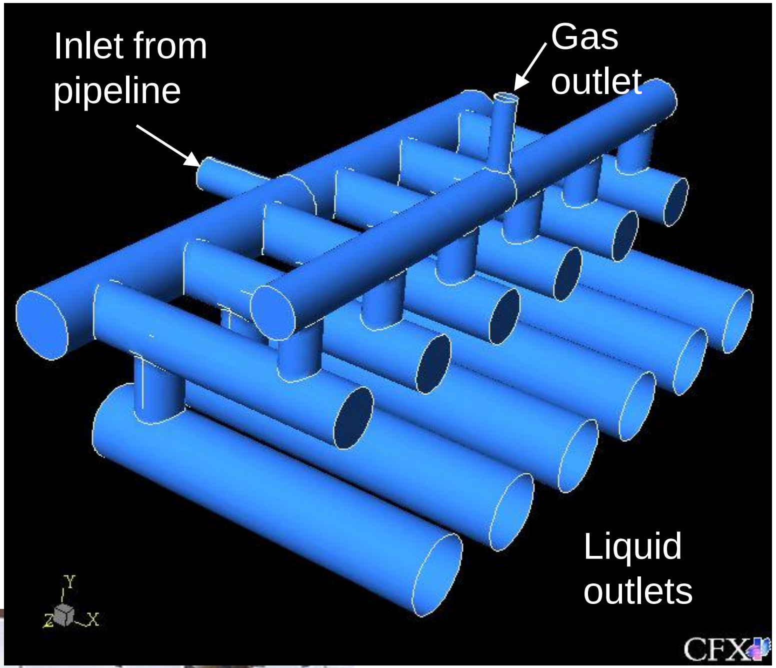


Slug Catcher at Hannibal Terminal

- Slug catchers acts as a buffer for volumetric and pressure shocks because of the slugging in upstream pipelines
- Huge investments required because of their large footprint with no option for a failure

We demonstrate the ability of

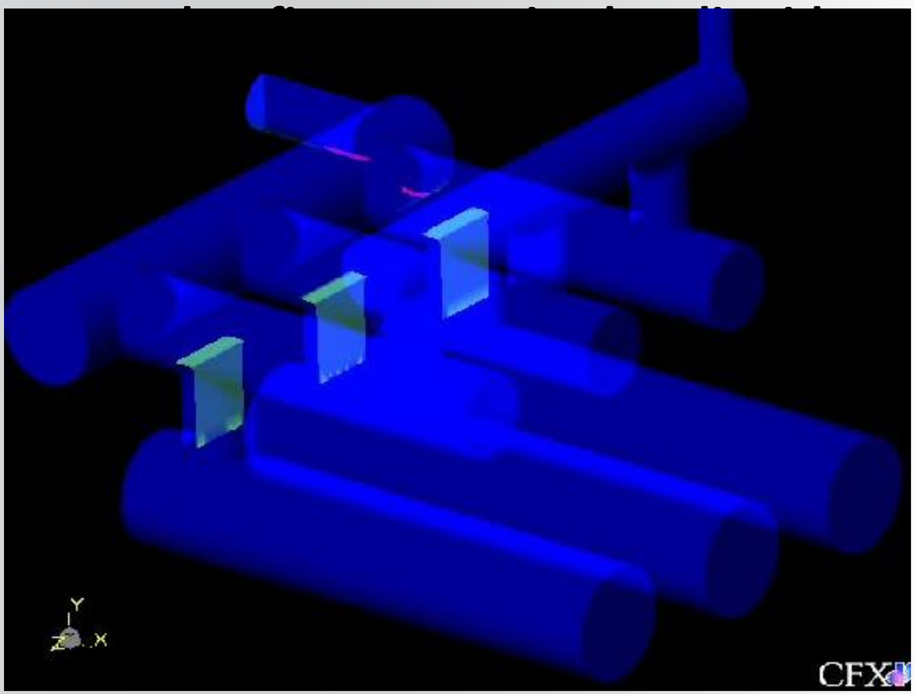
- ANSYS CFD to enable engineers to take critical engineering decisions regarding the working of these equipment
- Coupling with OLGA



Slug catcher at the Hannibal terminal, U.K.

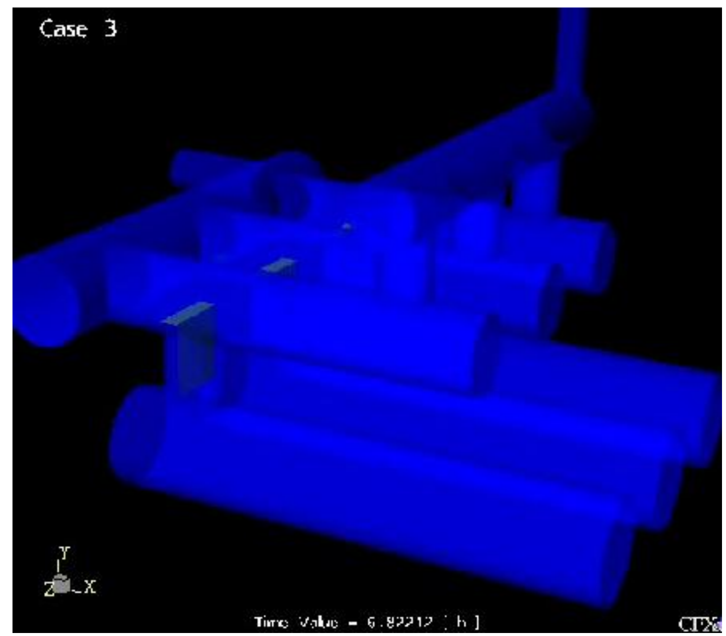
Slug Catcher - Results

- Current condition
  - No liquid carry-over to gas outlet
  - first separation finger (nearest symmetry plane) partially fills with liquid



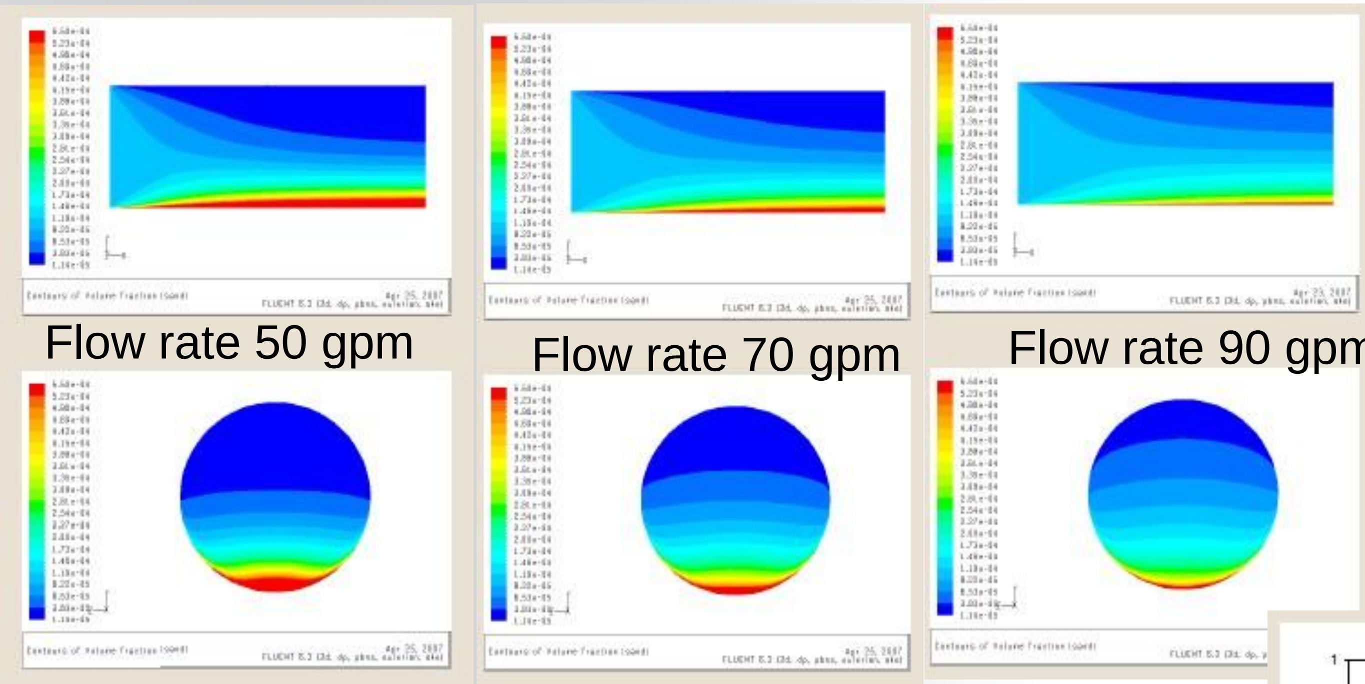
Can slug catcher cope with increase in capacity of pipeline? – Yes!

Liquid carry-over only in form of fine aerosol



Estimated cost of modifying slug catcher $25M

Sand Transport



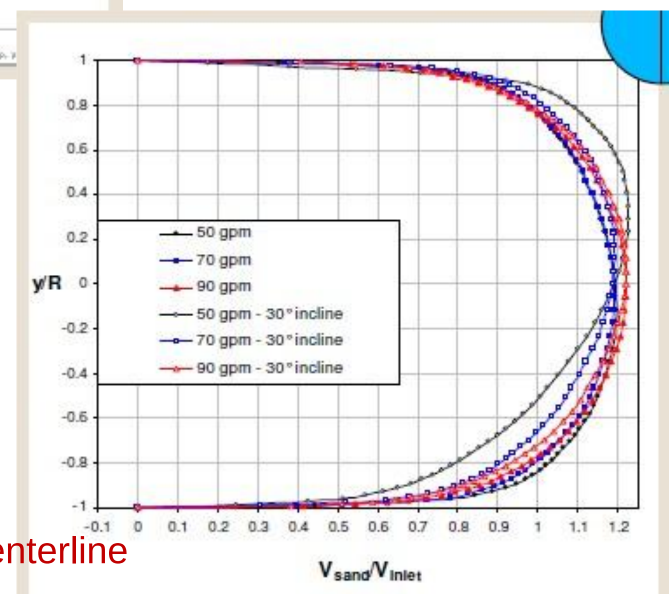
Flow rate 50 gpm

Flow rate 70 gpm

Flow rate 90 gpm

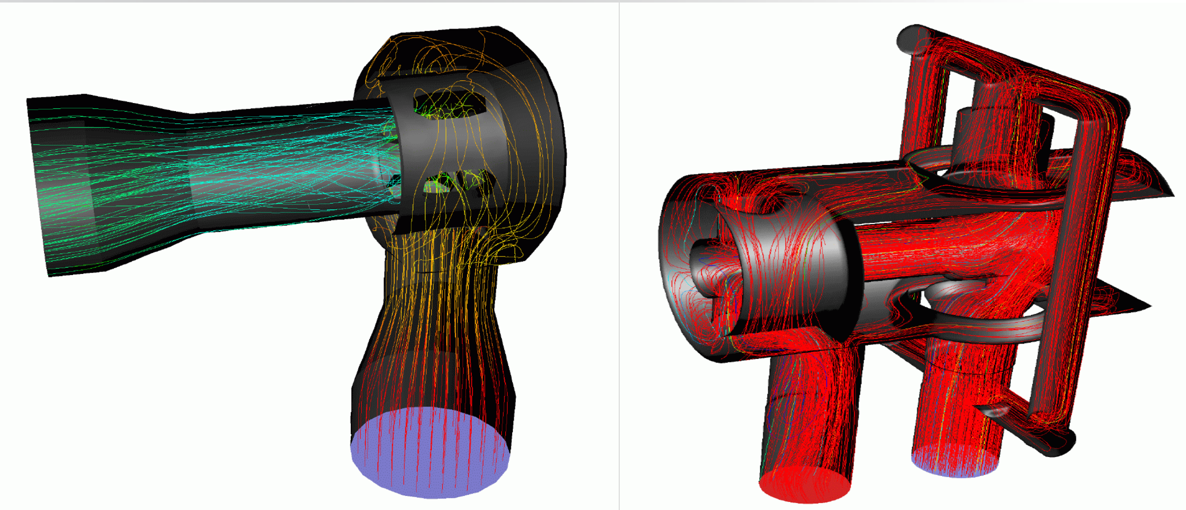
Slight skew of profile due to solids settling in horizontal pipe that increases in the inclined pipes

Concentration at the wall and a non-zero velocity of sand at the wall suggests the flow regime is moving bed.

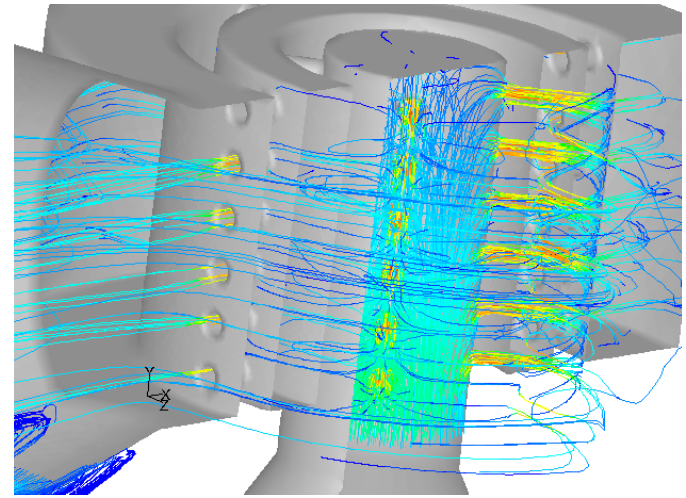


Sand Profiles at outlet vertical centerline

Oil and Gas Production Equipment
Valves, Chocks, Regulators

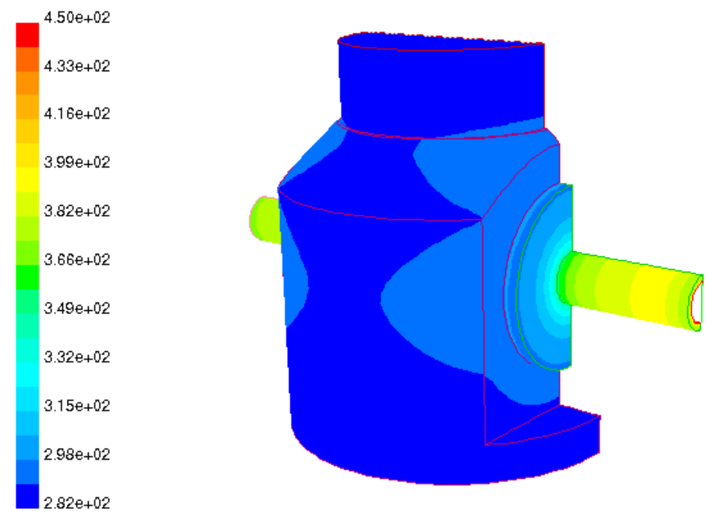


Choke Valve (left), Subsea Regulator (right), Courtesy of Hydril



Flow streamline for a petroleum control valve

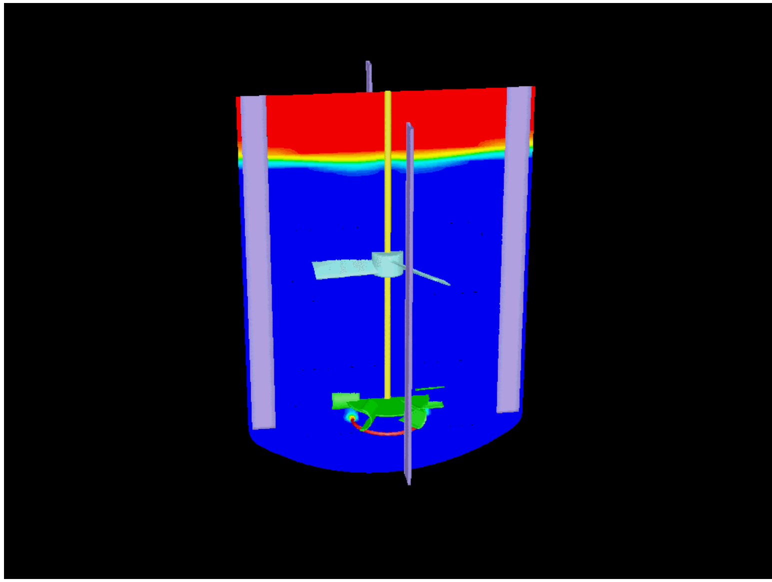
- **CAE Solutions**
  - Comprehensive engineering solutions for design, analysis, production and operation of these type of devices
  - Understand structural and thermal stresses to increase reliability and safety
  - Predict erosion spots and design to reduce its impact
  - Design to minimize cavitation
  - Improve pressure drop, and the range of the equipment operability
  - Accelerate design by performing parametric and design optimization



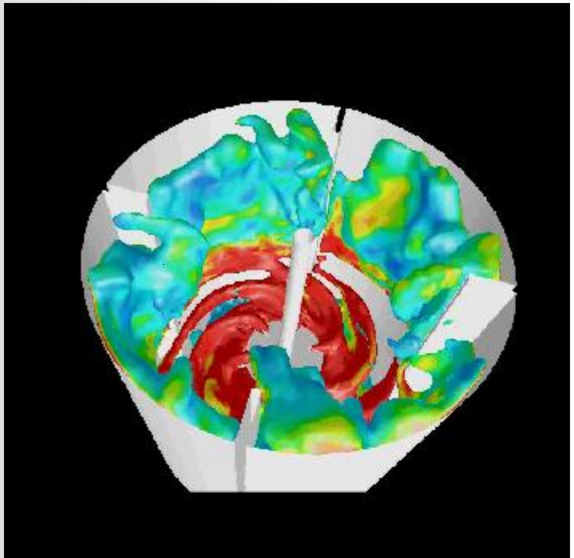
Surface temperature on a valve surface

Mixing

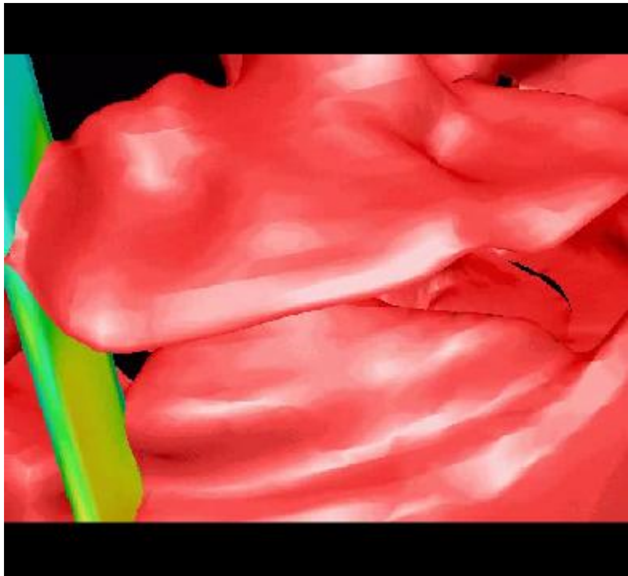
- **Complex Phenomena**
- **Many parameters: Operating condition, selection of feed location, impeller speed, scaleup**
- **Blending, reacting and suspension of multi-component multi-phase material**
- **CFD can provide**
  - **Optimize vessel geometry and select the right internals, sparger, dip tube and feed location, impeller speed**
  - **Calculate forces on impellers**



Gas dispersion in a mechanically agitated reactor



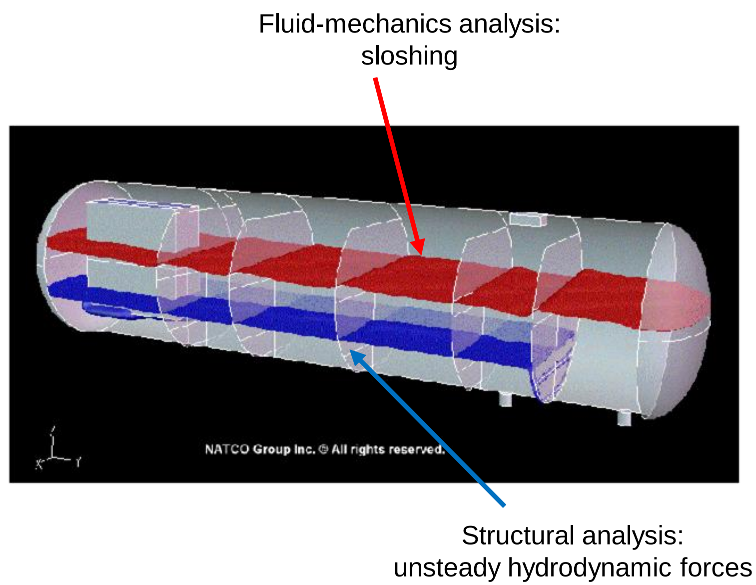
Macro mixing structures colored by vorticity contours in a mixing tank with 6-blade Rushton impeller

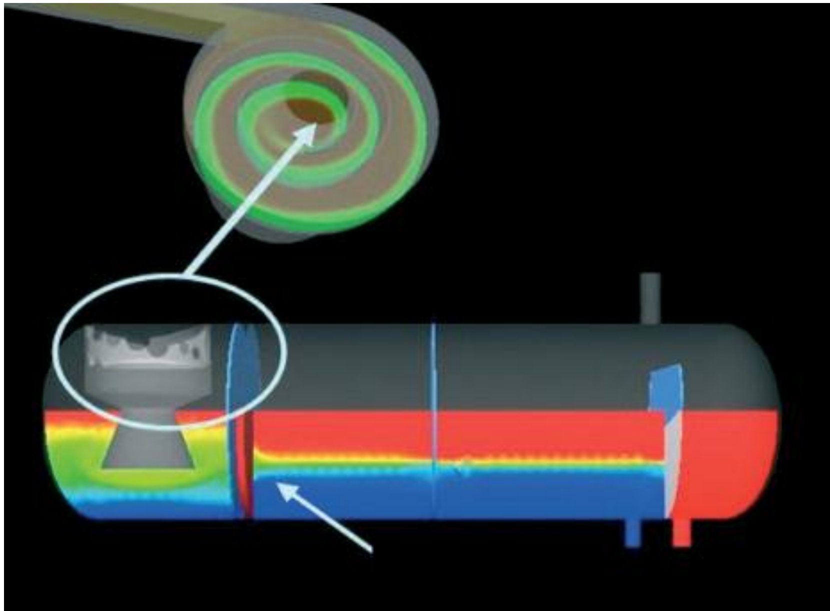


Oil and Gas Equipment

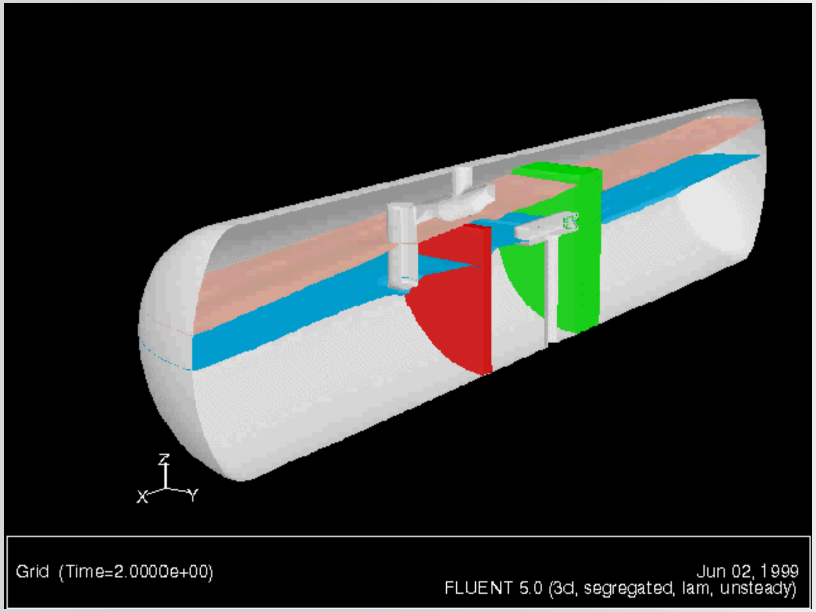
ANSYS multidisciplinary simulation in design, optimization and manufacturing of oil and gas separator

- **Baffle and vessel design**
- **Structural integrity**
- **Separator efficiency**
- **Virtual product design**
- **Design Explorer**
- **Fluid structure interaction**
- **Advanced multiphase models**
- **Population balance capabilities**

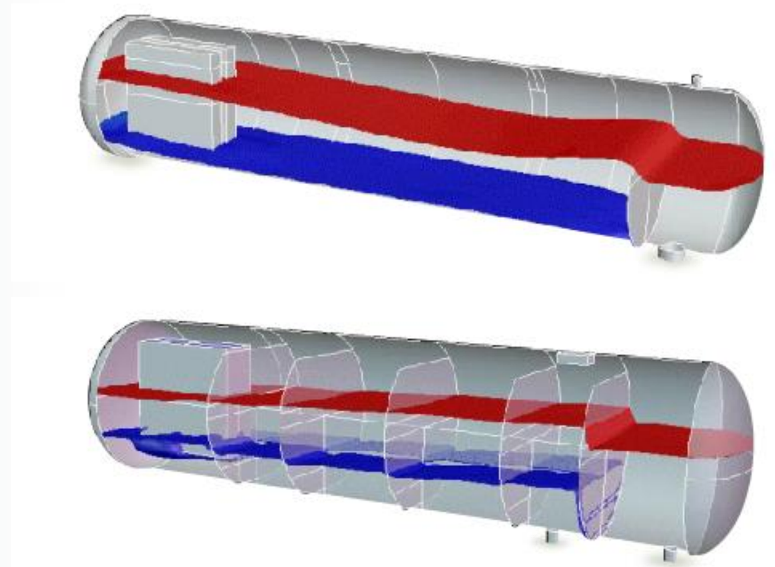




Sloshing Separator Tank Design



FPSO Separator



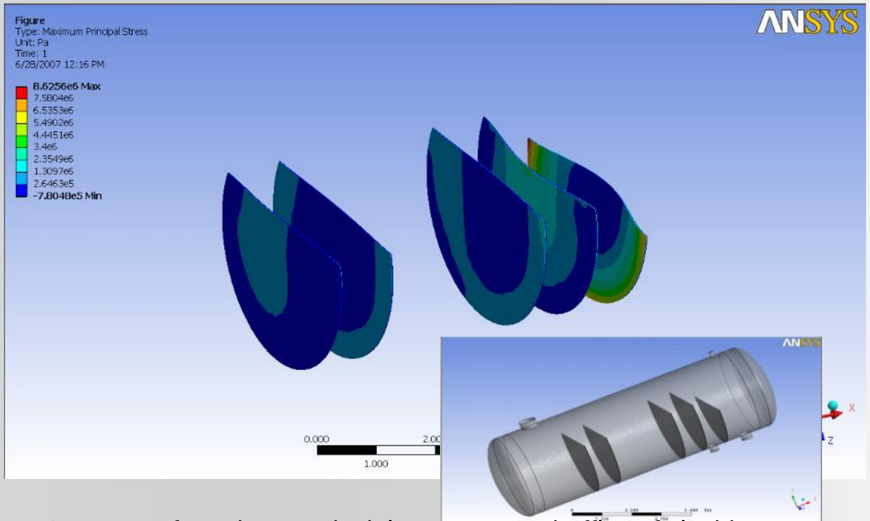
Simulated effect of baffles in reducing sloshing in a oil-water-gas separator

ANSYS Solutions

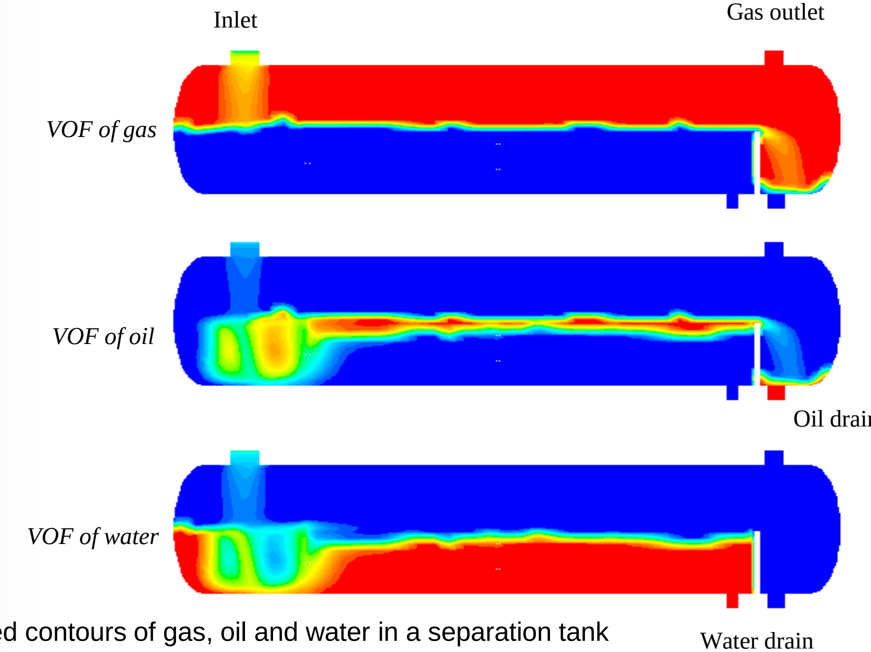
- **Estimate the hydrodynamic forces caused by sloshing in 6 degrees of freedom**
- **Evaluate damping and performance of internals such as baffles and coalescers**
- **Optimize the shape and location of inlets and outlets, and performance of any upstream gas separators**
- **Design for fatigue and structural stresses on vessel (pressure vessel codes), the supports and the internals**

Some images courtesy of NatcoGroup

Liquid-Liquid Separator Tank Design



Contours of maximum principle stress on a baffles of sloshing separation tank



Colored contours of gas, oil and water in a separation tank

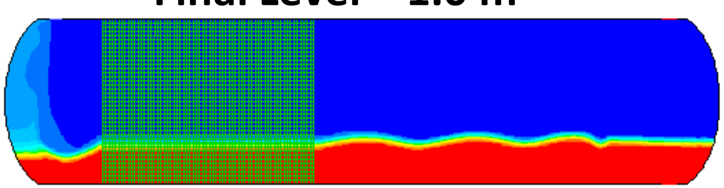
ANSYS CAE Solutions

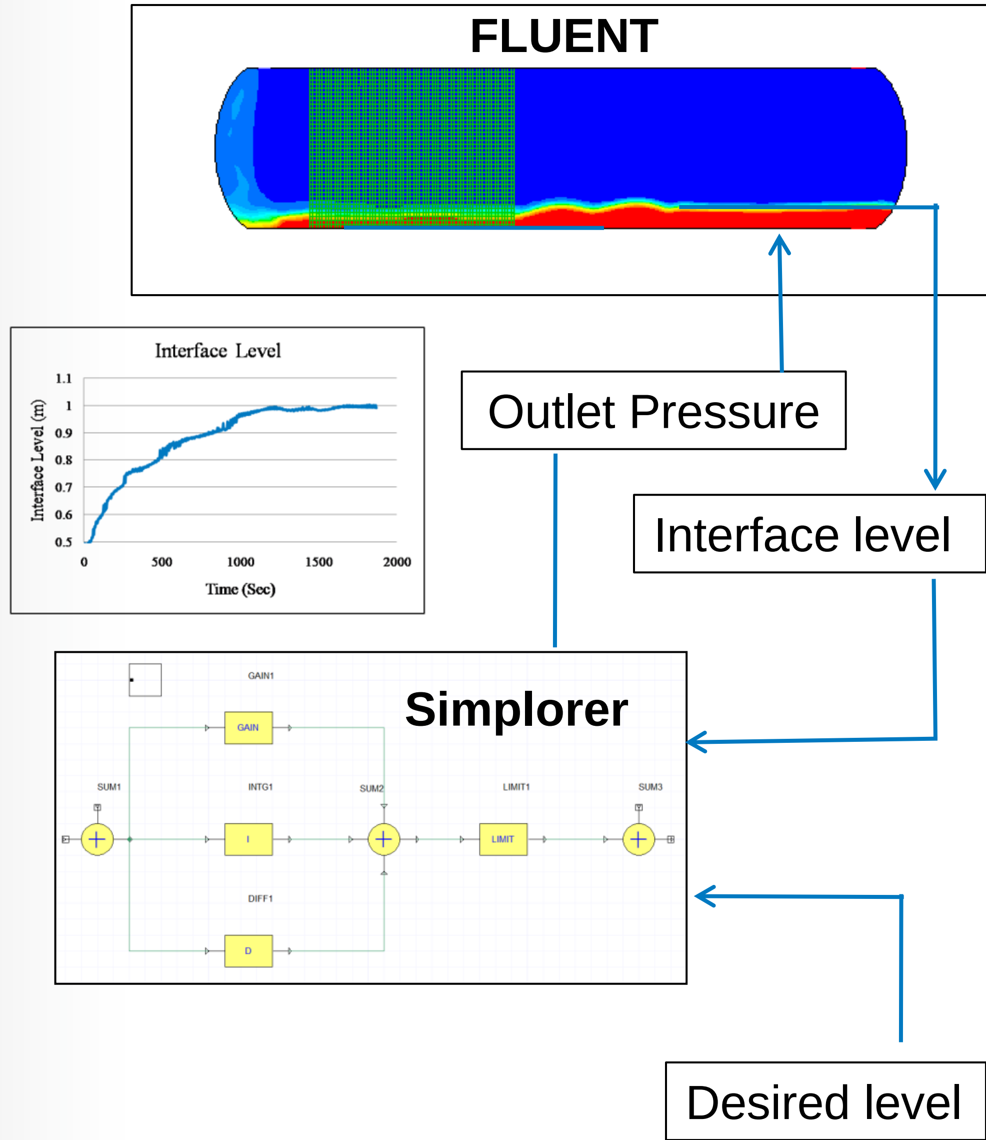
- **Account of multiphase flow and its behavior in different parts of the separator**
- **Include effect of particle size distribution, coalescence and breakup using population balance**
- **Optimize design and placement of internals including baffles, pores and packed sections, and size and location of inlets and outlets**
- **Provide insight for design of separator sections including sizing, pressure drop analysis and overall performance**

Gravity Separators - Co-Simulations with FLUENT and Simplorer

- **PID controllers are used if**
  - Desired output is known
  - But exact conditions leading to the desired output is not known
- **Gravity separators**
  - Desired interface level is known
  - Outlet pressure to maintain this level is not known

Final Level = 1.0 m





Jumper Pipe 2-way FSI - Multiphase

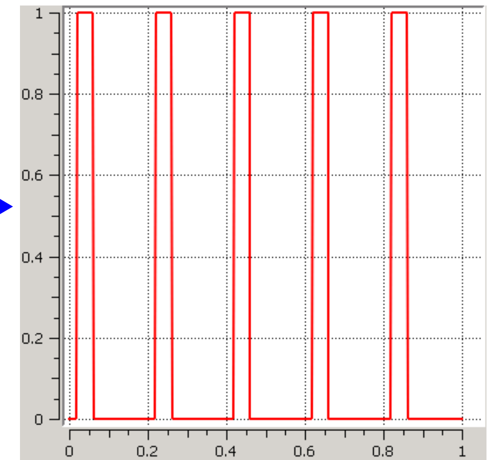
Fluid: water + compressible air

- Reference pressure: 150 atm

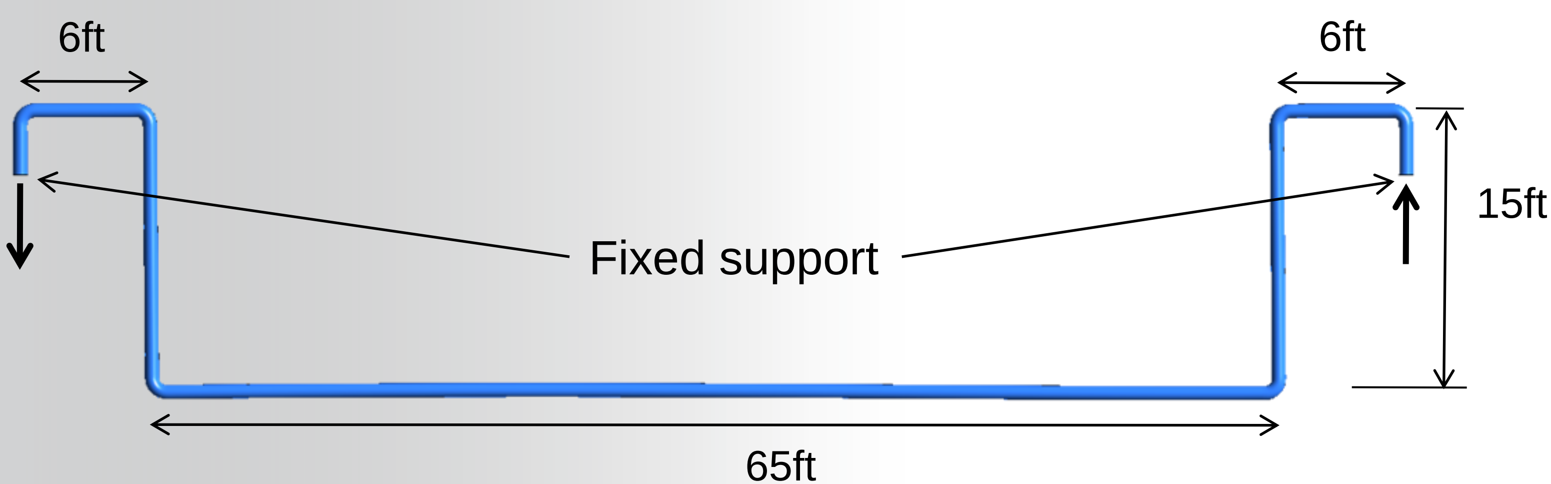
Fluid velocity: 15 m/s (5 bubbles/s)

Stainless steel pipe: 10” ID, 1” thick

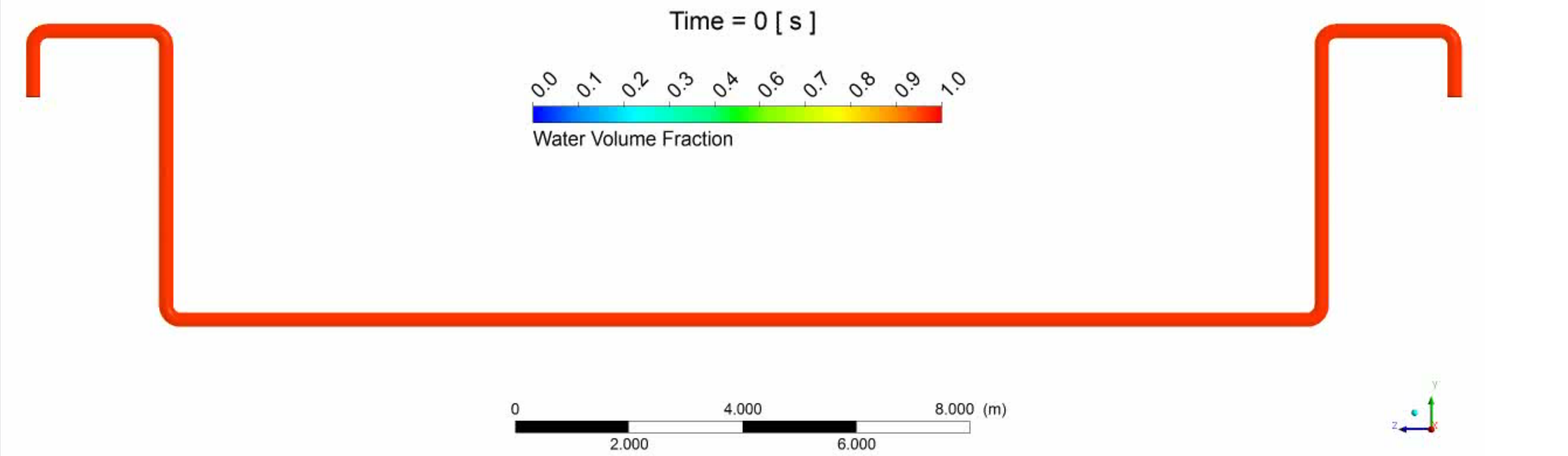
On Inlet and outlet: Fixed support

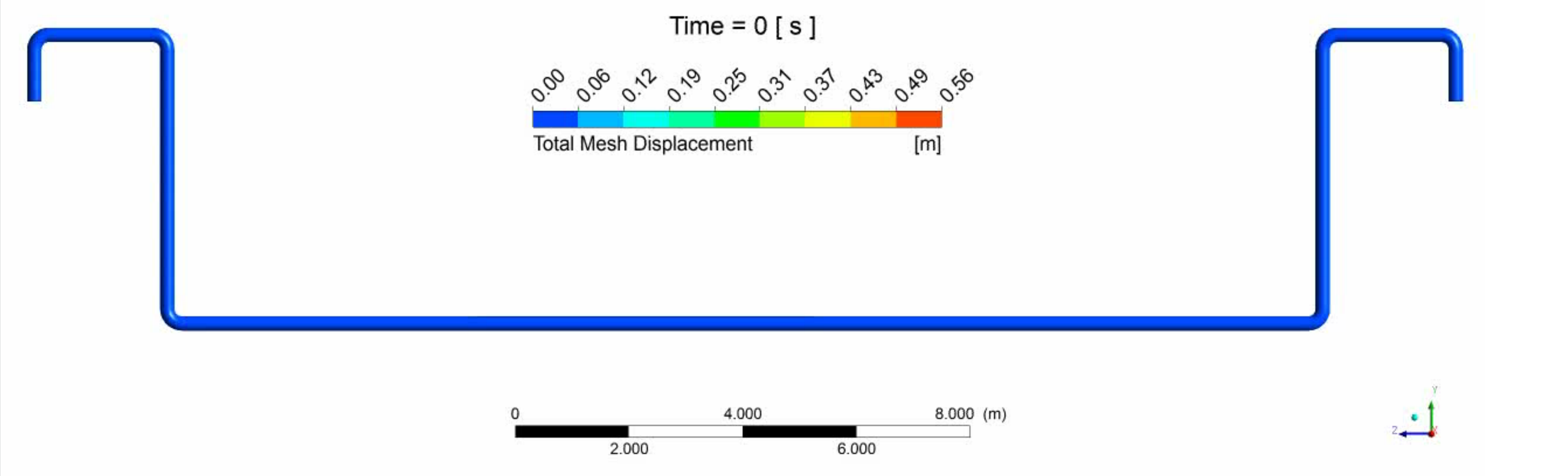


Inlet bubble profile



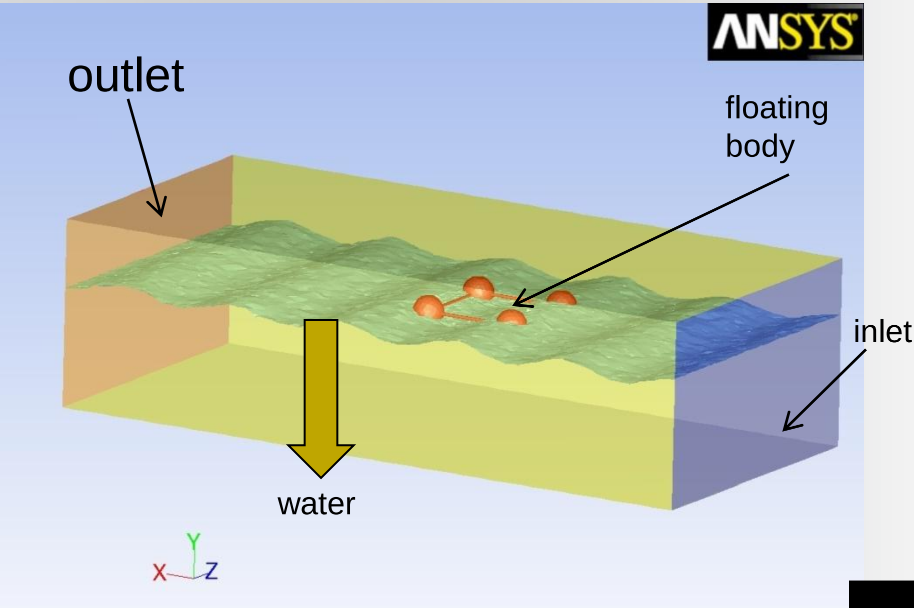
2-way FSI: Multiphase - Animations

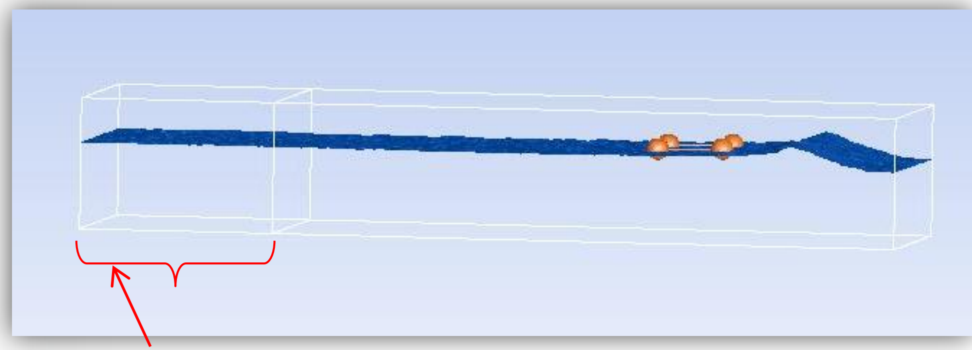




Displacement amplified for display

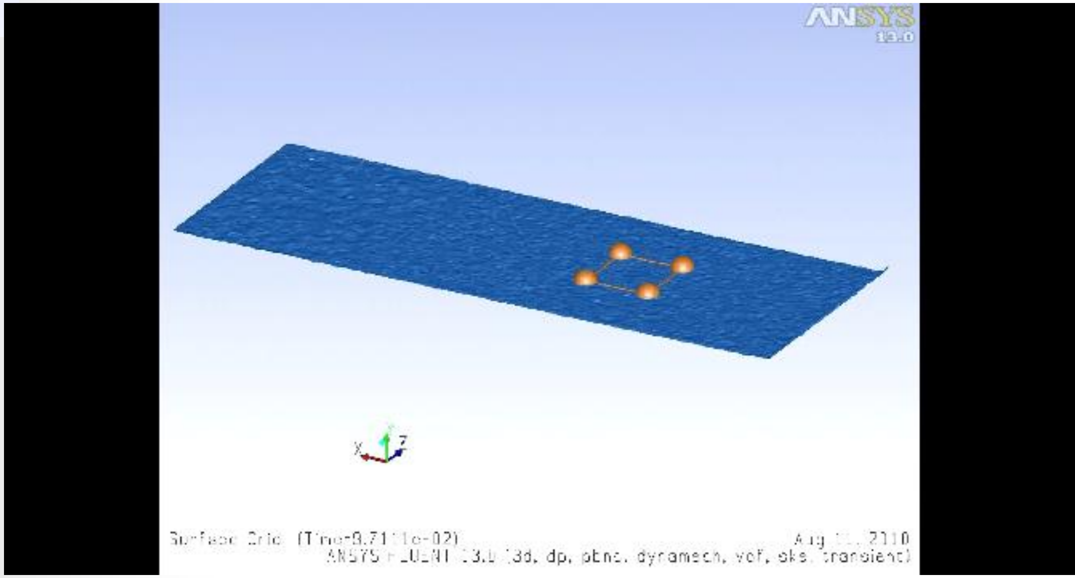
Wave Interaction with a Floating Structure



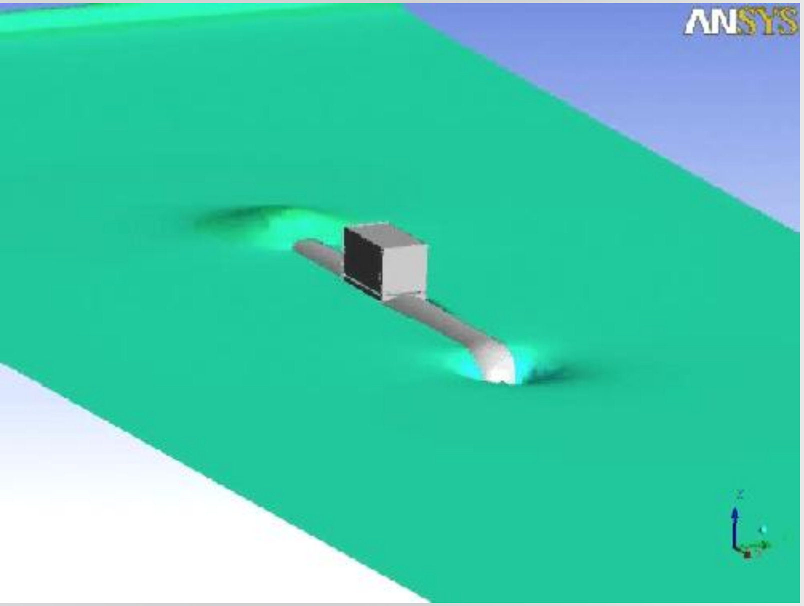


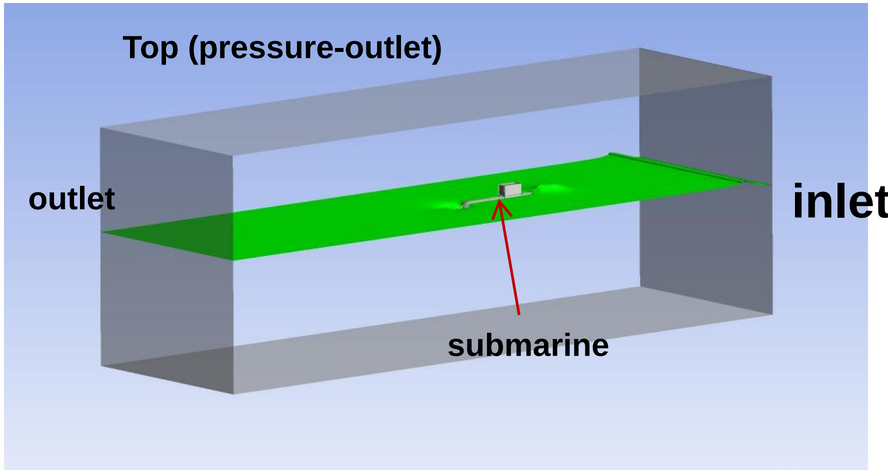
Numerical beach fluid zone near the outlet

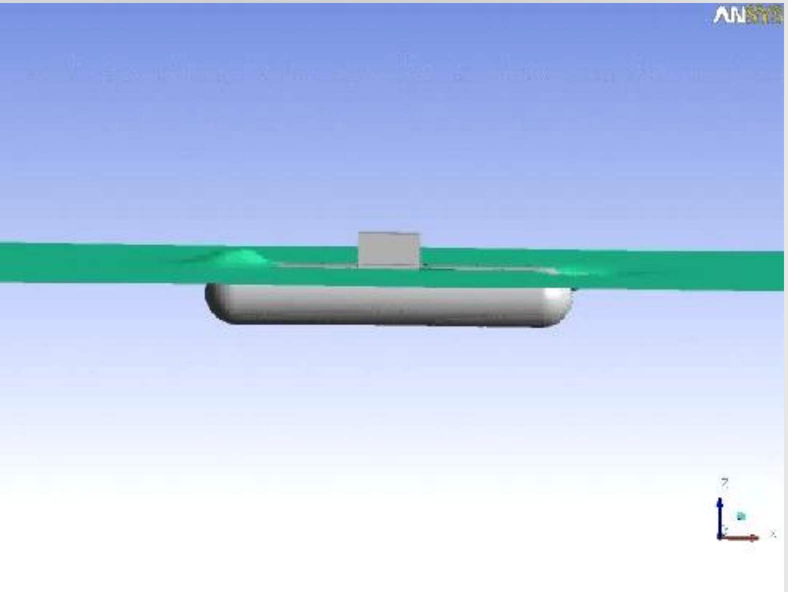
- **6DOF Implicit Solver**
- **Standard k-epsilon turbulence model**
- **Fifth order Stokes**
- **Numerical beach condition**

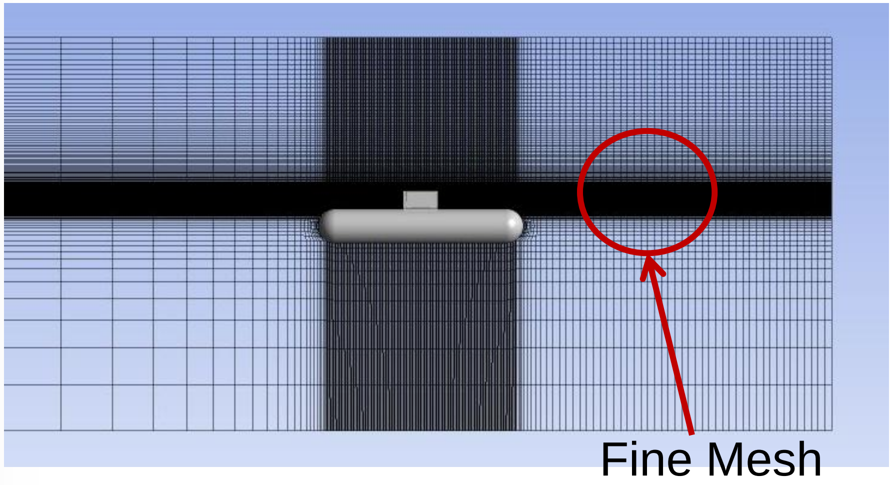


Wave Slamming



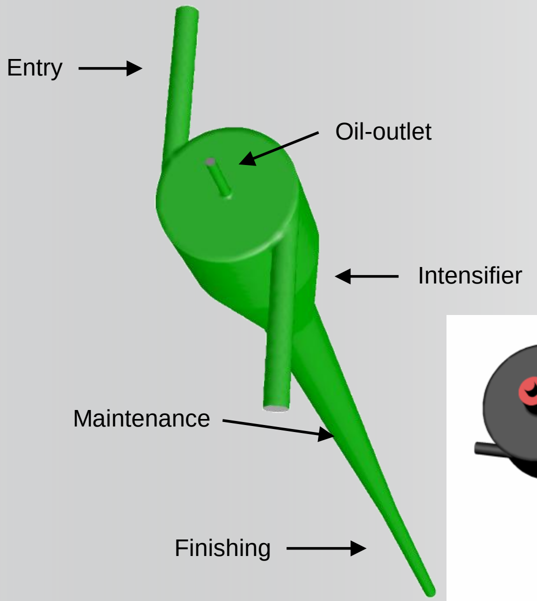


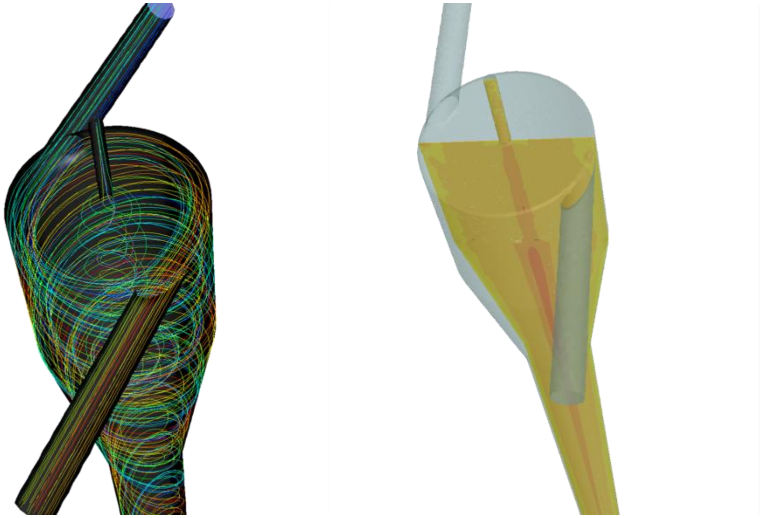




Open channel wave BC to generate waves
SST k-omega model
Fifth order Stokes
Numerical beach condition

Equipments Design/Trouble Shooting: Hydrocyclones



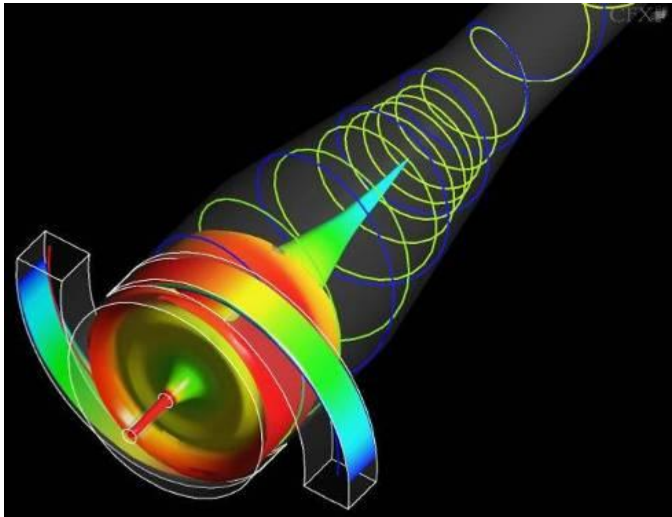


Pathlines deoiling hydrocyclone

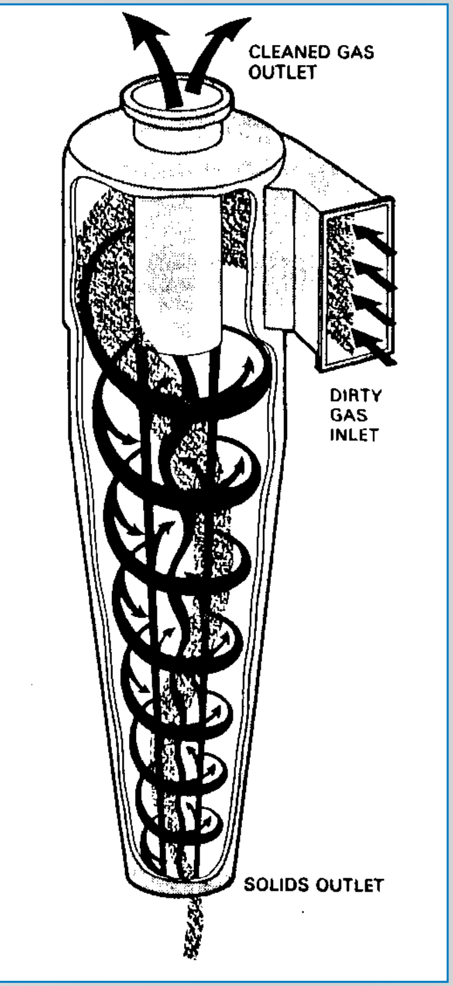
Volume fracture of oil (mixture)

ANSYS CAE Solutions

- Design inlet configuration and geometry for high angular velocity
- Evaluate separation efficiency for different oil to water and water to oil mixtures
- Optimize placement of vortex finder
- Develop multi-stage or collection of separators



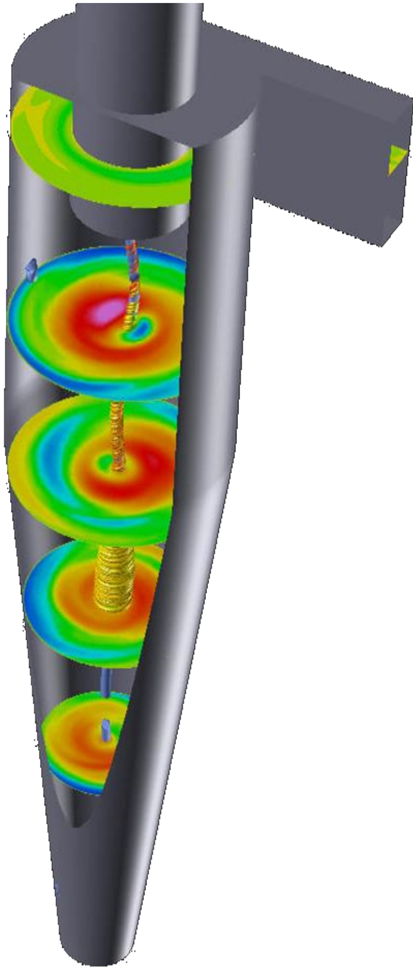
Equipments Design/Trouble Shooting: Cyclones



Schematic of complex flow motion in a cyclone separator

ANSYS Solutions

- **Optimize inlet design to reduce erosion, increase efficiency and find the range of device's usability**
- **Geometry and design optimization for various particle loading**
- **Relevant to many applications and any separator shapes, accounting for**
  - particles mass, diameter, loading
  - flow characteristics, pressure drop,
  - welding and structural stress, fabrication, erosion
  - performance in stages or in an assembly

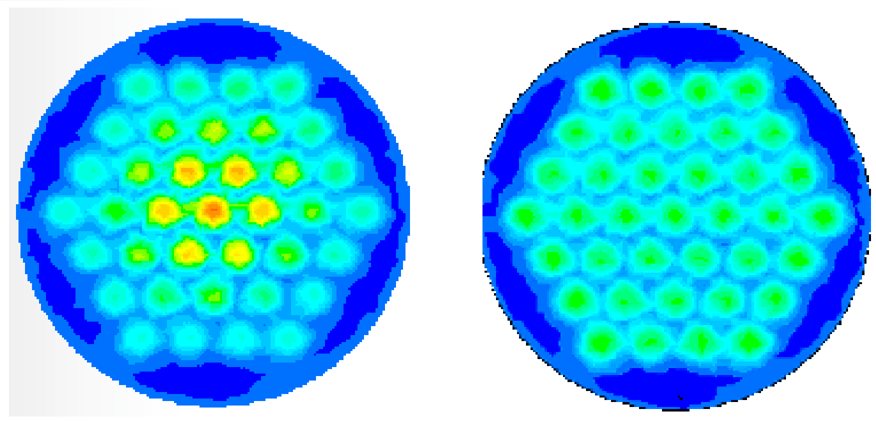


Composite CFD results illustrating the vortex core and flow velocity at various axial planes

Equipments Design/Trouble Shooting: Heat Exchangers

Challenges

- Heat exchanger efficiency
- Avoid fouling, maldistribution
- Sizing and type selection
- Thermal and structural design
- Fabrication and manufacturing practices



Maldistributed *Improved*

Axial velocity at the inlet to the tubes; no "hot spots" visible for the improved design

Under-performing, 324 tube, heat exchanger

ANSYS CAE Solutions

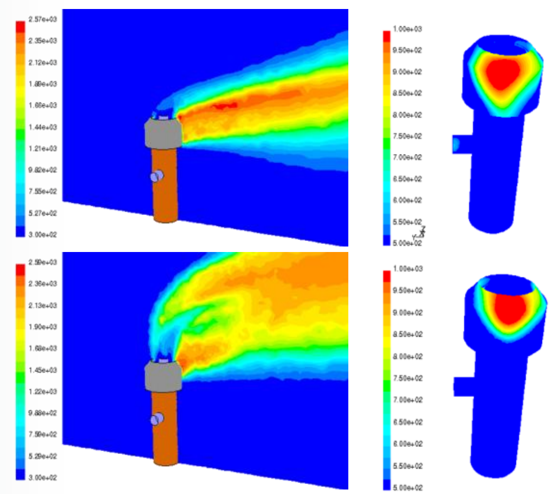
- Design to code using ASME pressure vessel tools and analysis
- Retrofit exciting devices for process improvement and efficiency
- Look at flow and heat transfer to design around dead or hot spots
- Design tubes, baffles and heat exchangers geometry to meet overall process objectives

Some images courtesy of Cal Gavin engineers

Combustion Systems: Flares

Challenges

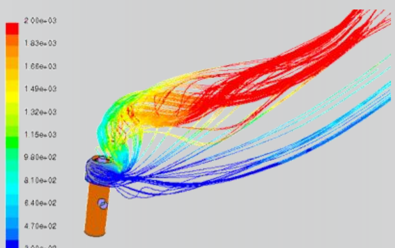
- **Control flame shape and flare performance for different fuel and wind velocity**
- **Avoid back mixing and flame blow out**
- **Design flare support system and placement**
- **Reduce maintenance cost**

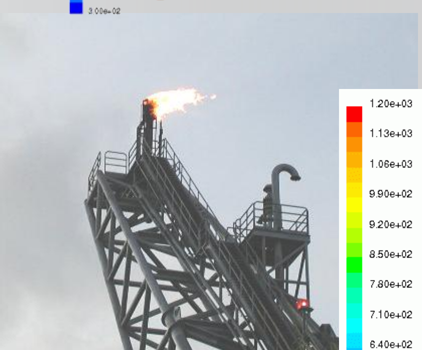


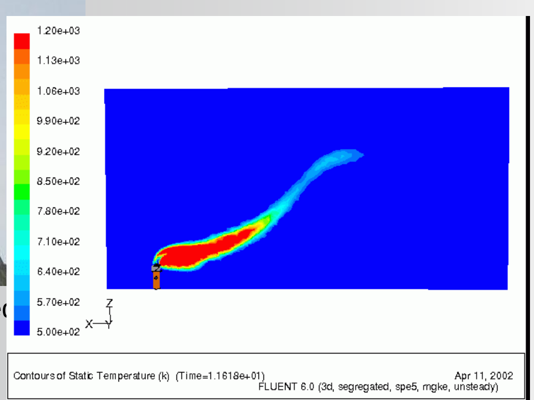
Fuel to Wind ratio 1:2

Fuel to Wind ratio 2:1

Flame shape and shroud surface temperature for two different fuel and wind ratios







Flare flow pathlines, colored

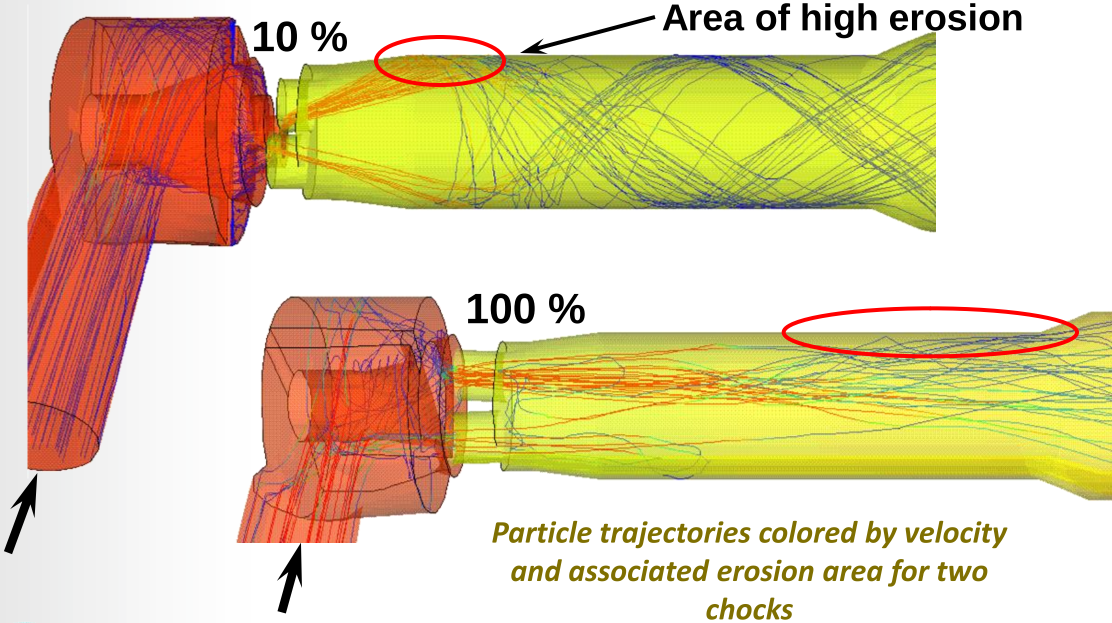
ANSYS CAE Solutions

- **Optimize flare design, shape and burner internals**
- **Compare performance of different arrangements and best placement**
- **Perform radiation and heat transfer studies from the flame**
- **Learn about thermal and structural stresses.**

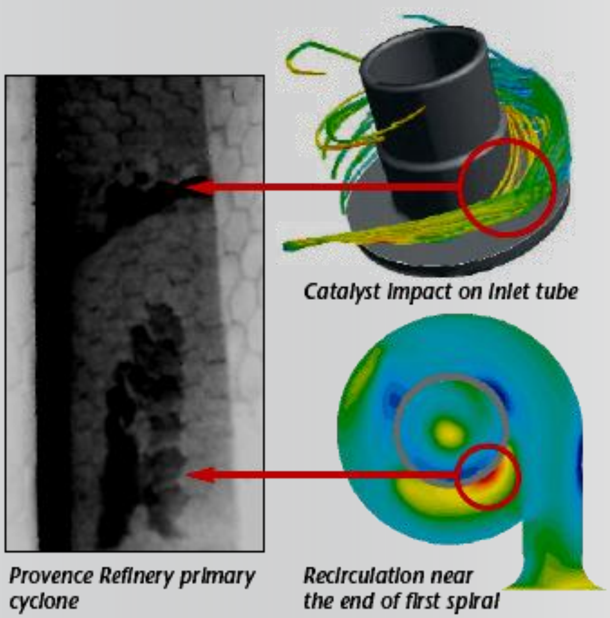
Erosion

– CFD modelling can find erosion rates for field conditions for equipment lifetime

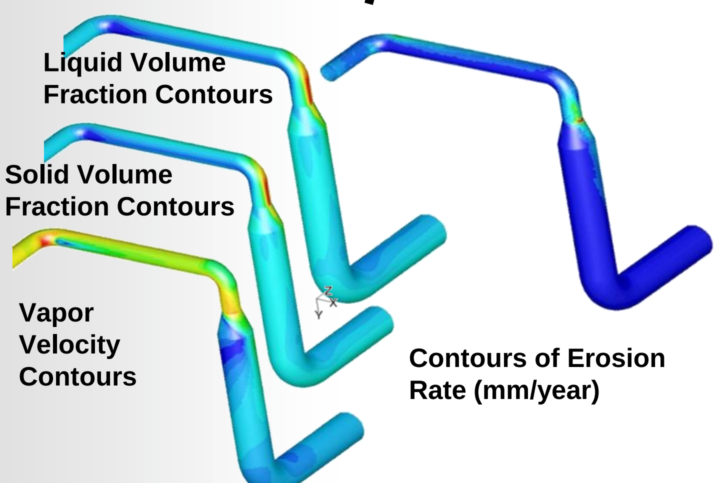
– Maximum erosion in complex flows and geometries can be predicted to with a good accuracy



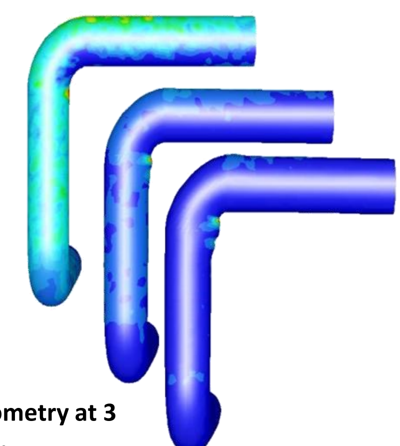
Particle trajectories colored by velocity and associated erosion area for two chocks



Courtesy of Total—Process and Refining Division



Erosion in a Pipe Assembly



Elbow geometry at 3 different times

ANSYS

Erosion

– Erosion in elbow and or reduction can leads to material depletion and leaks

– An estimate of life of given piping and evaluation of extend of wear are required

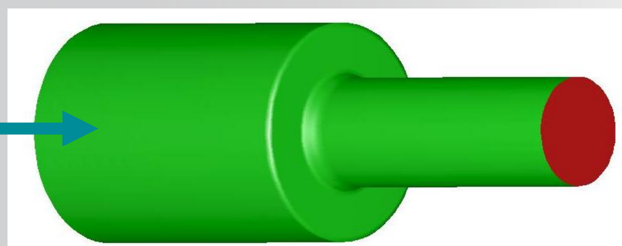
– Substantial economical maintenance and shut done costs

- Eroded material is removed leading to better material thickness predications

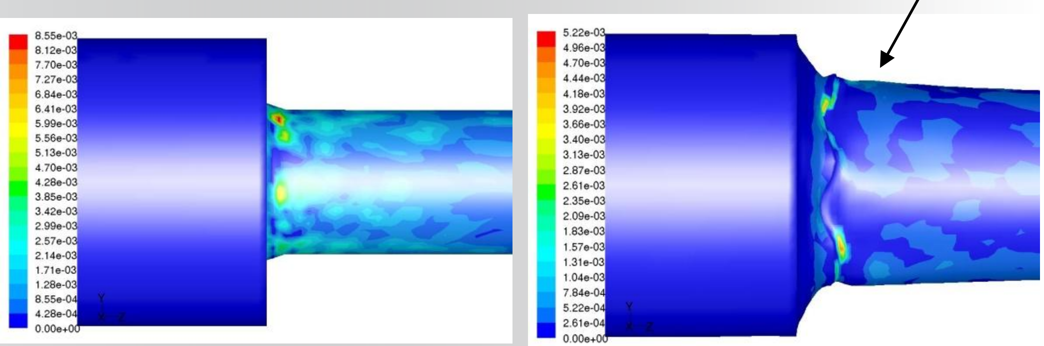
– Erosion impact is calculated as a function of:
  - Angle of impingement
  - Impact Velocity
  - Particle diameter
  - Particle mass
  - Collision frequency
  - Material type

– Wide Variety of Erosion Models
  – Mclaury et. Al Erosion Model
  – Salama & Venkatesh Erosion Model
  – Tulsa Erosion Model
  – DNV Erosion Model
  – Erosion Model for Dense Slurry Flows

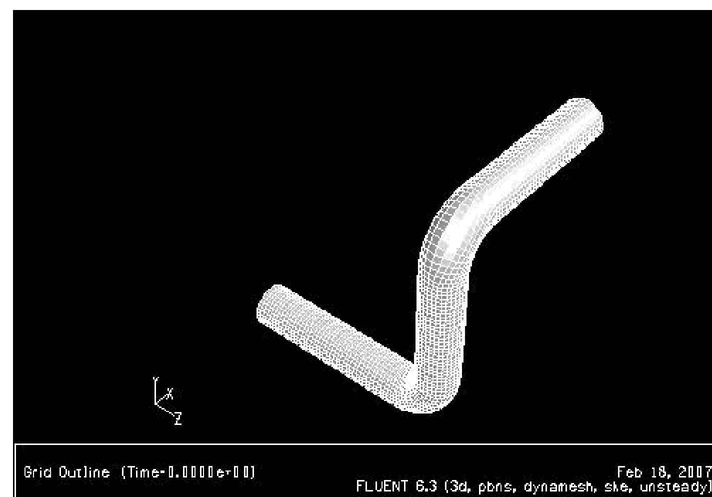
FLOW



Larger ID
After 42 hr



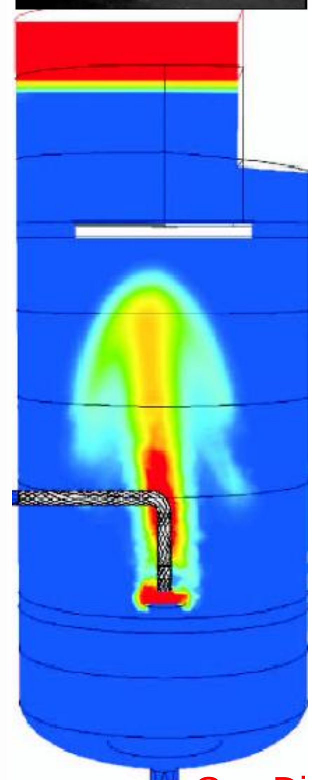
Plots of erosion contours in a 4 inch test case

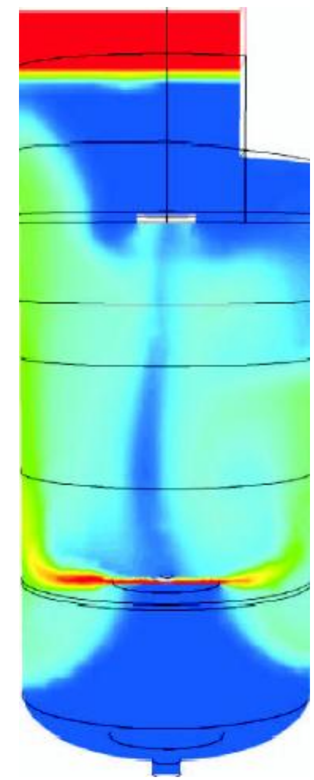


Oil and Gas Production Equipment Induced Gas Flotation (IGF) System

ANSYS CAE Solutions

- Simulate existing standard injector used in gas flotation devices to understand why it is not working
- Design new gas distributor and perform detailed studies to observe their effectiveness
- Account of multiphase flow and its behavior in different part of the IGF
- New injector and baffling system created well distributed gas bubbles and eliminated undesired recirculation zones

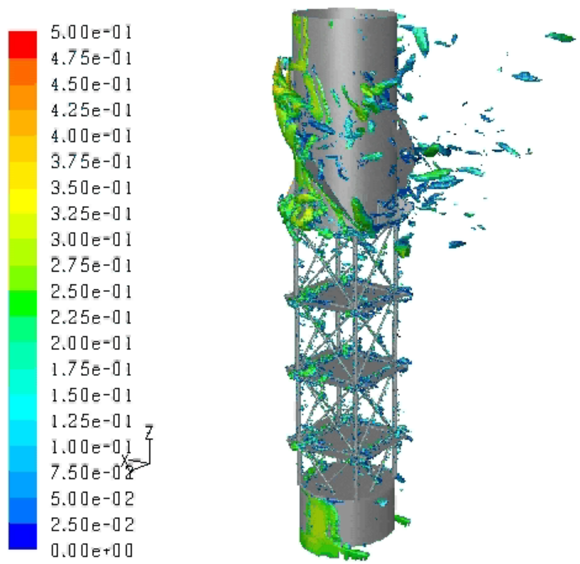




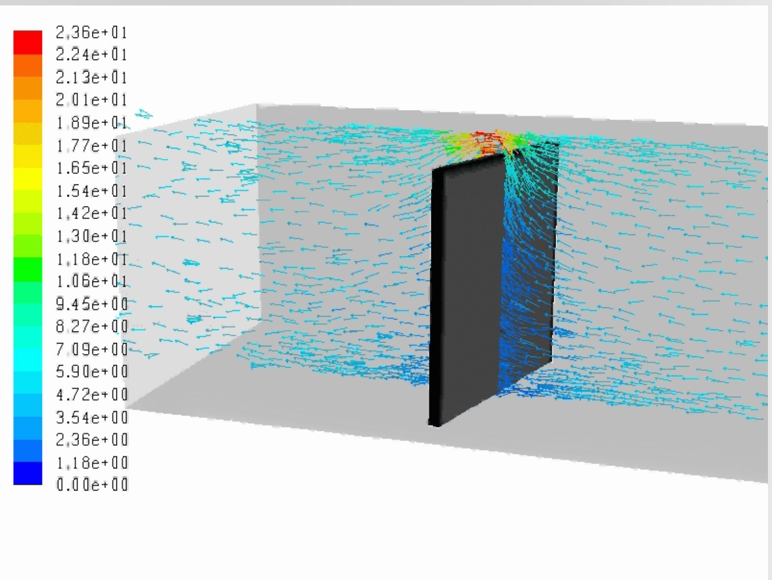
Gas Distributor Optimization

Fluid-Structure Interactions

- **Fluid Structure Interactions (FSI) available in ANSYS Workbench**
- **Iterate seamlessly between ANSYS CFD and ANSYS Structural**
  - One way
    - Pressure loadings
    - Thermal Stressing
  - Two way, dynamic motion of structures



Vortex Induced Vibration
Courtesy of Technip USA

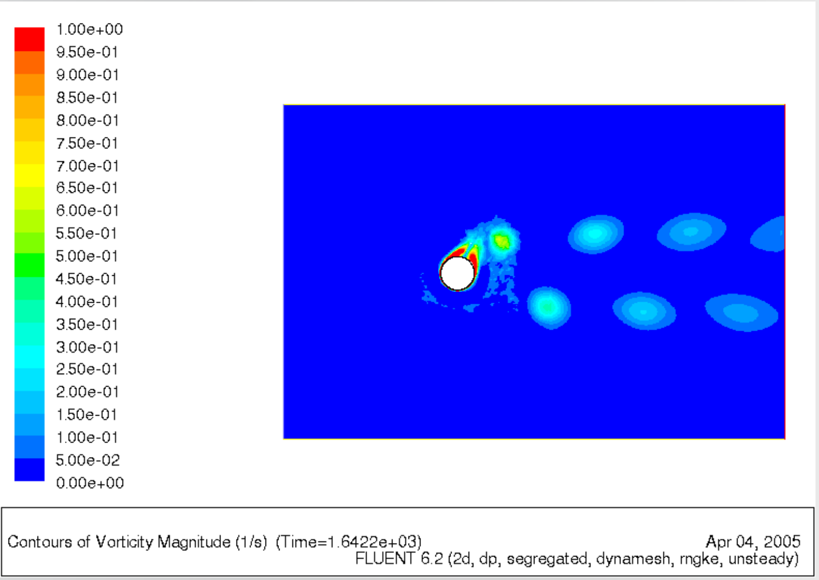


Velocity vectors through a channel with flexible flap

- **Examples**
  - Impeller Deformations
  - Flutter
  - Vortex induced vibrations (VIV)
  - Sloshing of tanks
  - Ship Motion, etc.

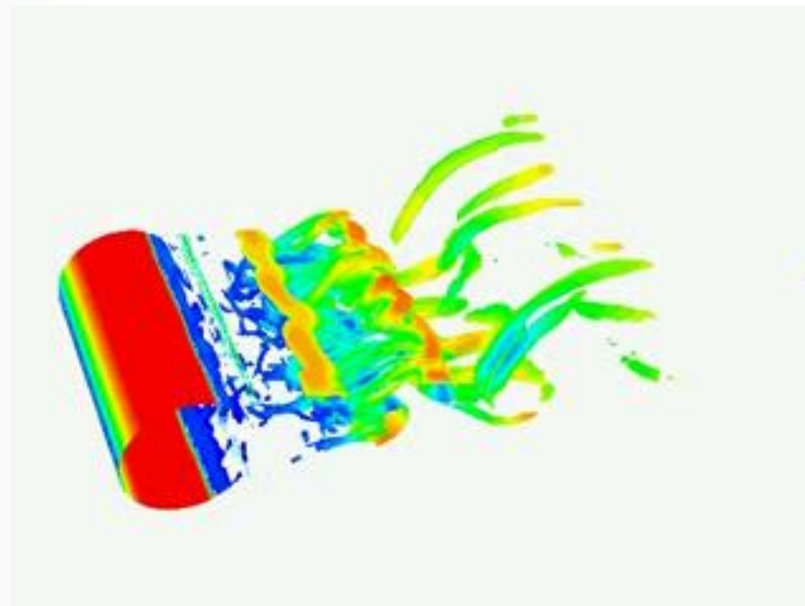
VIV/VIM

ANSYS

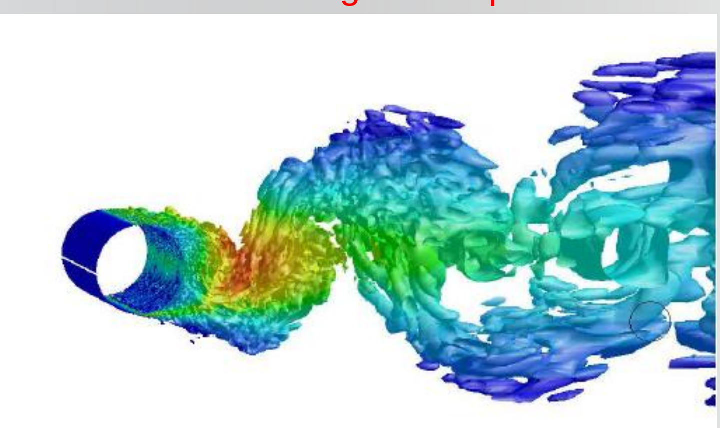


RANS Low Re Results: 2-DOF

Vortex shedding causes motion of risers, which can cause fatigue or even collision



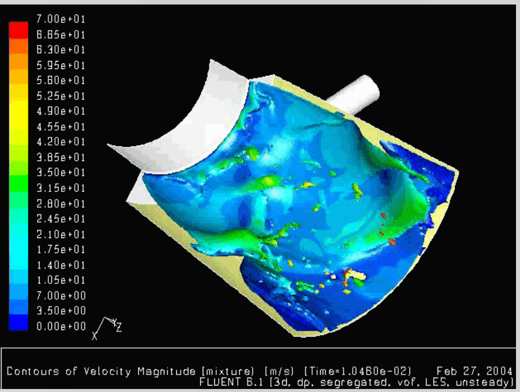
Results from LES and DNS simulations shows good agreement in coefficient of lift, drag, Strouhal number and lift fluctuation against experiments.

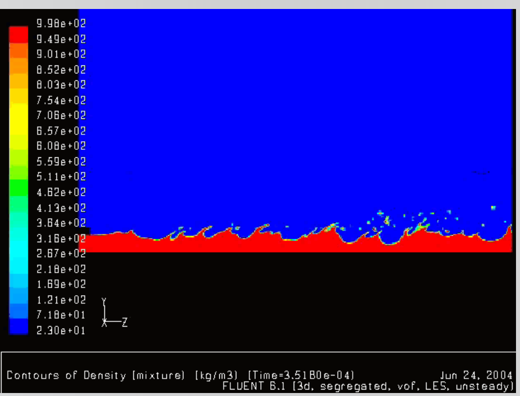


Re = 2 Million Stationary Cylinder

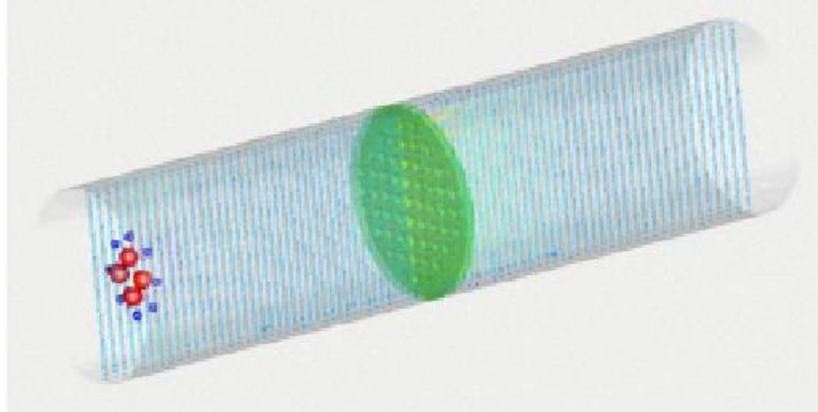
	C_{pb}	C_d	C_l'	St
DES	0.51	0.388	0.07	0.37
Experiment	0.47-0.6	0.19-0.55	0.035-0.18	0.19-0.50

Questions?



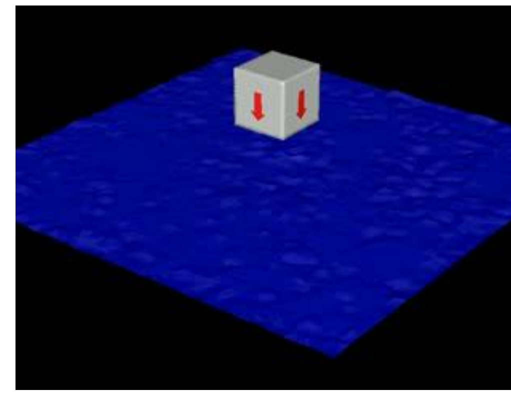


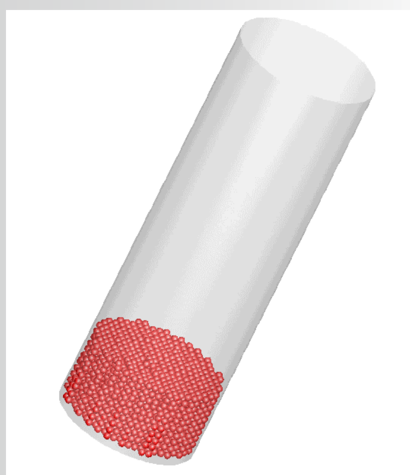
Atomization of a liquid film



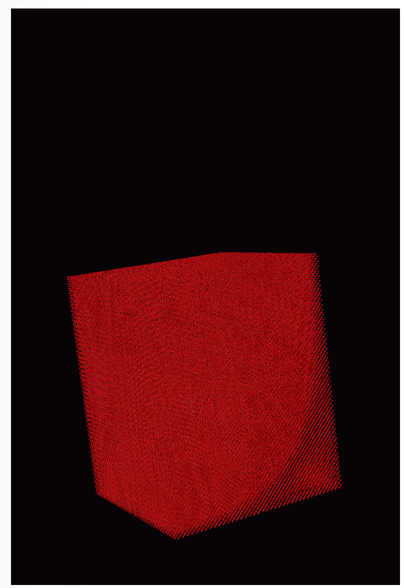
Filter

6DOF

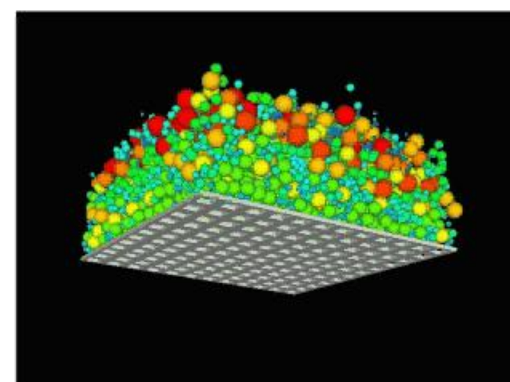




Fluidized Bed



Bubble Rise in Slurry



Vibrating screen