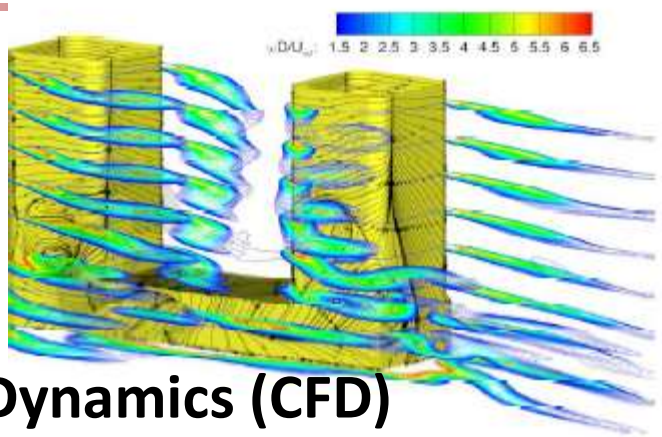
전산유체역학을 이용한
선박 및 해양플랜트 유동해석

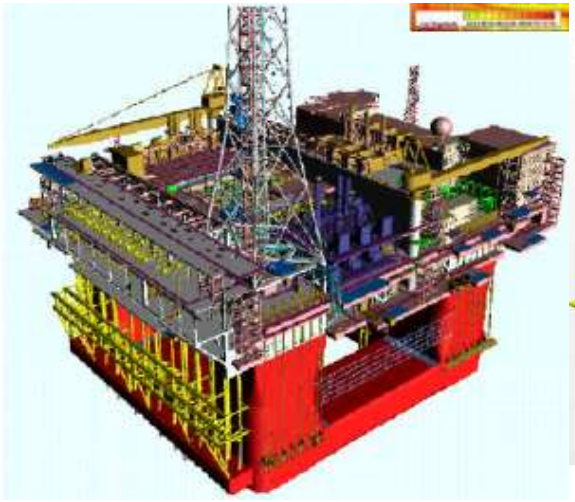
창원대학교
조선해양공학과
구본국

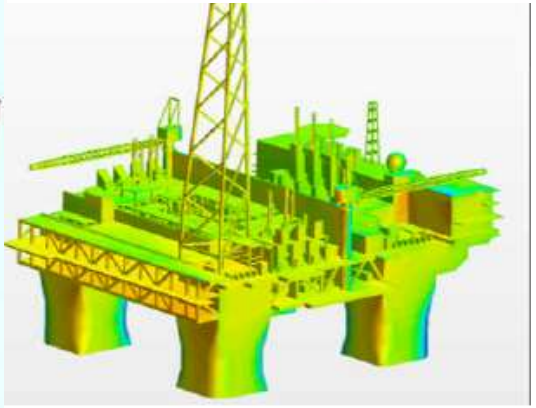
Outline

1. Part I (Self-introduction)
    I. Academic experience
2. Part II (CFD)
    I. Computational Fluid Dynamics (CFD)
3. Part III (Simulations)
    I. Wave breaking
    II. Ship flows
    III. Research in SHI
4. Part II
    I. Q & A









Part I
Self-introduction

Self-introduction

1. Education and work experience:

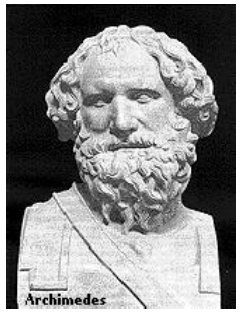
- B.S : Pusan National University, Naval Architecture and Ocean Engineering
- M.S : Pusan National University, Naval Architecture and Ocean Engineering
  (Turbulent drag reduction on ships)
- M.S : UCLA, Mechanical Engineering, RA/TA
  (Turbulent flow analysis)
- Ph.D: The University of Iowa, Mechanical Engineering, RA/TA
  (Wave breaking and surface-interface body flow simulation)
- Samsung Heavy Industries: Offshore Tech. Research Institute
  (Research on hydrodynamic analysis for offshore structures)

2. Research Interests:

- Multi-phase flow (wave breaking)
- Surface-piercing blunt body flow
- Wave run-up and air gap
- Ship maneuvering simulation
- Wave load analysis of offshore structures
- Wave impact on offshore structures
  (slamming, green water, etc)
- Turbulent modeling (LES, DES, RANS)

Map

Faces of Fluid Mechanics



Archimedes
(C. 287-212 BC)



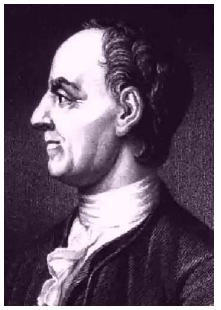
Newton
(1642-1727)



Leibniz
(1646-1716)



Bernoulli
(1667-1748)



Euler
(1707-1783)



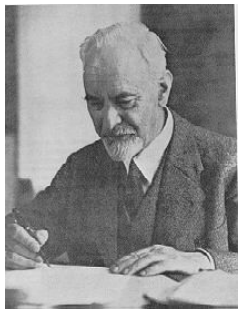
Navier
(1785-1836)



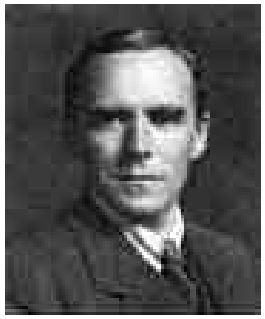
Stokes
(1819-1903)



Reynolds
(1842-1912)



Prandtl
(1875-1953)



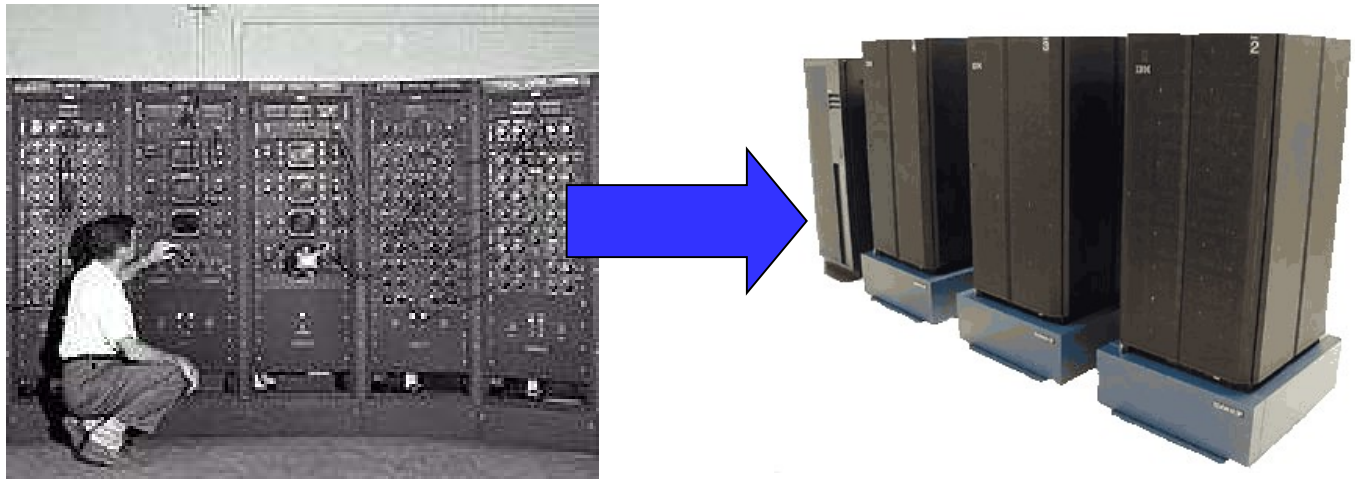
Taylor
(1886-1975)



Kolmogorov
(1903-1987)

What is CFD?

- Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)
- Historically only Analytical Fluid Dynamics (AFD) and Experimental Fluid Dynamics (EFD).
- CFD made possible by the advent of digital computer and advancing with improvements of computer resources



Why use CFD?

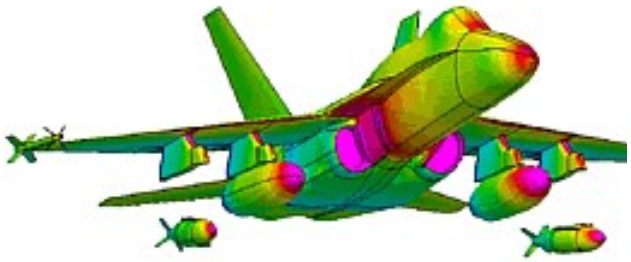


- Analysis and Design
  1. Simulation-based design instead of “build & test”
     - More cost effective and more rapid than EFD
     - CFD provides high-fidelity database for diagnosing flow field
  2. Simulation of physical fluid phenomena that are difficult for experiments
     - Full scale simulations (e.g., ships and airplanes)
     - Environmental effects (wind, weather, etc.)
     - Hazards (e.g., explosions, radiation, pollution)
     - Physics (e.g., planetary boundary layer, stellar evolution)
- Knowledge and exploration of flow physics

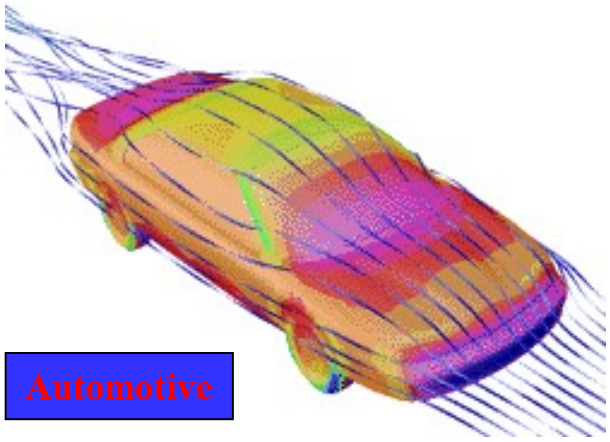
Where is CFD used?

- Where is CFD used?
  - ***Aerospace***
  - ***Automotive***
  - ***Biomedical***
  - Chemical Processing
  - HVAC
  - Hydraulics
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports

Aerospace

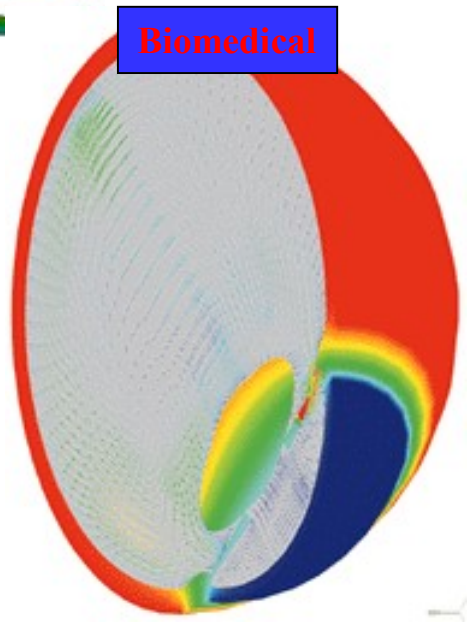


F18 Store Separation



Automotive

Biomedical

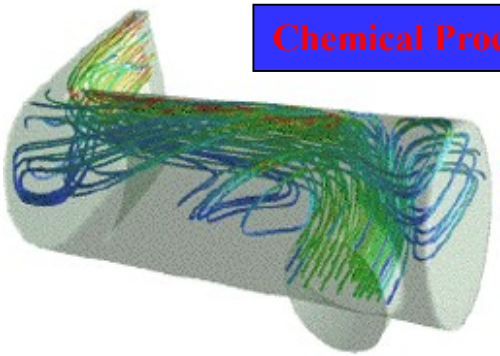


Temperature and natural convection currents in the eye following laser heating.

Where is CFD used?

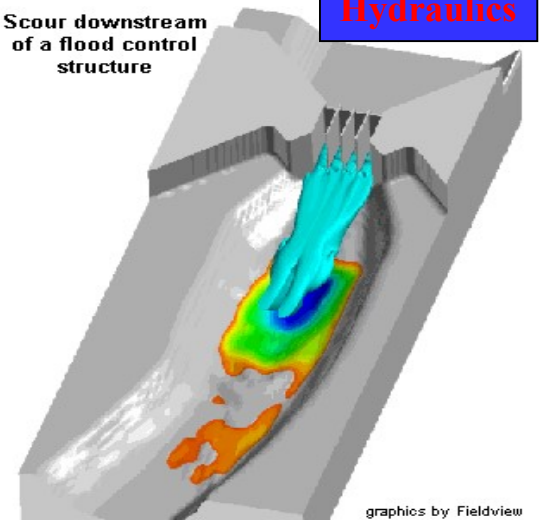
- Where is CFD used?
  - Aerospacee
  - Automotive
  - Biomedical
  - ***Chemical Processing***
  - ***HVAC***
  - ***Hydraulics***
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports

Chemical Processing

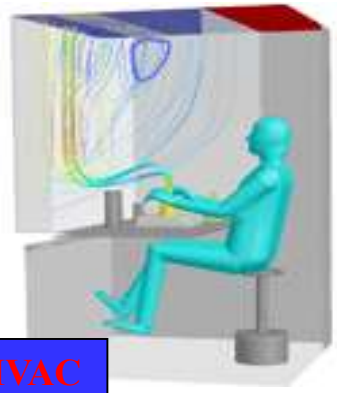


Polymerization reactor vessel - prediction of flow separation and residence time effects.

Hydraulics



HVAC

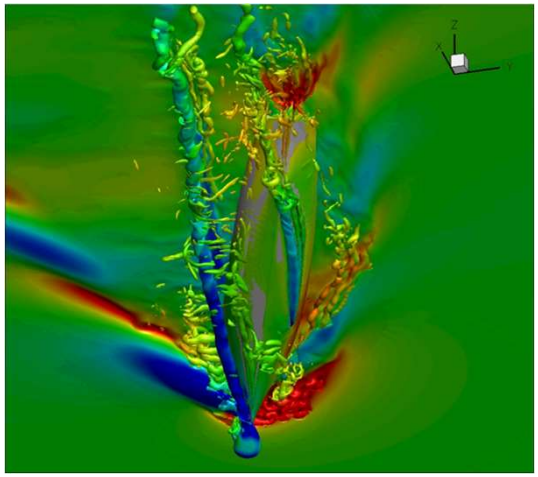


Streamlines for workstation ventilation

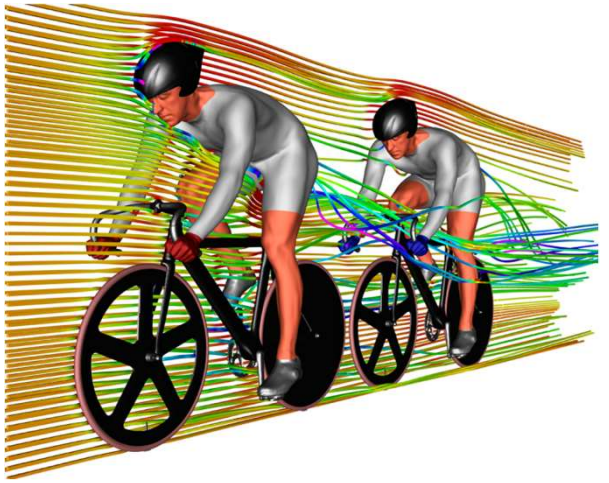
Where is CFD used?

- Where is CFD used?
  - Aerospace
  - Automotive
  - Biomedical
  - Chemical Processing
  - HVAC
  - Hydraulics
  - ***Marine***
  - ***Oil & Gas***
  - ***Power Generation***
  - ***Sports***

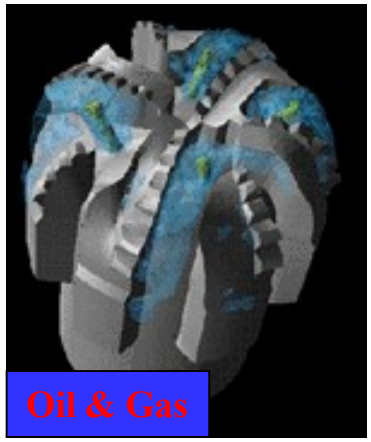
Marine



Sports



http://gallery.ensight.com/keyword/sport/1/701168838_KVnHn#!i=701168838&k=KVnHn



Oil & Gas

Flow of lubricating mud over drill bit



Power Generation

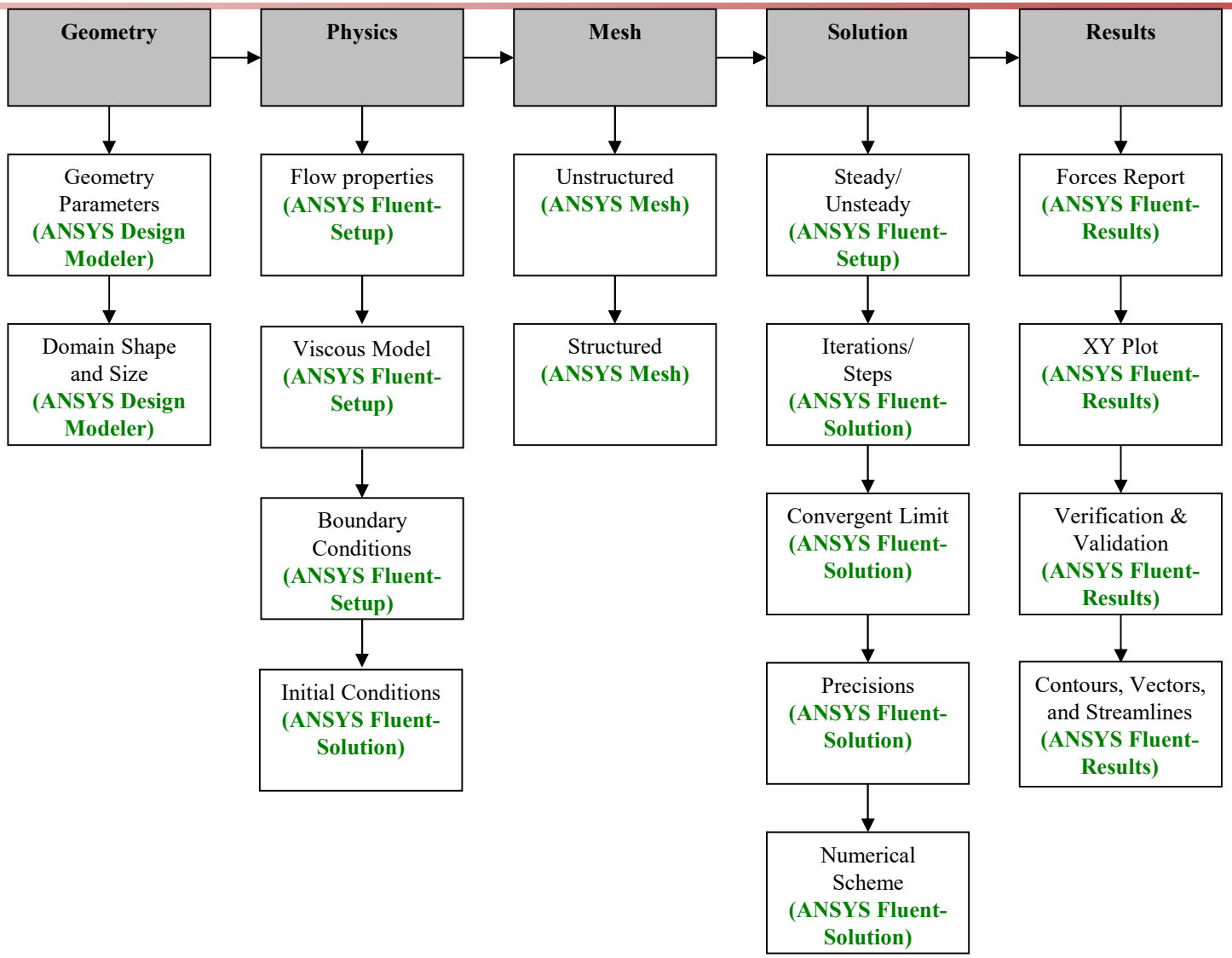
Flow around cooling towers

CFD process

- Purposes of CFD codes will be different for different applications: investigation of bubble-fluid interactions for bubbly flows, study of wave induced massively separated flows for free-surface, etc.
- Depend on the specific purpose and flow conditions of the problem, different CFD codes can be chosen for different applications (aerospace, marines, combustion, multi-phase flows, etc.)
- Once purposes and CFD codes chosen, "CFD process" is the steps to set up the IBVP problem and run the code:

1. Geometry
2. Physics (Setup)
3. Mesh
4. Solve
5. Results

CFD Process



Green regions indicate ANSYS modules

Modeling

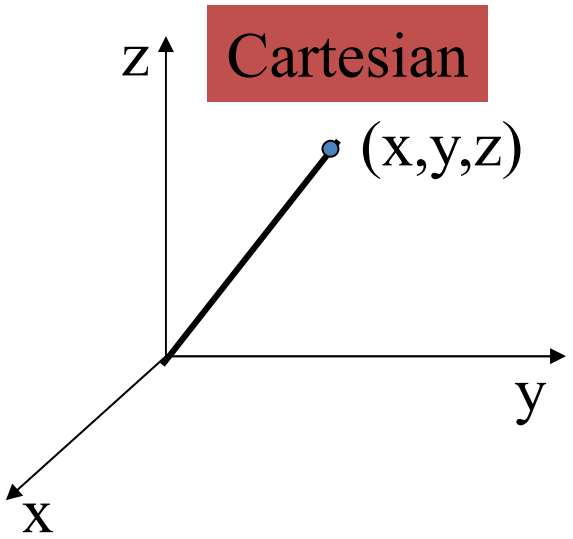
- Modeling is the mathematical physics problem formulation in terms of a continuous initial boundary value problem (IBVP)
- IBVP is in the form of Partial Differential Equations (PDEs) with appropriate boundary conditions and initial conditions.
- Modeling includes:

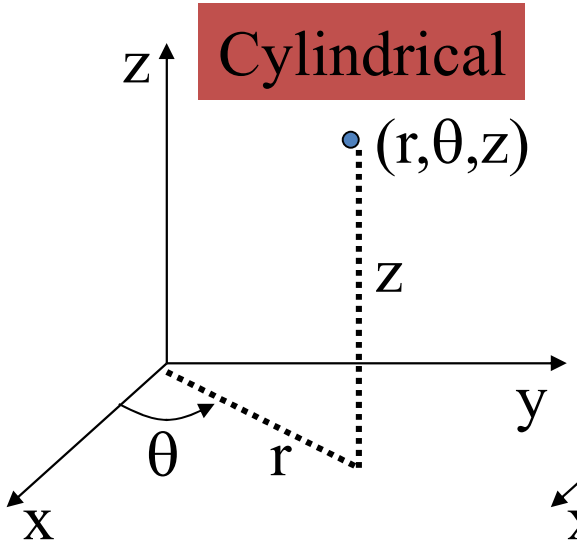
1. Geometry and domain
2. Coordinates
3. Governing equations
4. Flow conditions
5. Initial and boundary conditions
6. Selection of models for different applications

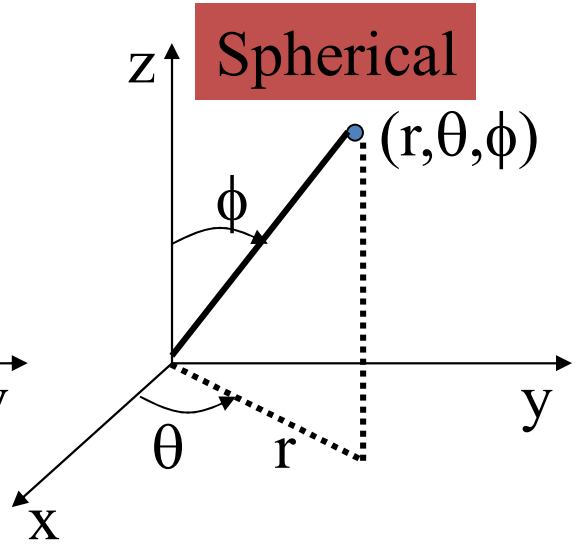
Modeling (geometry and domain)

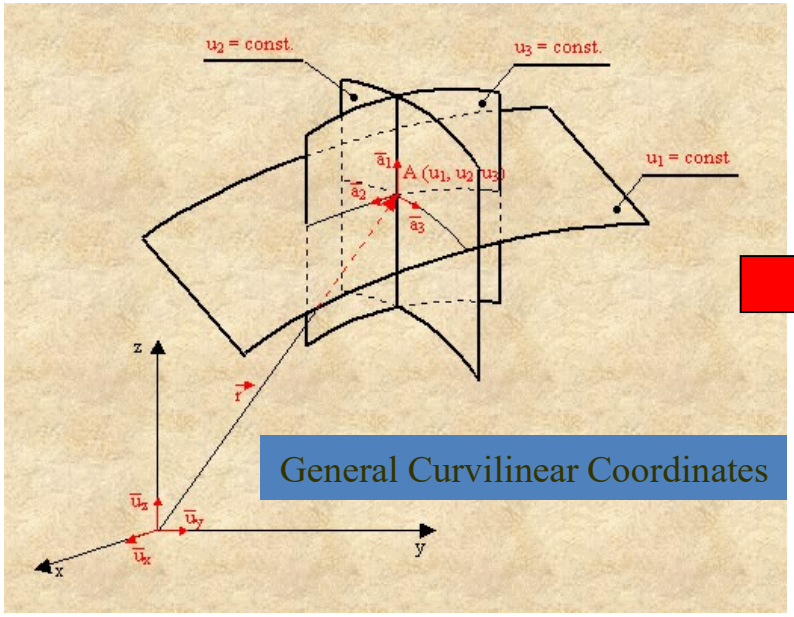
- Simple geometries can be easily created by few geometric parameters (e.g. circular pipe)
- Complex geometries must be created by the partial differential equations or importing the database of the geometry(e.g. airfoil) into commercial software
- Domain: size and shape
- Typical approaches
  - Geometry approximation
  - CAD/CAE integration: use of industry standards such as Parasolid, ACIS, STEP, or IGES, etc.
  - The three coordinates: Cartesian system (x,y,z), cylindrical system (r, θ, z), and spherical system(r, θ, Φ) should be appropriately chosen for a better resolution of the geometry (e.g. cylindrical for circular pipe).

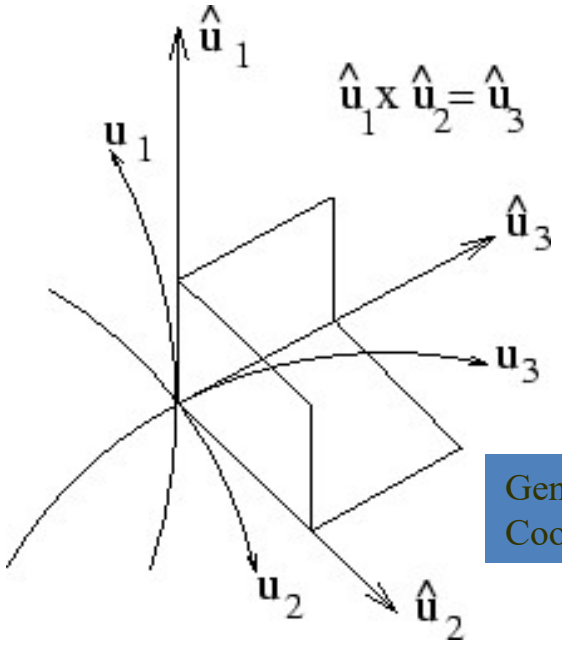
Modeling (coordinates)











Modeling (governing equations)

- Navier-Stokes equations (3D in Cartesian coordinates)

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial \hat{p}}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right]$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial \hat{p}}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial \hat{p}}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]$$

Local acceleration

Convection

Piezometric pressure gradient

Viscous terms

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

Continuity equation

$$p=\rho RT$$

Equation of state

$$R\frac{D^2R}{Dt^2}+\frac{3}{2}\left(\frac{DR}{Dt}\right)^2=\frac{p_v-p}{\rho_L}$$

Rayleigh Equation

Modeling (flow conditions)



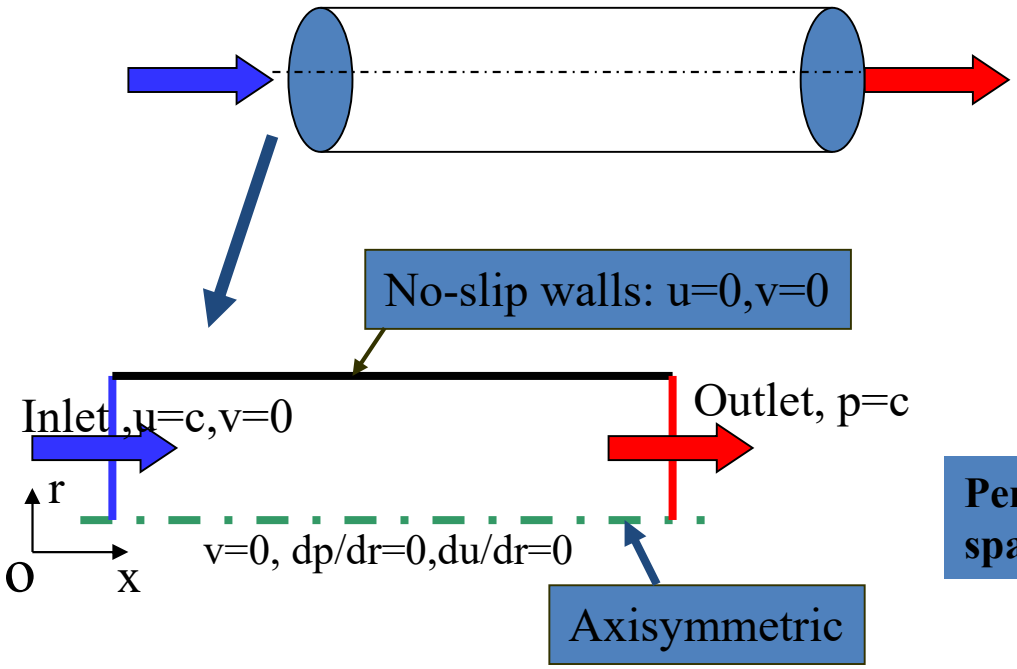
- Based on the physics of the fluids phenomena, CFD can be distinguished into different categories using different criteria
  - Viscous vs. inviscid (Re: Reynolds No.)
  - External flow or internal flow (wall bounded or not)
  - Turbulent vs. laminar (Re)
  - Incompressible vs. compressible (Ma: Mach No.)
  - Single- vs. multi-phase (Ca: Cavitation No.)
  - Thermal/density effects (Pr: Prandtl No., γ, Gr: Grashof No., Ec)
  - Free-surface flow (Fr) and surface tension (We)
  - Chemical reactions and combustion (Pe, Da)
  - etc…

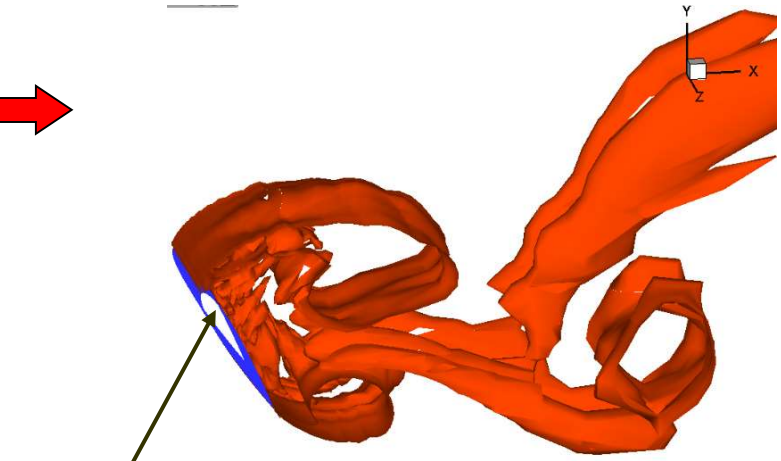
Modeling (initial conditions)

- Initial conditions (ICS, steady/unsteady flows)
  - ICs should not affect final results and only affect convergence path, i.e. number of iterations (steady) or time steps (unsteady) need to reach converged solutions.
  - More reasonable guess can speed up the convergence
  - For complicated unsteady flow problems, CFD codes are usually run in the steady mode for a few iterations for getting a better initial conditions

Modeling(boundary conditions)

- Boundary conditions: No-slip or slip-free on walls, periodic, inlet (velocity inlet, mass flow rate, constant pressure, etc.), outlet (constant pressure, velocity convective, numerical beach, zero-gradient), and non-reflecting (for compressible flows, such as acoustics), etc.





Periodic boundary condition in spanwise direction of an airfoil

Modeling (selection of models)

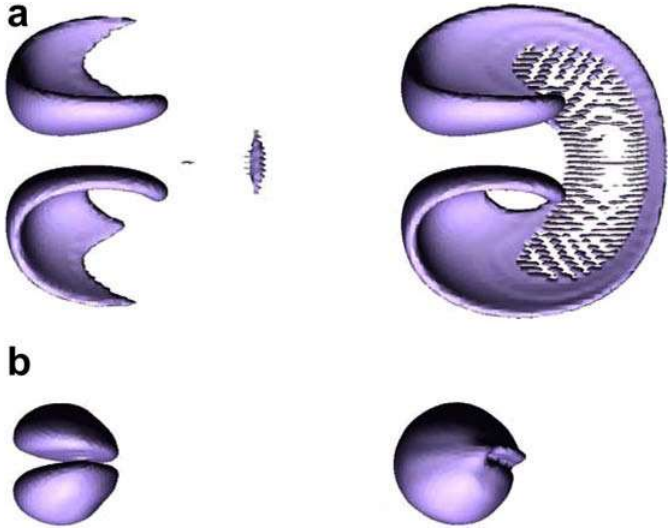


- CFD codes typically designed for solving certain fluid phenomenon by applying different models
  - Viscous vs. inviscid (Re)
  - Turbulent vs. laminar (Re, Turbulent models)
  - Incompressible vs. compressible (Ma, equation of state)
  - Single- vs. multi-phase (Ca, cavitation model, two-fluid model)
  - Thermal/density effects and energy equation (Pr, γ, Gr, Ec, conservation of energy)
  - Free-surface flow (Fr, level-set & surface tracking model) and surface tension (We, bubble dynamic model)
  - Chemical reactions and combustion (Chemical reaction model)
  - etc…

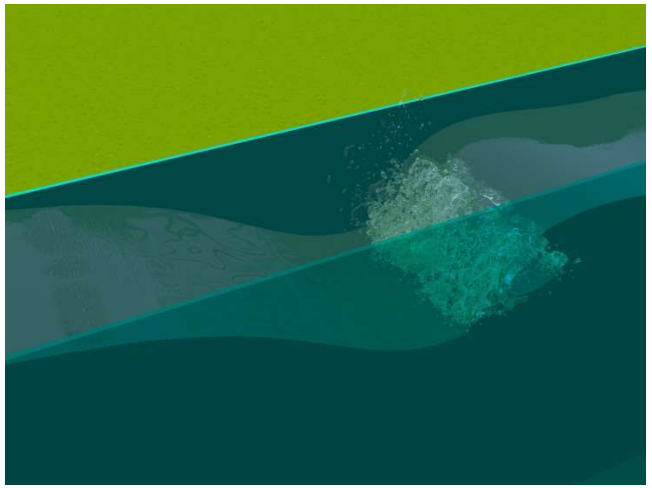
Modeling (Turbulence and free surface models)

- Turbulent flows at high Re usually involve both large and small scale vortical structures and very thin turbulent boundary layer (BL) near the wall
- **Turbulent models**:
  - DNS: most accurately solve NS equations, but too expensive for turbulent flows
  - RANS: predict mean flow structures, efficient inside BL but excessive diffusion in the separated region.
  - LES: accurate in separation region and unaffordable for resolving BL
  - DES: RANS inside BL, LES in separated regions.
- **Free-surface models**:
  - Surface-tracking method: mesh moving to capture free surface, limited to small and medium wave slopes
  - Single/two phase level-set method: mesh fixed and level-set function used to capture the gas/liquid interface, capable of studying steep or breaking waves.

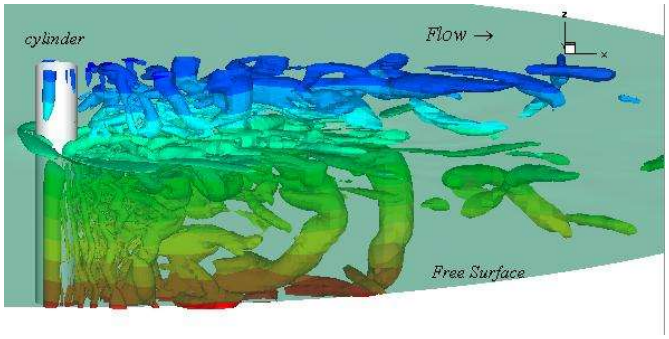
Modeling (examples)



Deformation of a sphere.(a)maximum stretching;
(b) recovered shape. Left: LS; right: VOF.



Wave breaking in bump flow simulation



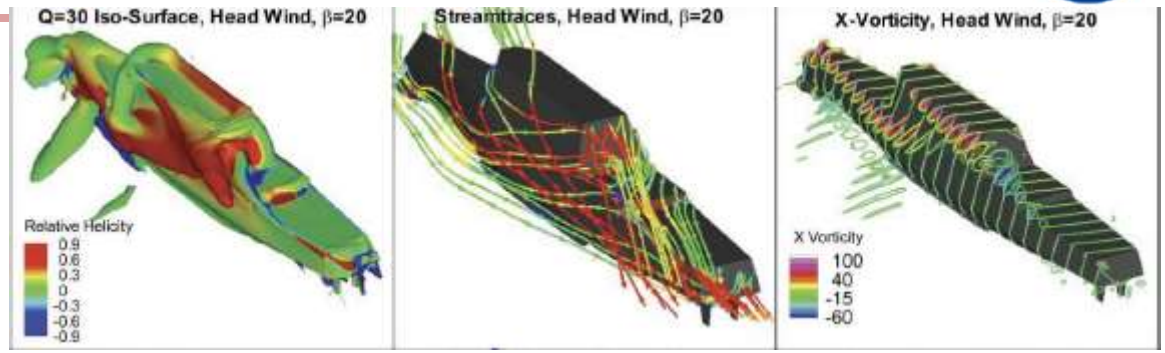
Two-phase flow past a surface-piercing cylinder
showing vortical structures colored by pressure

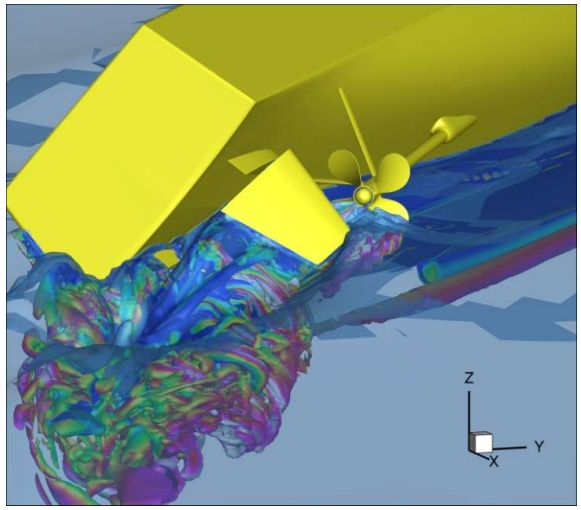


Wedge flow simulation

Modeling (examples, cont'd)

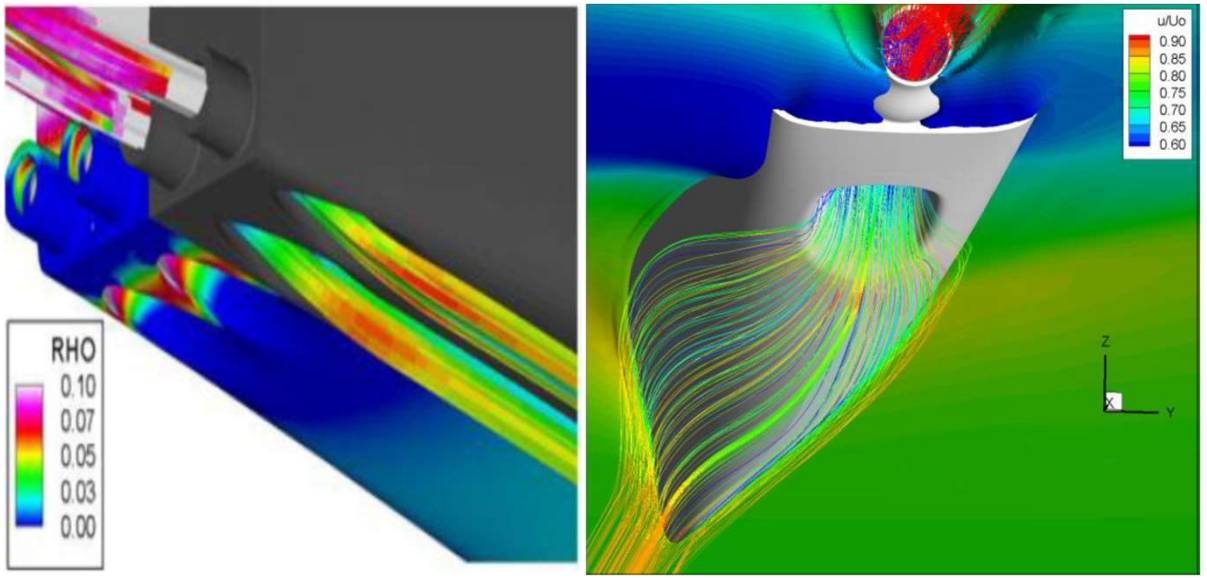
Air flow for ONR Tumblehome in PMM maneuvers



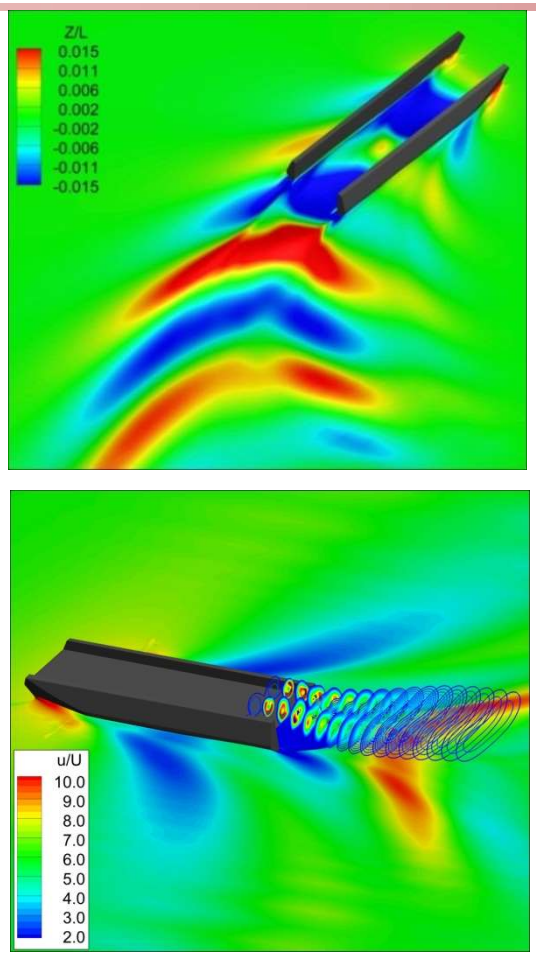


Broaching of ONR Tumblehome with rotating propellers

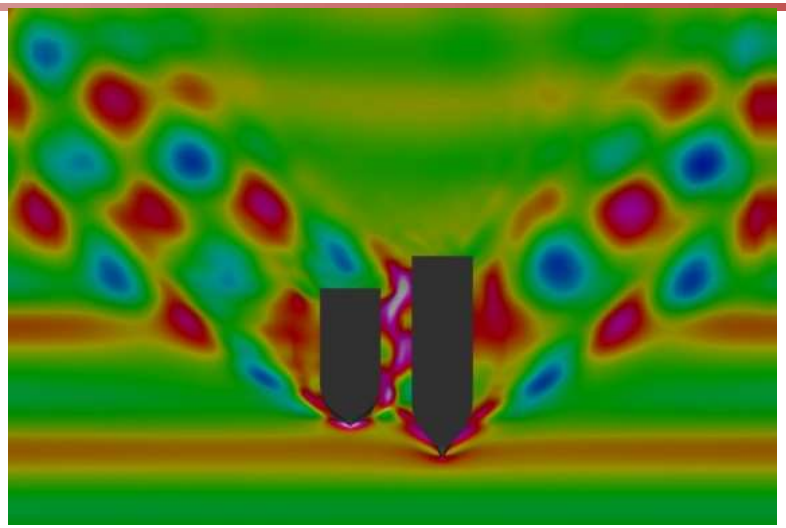
Waterjet flow modeling for JHSS and Delft catamaran

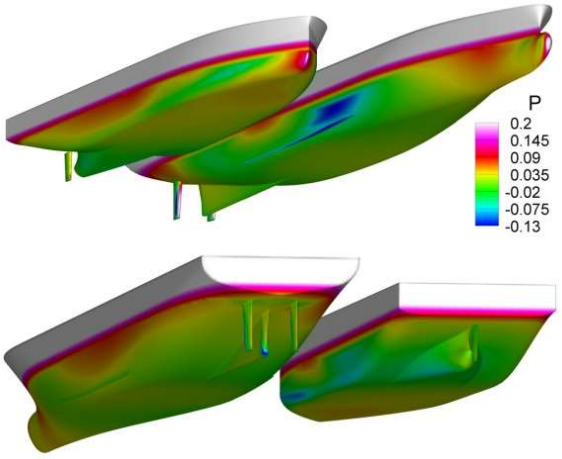


Modeling (examples, cont'd)



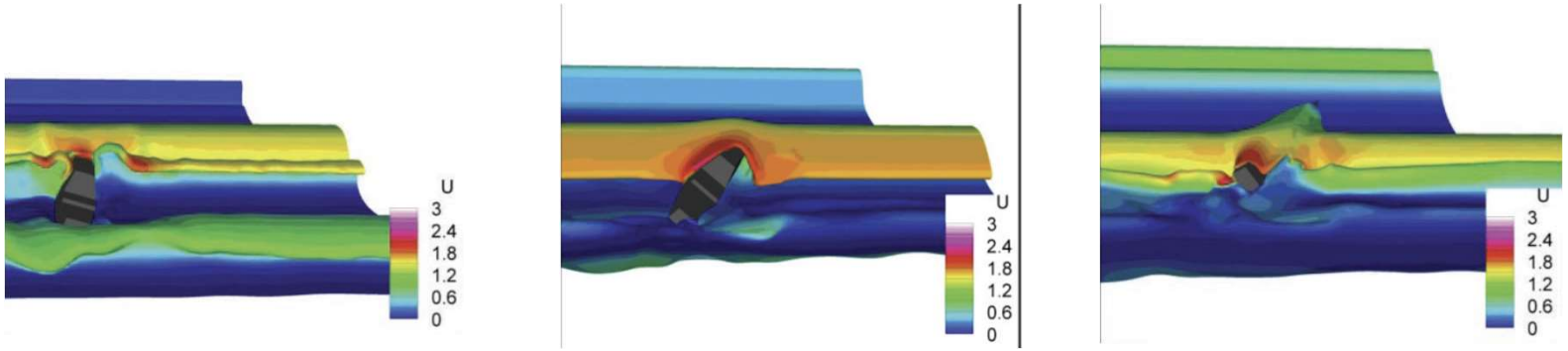
T-Craft (SES/ACV) turning circle in calm water with water jet propulsion (top) and straight ahead with air-fan propulsion (bottom)



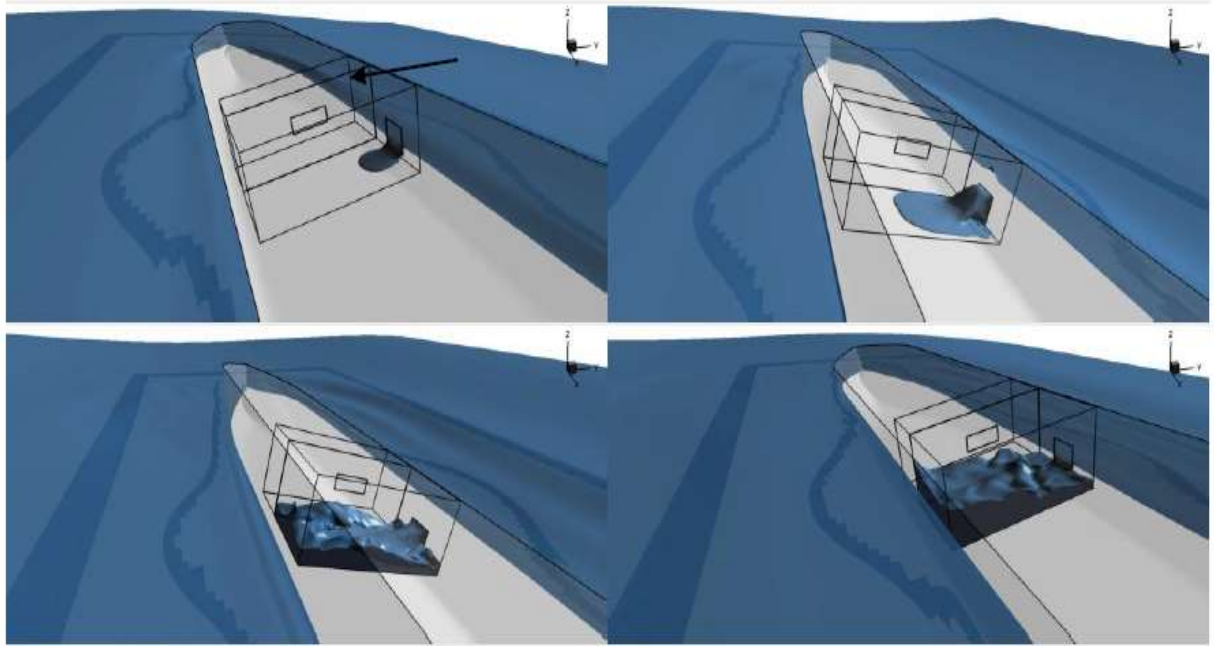


Regular head wave simulation for side by side ship-ship interactions

Modeling (examples, cont'd)

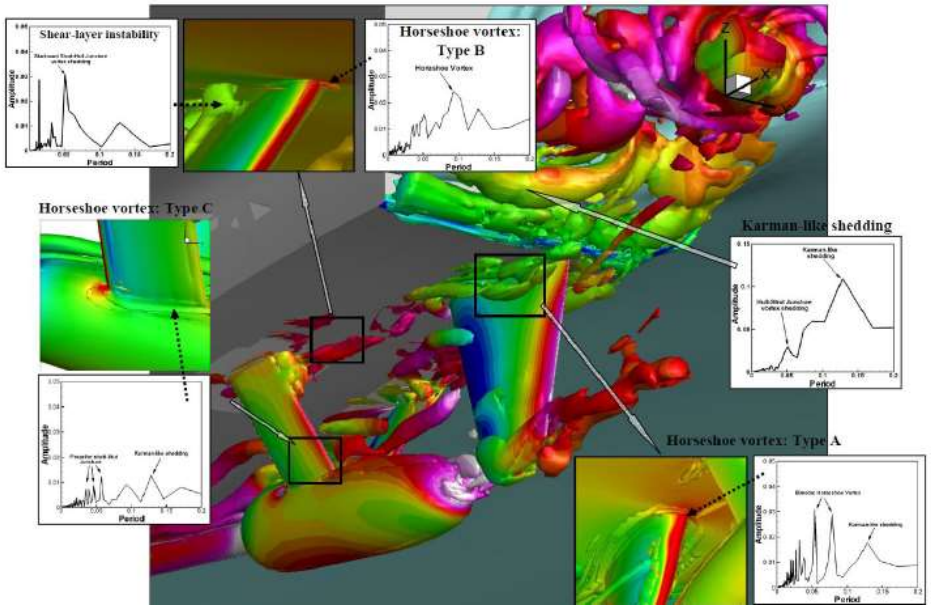


Ship in three-sisters rogue (freak) waves



Damaged stability for SSRC cruiser with two-room compartment in beam waves

Vortical Structures and Instability Analysis



DTMB 5415 at β=20° DES Computation

Re=4.85×10^6,Fr=0.28

Isosurface of Q=300 colored using piezometric pressure

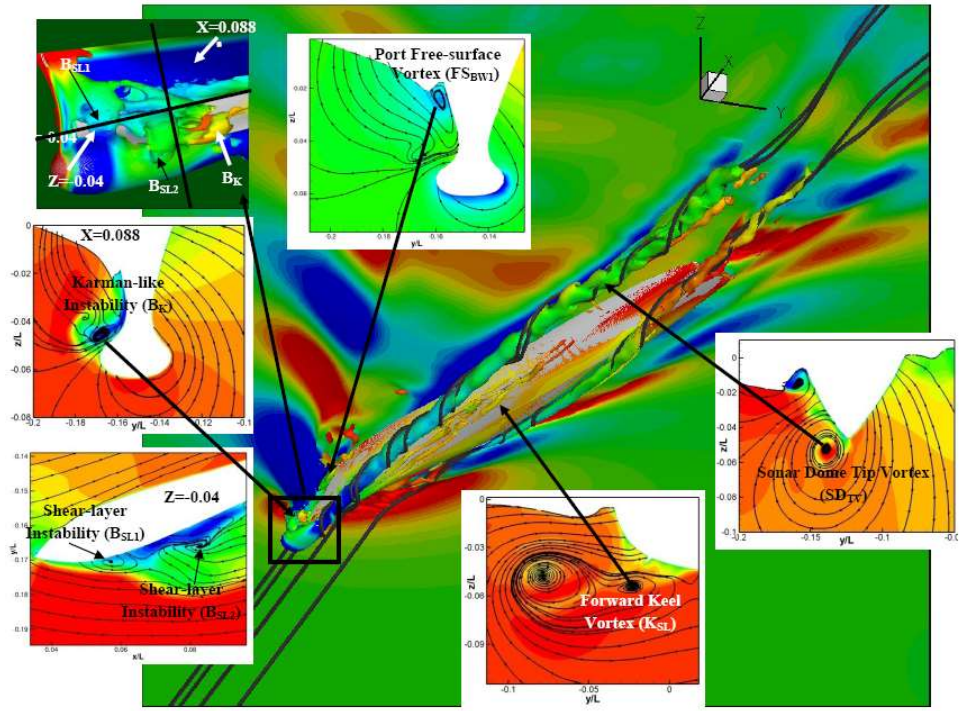
- The sonar dome (SD_{TV}) and bilge keel (BK_{TV}) vortices exhibits helical instability breakdown.
- Shear-layer instabilities: port bow (B_{SL1}, B_{SL2}) and fore-body keel (K_{SL}).
- Karman-like instabilities on port side bow (B_K) .
- Wave breaking vortices on port (FS_{BW1}) and starboard (FS_{BW2}). Latter exhibits horse shoe type instability.

Fully appended Athena DES Computation

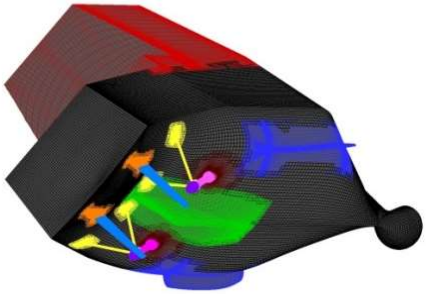
Re=2.9×10^8, Fr=0.25

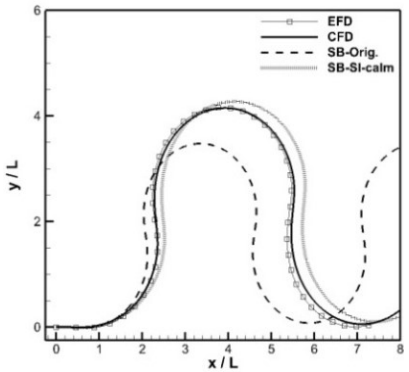
Isosurface of Q=300 colored using piezometric pressure

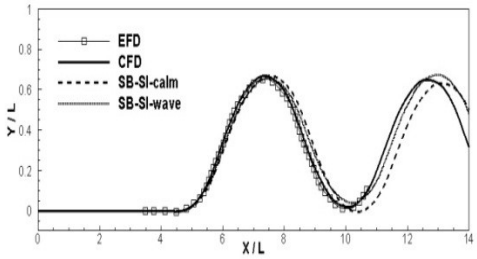
- Karman-like shedding from Transom Corner
- Horse-shoe vortices from hull-rudder (Case A) and strut-hull (Case B) junction flow.
- Shear layer instability at hull-strut intersection



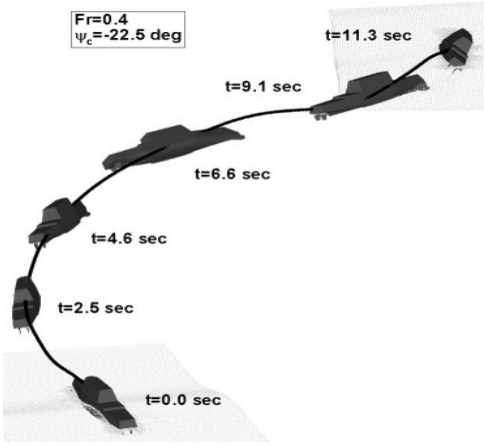
Modeling (examples, cont'd)

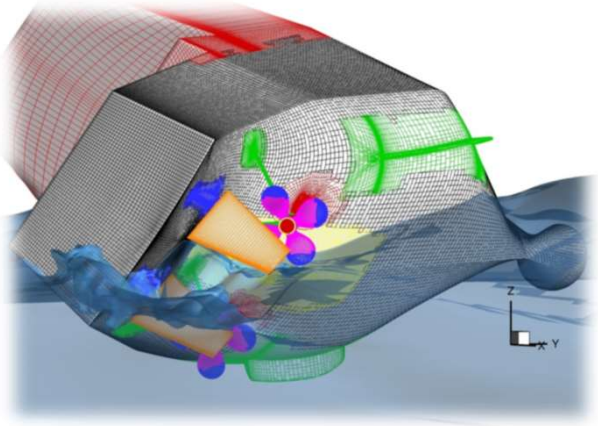






CFD simulations to improve system identification (SI) technique





Broaching simulation of free running ONR Tumblehome

Numerical methods

- The continuous Initial Boundary Value Problems (IBVPs) are discretized into algebraic equations using numerical methods. Assemble the system of algebraic equations and solve the system to get approximate solutions
- Numerical methods include:

1. Discretization methods
2. Solvers and numerical parameters
3. Grid generation and transformation
4. High Performance Computation (HPC) and post-processing

Discretization methods

- Finite difference methods (straightforward to apply, usually for regular grid) and finite volumes and finite element methods (usually for irregular meshes)
- Each type of methods above yields the same solution if the grid is fine enough. However, some methods are more suitable to some cases than others
- Finite difference methods for spatial derivatives with different order of accuracies can be derived using Taylor expansions, such as 2nd order upwind scheme, central differences schemes, etc.
- Higher order numerical methods usually predict higher order of accuracy for CFD, but more likely unstable due to less numerical dissipation
- Temporal derivatives can be integrated either by the explicit method (Euler, Runge-Kutta, etc.) or implicit method (e.g. Beam-Warming method)

Discretization methods (Cont'd)

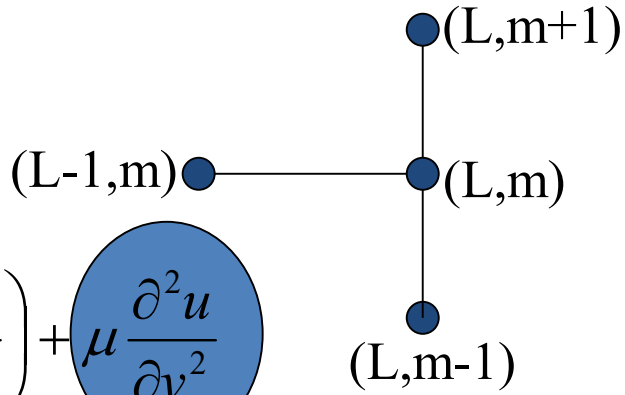
- Explicit methods can be easily applied but yield conditionally stable Finite Different Equations (FDEs), which are restricted by the time step; Implicit methods are unconditionally stable, but need efforts on efficiency.
- Usually, higher-order temporal discretization is used when the spatial discretization is also of higher order.
- Stability: A discretization method is said to be stable if it does not magnify the errors that appear in the course of numerical solution process.
- Pre-conditioning method is used when the matrix of the linear algebraic system is ill-posed, such as multi-phase flows, flows with a broad range of Mach numbers, etc.
- Selection of discretization methods should consider efficiency, accuracy and special requirements, such as shock wave tracking.

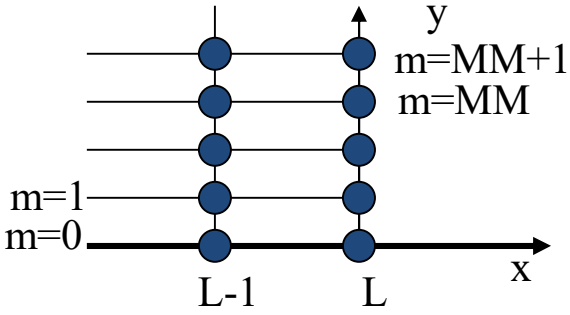
Discretization methods (example)

- 2D incompressible laminar flow boundary layer

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial}{\partial x}\left(\frac{p}{e}\right)+\mu\frac{\partial^2 u}{\partial y^2}$$





$$u\frac{\partial u}{\partial x}=\frac{u_m^l}{\Delta x}\left[u_m^l-u_m^{l-1}\right]$$

$$v\frac{\partial u}{\partial y}=\frac{v_m^l}{\Delta y}\left[u_{m+1}^l-u_m^l\right]$$

$$=\frac{v_m^l}{\Delta y}\left[u_m^l-u_{m-1}^l\right]$$

$$\mu\frac{\partial^2 u}{\partial y^2}=\frac{\mu}{\Delta y^2}\left[u_{m+1}^l-2u_m^l+u_{m-1}^l\right]$$

FD $\mathrm{Sign}(v_m^l)<0$

BD $\mathrm{Sign}(v_m^l)>0$

2nd order central difference
i.e., theoretical order of accuracy
$P_{kest}= 2$.

1st order upwind scheme, i.e., theoretical order of accuracy $P_{kest}= 1$

Discretization methods (example)

$$\underbrace{\left[\frac{u_m^l}{\Delta x}+v_m^l\frac{1}{\Delta y}FD-\frac{2\mu}{\Delta y^2}\right]}_{B_2}u_m^l+\underbrace{\left[\frac{\mu}{\Delta y^2}+\frac{v_m^l}{\Delta y}FD\right]}_{B_3}u_{m+1}^l+\underbrace{\left[\frac{\mu}{\Delta y^2}-\frac{v_m^l}{\Delta y}BD\right]}_{B_1}u_{m-1}^l$$

$$=\underbrace{\frac{u_m^l}{\Delta x}}_{B_4}u_m^{l-1}-\frac{\partial}{\partial x}(p/e)_m^l$$

$$B_1u_{m-1}^l+B_2u_m^l+B_3u_{m+1}^l=B_4u_m^{l-1}-\frac{\partial}{\partial x}\left(p/e\right)_m^l$$

$$\begin{bmatrix} B_2 & B_3 & 0 & 0 & 0 & 0 & 0 & 0\\ B_1 & B_2 & B_3 & 0 & 0 & 0 & 0 & 0\\ & & \bullet & \bullet & \bullet & \bullet & & \\ 0 & 0 & 0 & 0 & 0 & B_1 & B_2 & B_3\\ 0 & 0 & 0 & 0 & 0 & 0 & B_1 & B_2 \end{bmatrix}\times\begin{bmatrix} u_1^l\\ \bullet\\ \bullet\\ \bullet\\ u_{mm}^l \end{bmatrix}=\begin{bmatrix} B_4u_1^{l-1}-\frac{\partial}{\partial x}\left(\frac{p}{e}\right)_1^l\\ \bullet\\ \bullet\\ \bullet\\ B_4u_{mm}^{l-1}-\frac{\partial}{\partial x}\left(\frac{p}{e}\right)_{mm}^l \end{bmatrix}$$

Solve it using Thomas algorithm

To be stable, Matrix has to be Diagonally dominant.

Solvers and numerical parameters

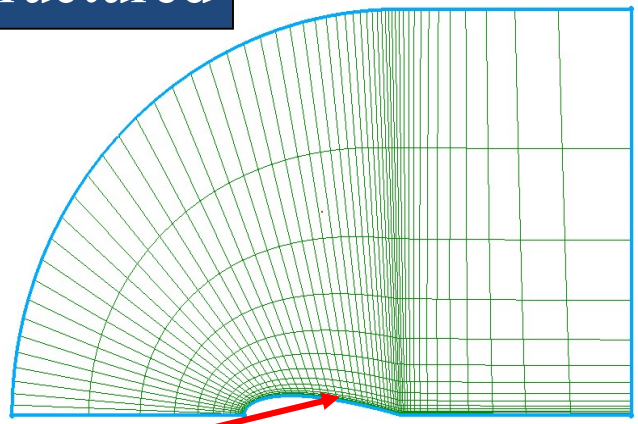


- **Solvers** include: tridiagonal, pentadiagonal solvers, PETSC solver, solution-adaptive solver, multi-grid solvers, etc.
- **Solvers** can be either direct (Cramer's rule, Gauss elimination, LU decomposition) or iterative (Jacobi method, Gauss-Seidel method, SOR method)
- **Numerical parameters** need to be specified to control the calculation.
  - Under relaxation factor, convergence limit, etc.
  - Different numerical schemes
  - Monitor residuals (change of results between iterations)
  - Number of iterations for steady flow or number of time steps for unsteady flow
  - Single/double precisions

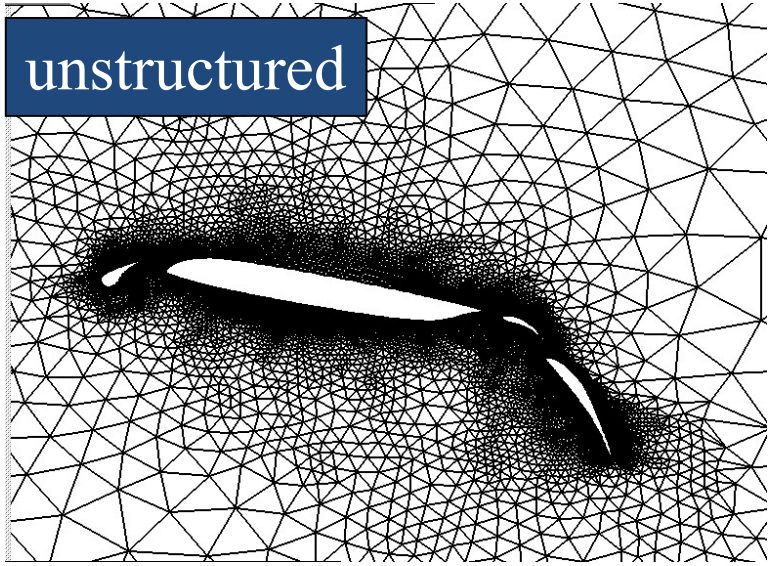
Numerical methods (grid generation)

- Grids can either be structured (hexahedral) or unstructured (tetrahedral). Depends upon type of discretization scheme and application
  - Scheme
    - ❑Finite differences: structured
    - ❑Finite volume or finite element: structured or unstructured
  - Application
    - ❑Thin boundary layers best resolved with highly-stretched structured grids
    - ❑Unstructured grids useful for complex geometries
    - ❑Unstructured grids permit automatic adaptive refinement based on the pressure gradient, or regions interested (FLUENT)

structured



unstructured



High performance computing

- CFD computations (e.g. 3D unsteady flows) are usually very expensive which requires parallel high performance supercomputers (e.g. IBM 690) with the use of multi-block technique.
- As required by the multi-block technique, CFD codes need to be developed using the Massage Passing Interface (MPI) Standard to transfer data between different blocks.
- Emphasis on improving:
  - Strong scalability, main bottleneck pressure Poisson solver for incompressible flow.
  - Weak scalability, limited by the memory requirements.

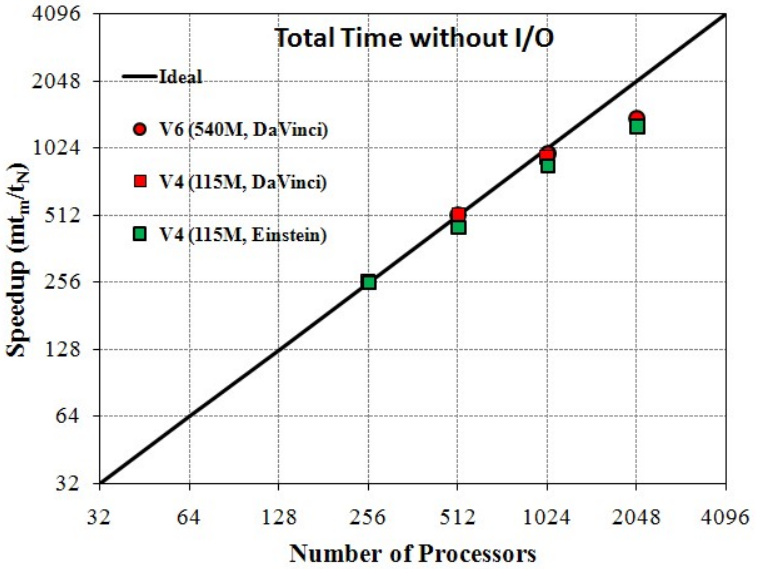


Figure: Strong scalability of total times without I/O for CFDShip-Iowa V6 and V4 on NAVO Cray XT5 (Einstein) and IBM P6 (DaVinci) are compared with ideal scaling.

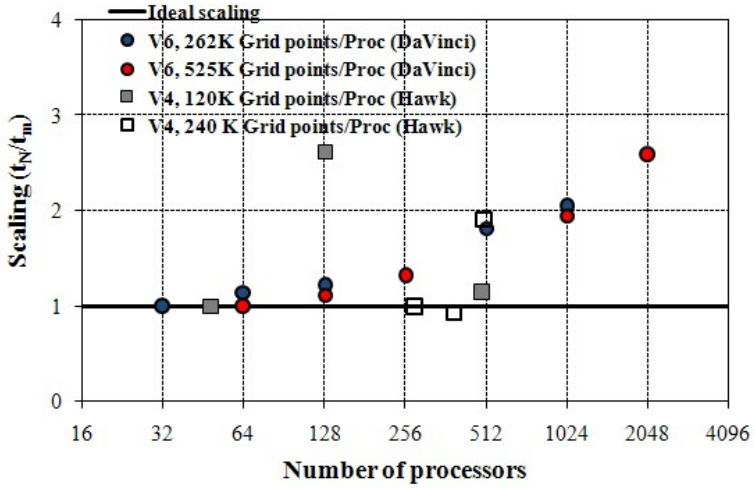
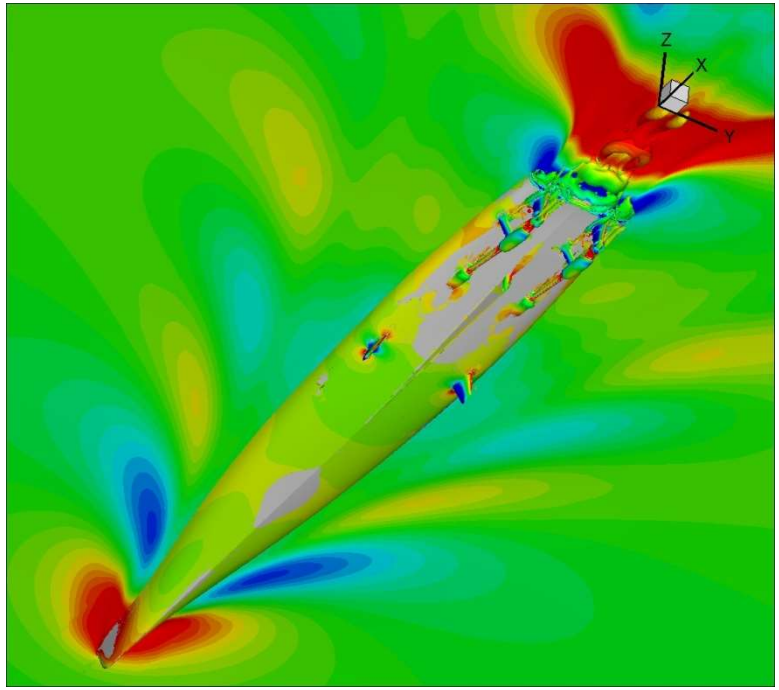


Figure: Weak scalability of total times without I/O for CFDShip-Iowa V6 and V4 on IBM P6 (DaVinci) and SGI Altix (Hawk) are compared with ideal scaling.

Post-Processing

- Post-processing: 1. Visualize the CFD results (contour, velocity vectors, streamlines, pathlines, streak lines, and iso-surface in 3D, etc.), and 2. CFD UA: verification and validation using EFD data
- Post-processing usually through using commercial software



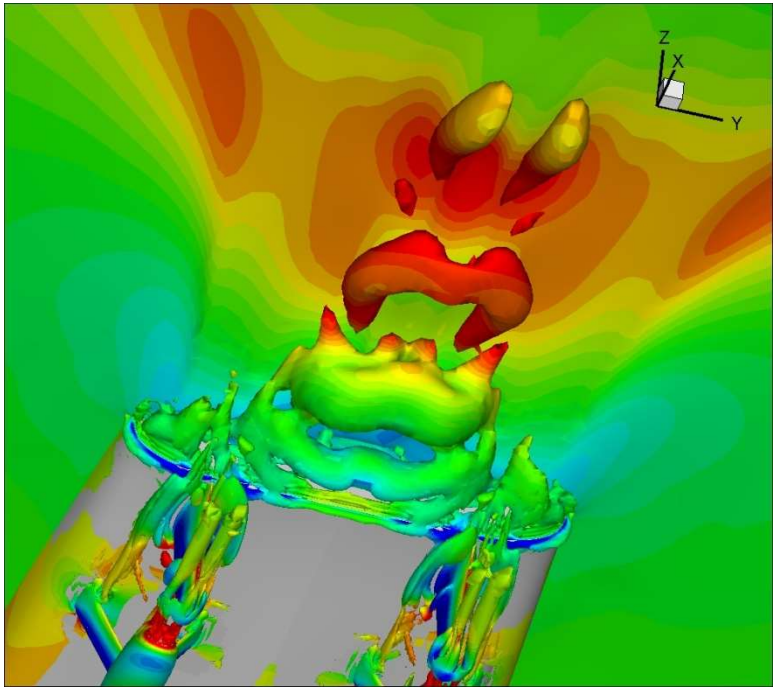


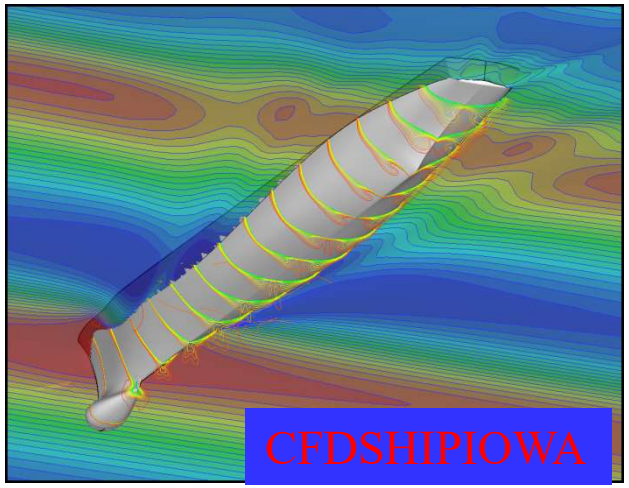
Figure: Isosurface of Q=300 colored using piezometric pressure, free=surface colored using z for fully appended Athena, Fr=0.25, Re=2.9×10^8. Tecplot360 is used for visualization.

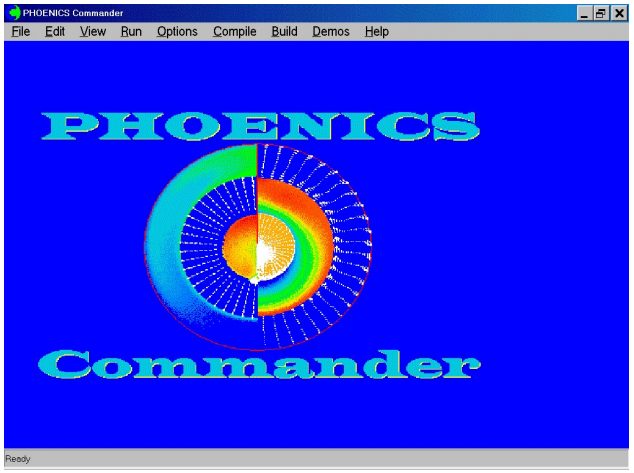
Types of CFD codes

- Commercial CFD code: ANSYS FLUENT, Star-CD, CFDRC, CFX/AEA, etc.
- Research CFD code: CFDSHIP-IOWA
- Public domain software (PHI3D, HYDRO, and WinpipeD, etc.)
- Other CFD software includes the Grid generation software (e.g. Gridgen, Gambit) and flow visualization software (e.g. Tecplot, FieldView)

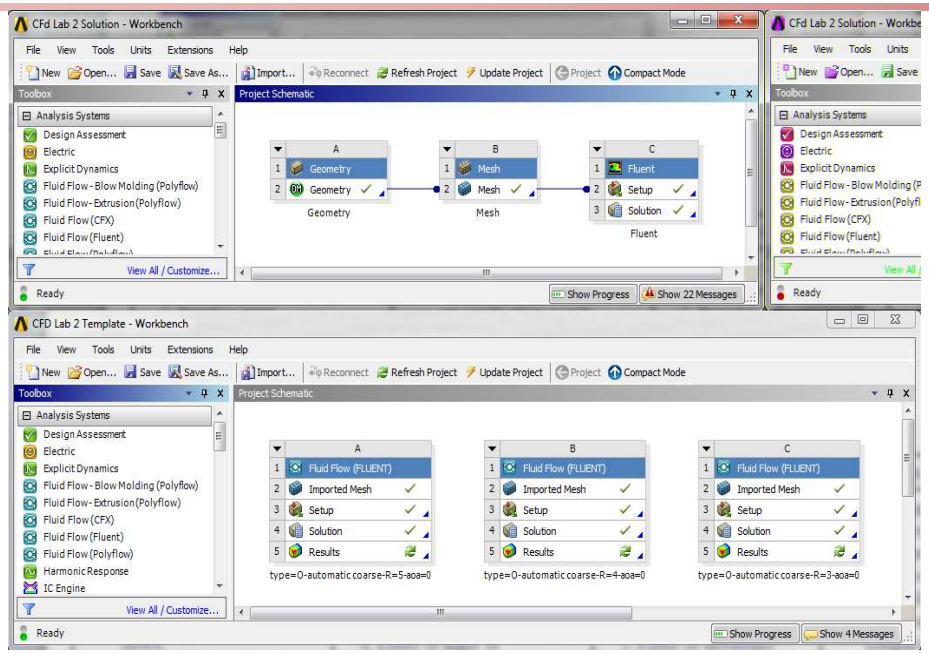
CD adapco Group

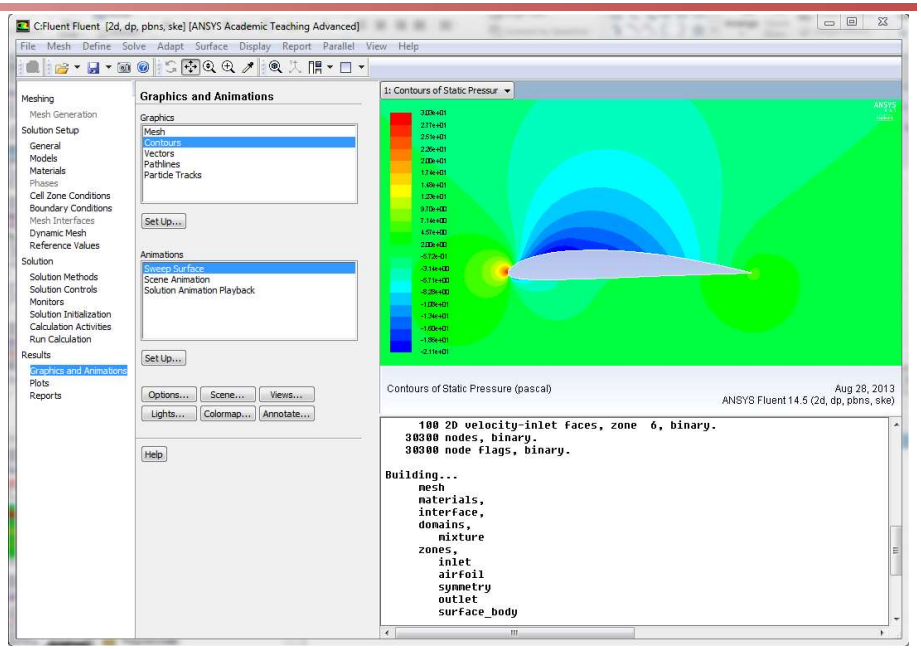
Gridgen
Reliable CFD Meshing





ANSYS Interface



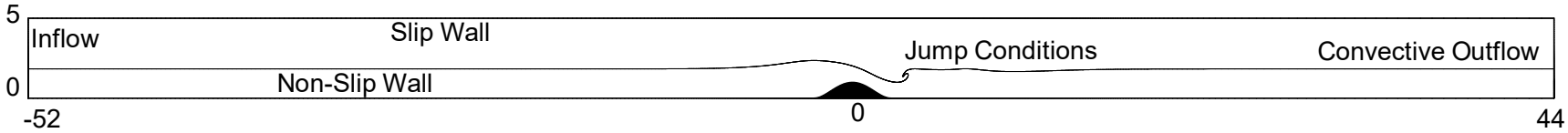


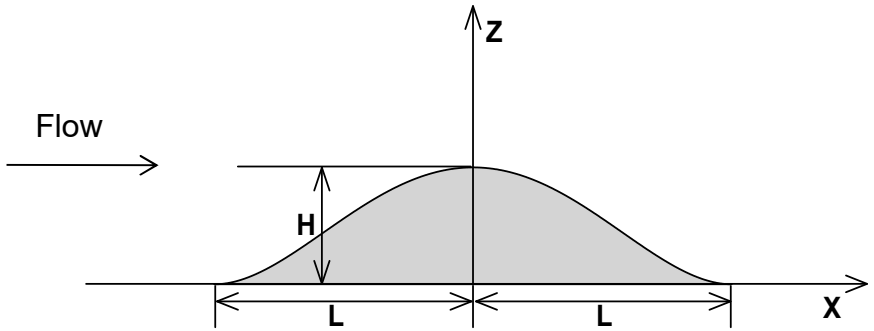


Part III
Simulation

I. Wave Breaking

Previous research -Plunging wave breaking



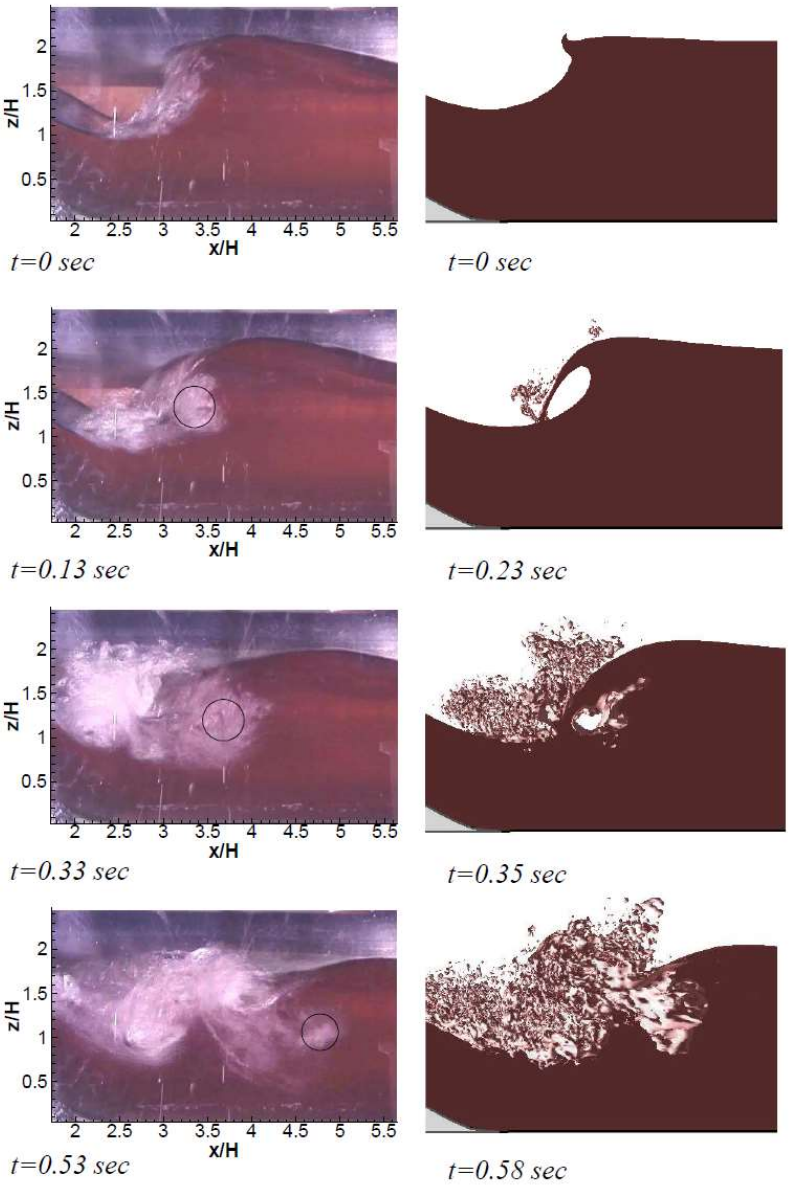


- The experimental study with detailed flow conditions and the corresponding CFD study are presented in (Kang et al., 2011; Koo et al., 2011), respectively.

Grid: 2.2 Billion (1920×1280x896), miminum Δh 0.25mm
Simulation was performed on Navy DoD HPC Einstein (Cray XT5), 4096 processes.

Previous research - Plunging wave breaking

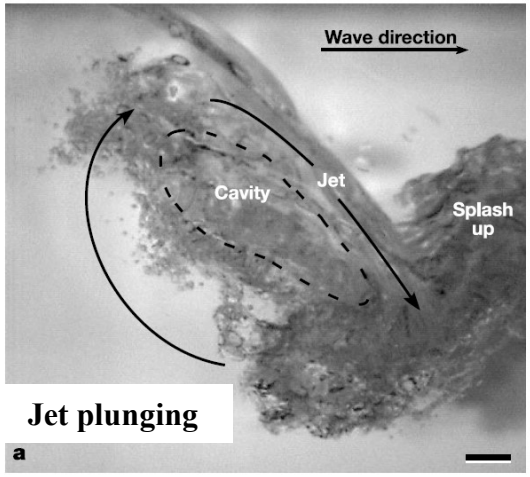
- The constant inlet velocity imposed at the left boundary is u = 0.87m/s.
- The initial interface elevation is 0.2286m and a uniform velocity field is prescribed in the water domain with the air phase at rest.
- Reynolds number, Re, is 99441, Froude number, Fr, is 0.82 based on the bulk inlet velocity and the bump height (H = 0.1143 m).
- The flow conditions of current 3D simulation are similar to those used in (Wang et al., IJMF 2009) on a 2D Cartesian domain and immersed boundary method.
- The experimental study with detailed flow conditions and the corresponding 2D CFD study are presented in (Kang et al., 2011; Koo et al., 2011), respectively.
- Systematically refined grids were used: 480×320×224, 960×640×448, and 1920×1280×896 (2.2 Billion).

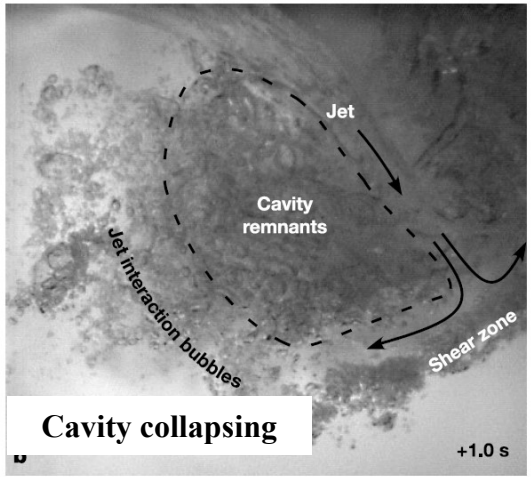


Wave breaking process behind a submerged bump.
Left: Experiment; right: Simulation (grid: 768×256×64).

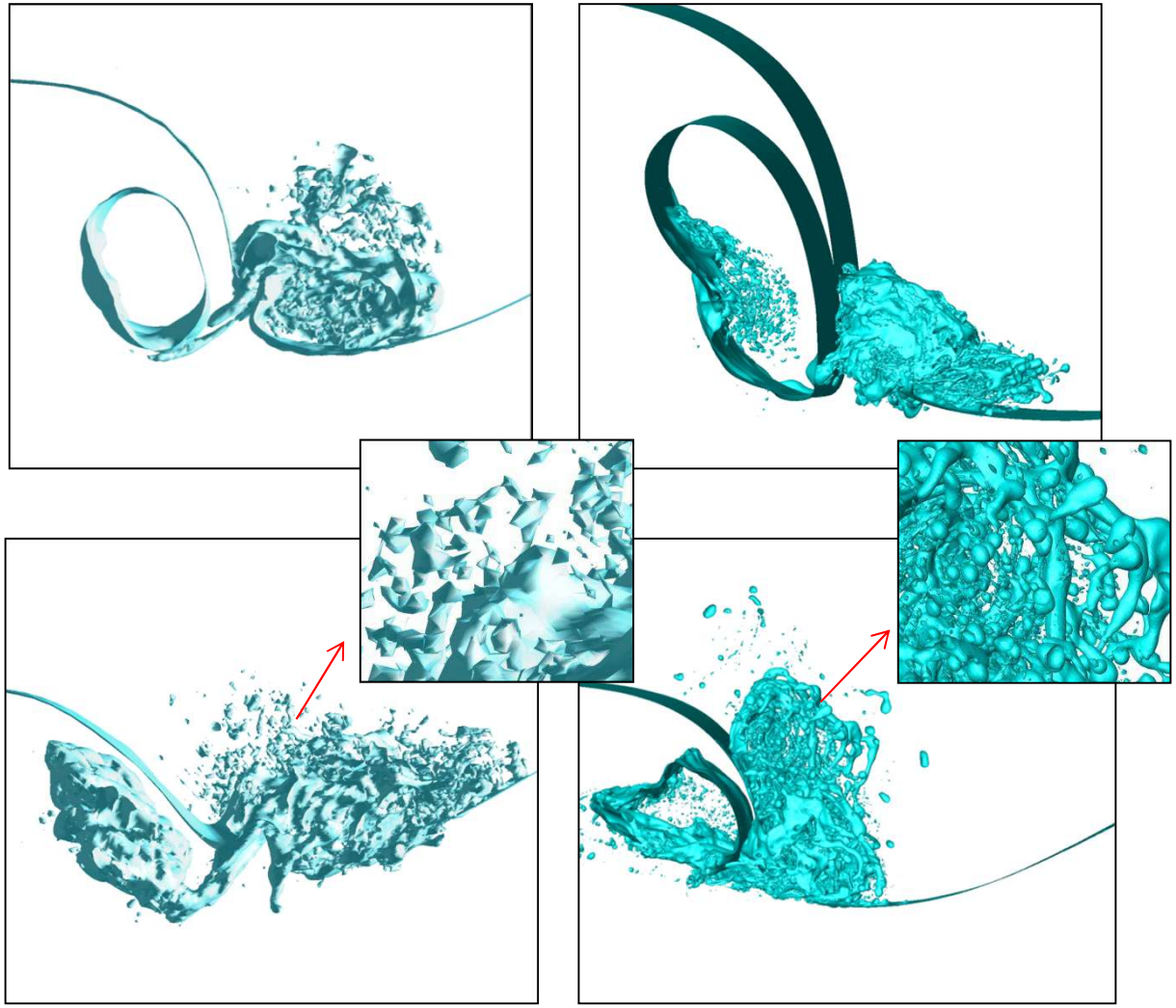
Previous research - Plunging wave breaking

Experiment (Deane & Stokes, Nature 2002)





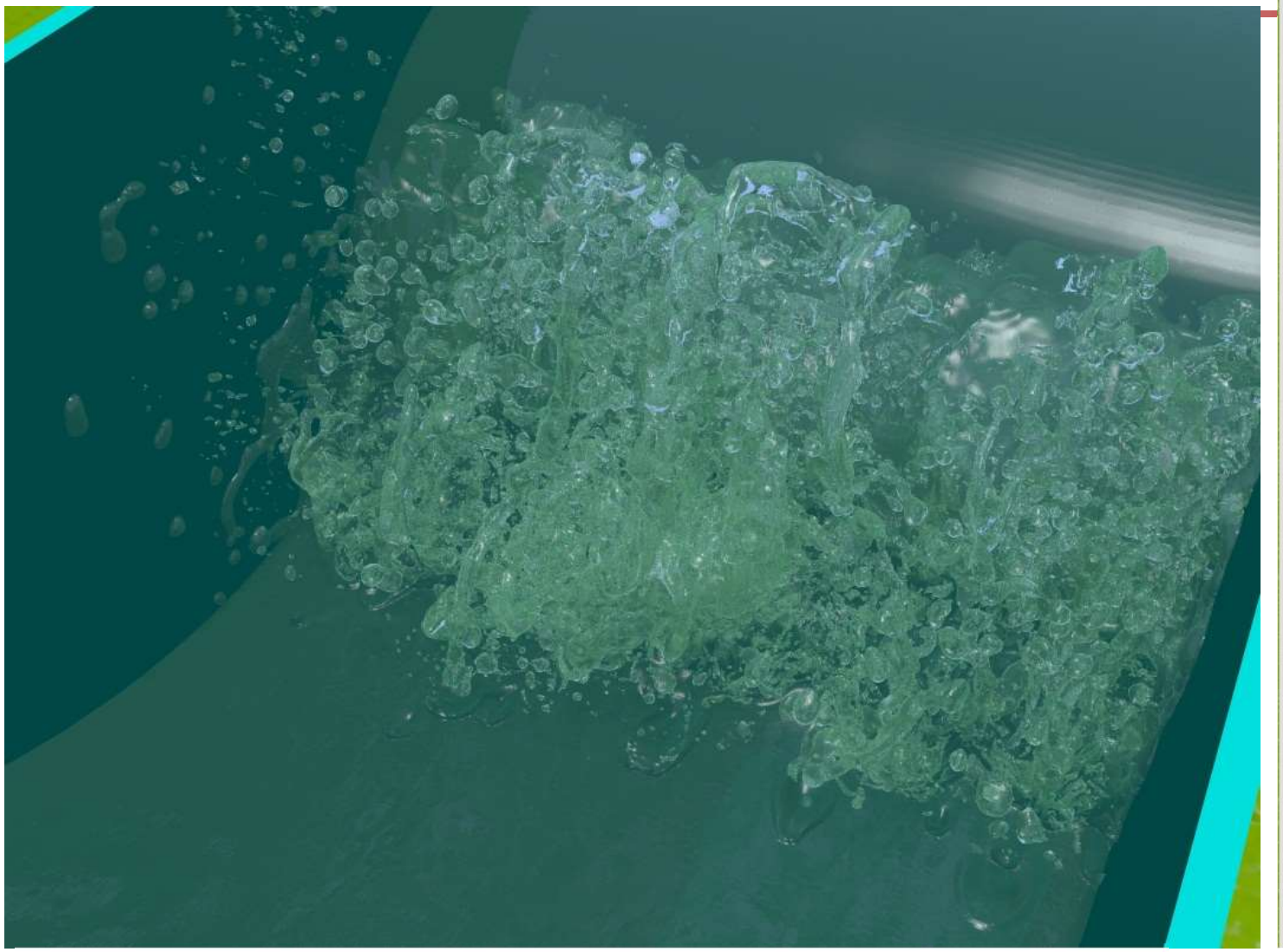
Grid: 480×320×224 (34M)



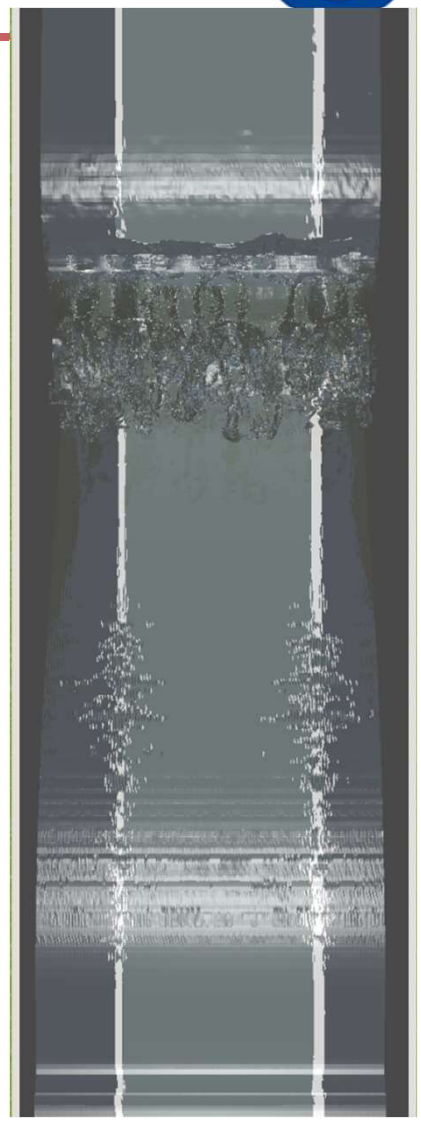
Grid: 1920×1280×896 (2.2B)

- Deane & Stokes (Nature 2002) showed two distinct conditions driving bubble creation in breaking waves.
  - Bubbles of sizes from 2 mm down to at least 0.1 mm are from the jet plunging period
  - Larger bubbles from 2 mm to ≥ 10 mm are created due to the collapse of the cavity

Previous research - Plunging wave breaking



Grid: 2.2 Billion (1920×1280x896), miminum Δh 0.25mm
Simulation was performed on Navy DoD HPC Einstein (Cray XT5), 4096 processes.



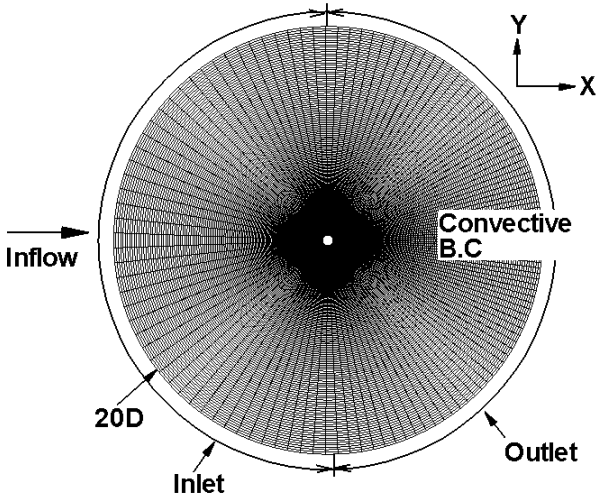
Top view of the wave breaking behind a submerged bump, grid 768×256×64.

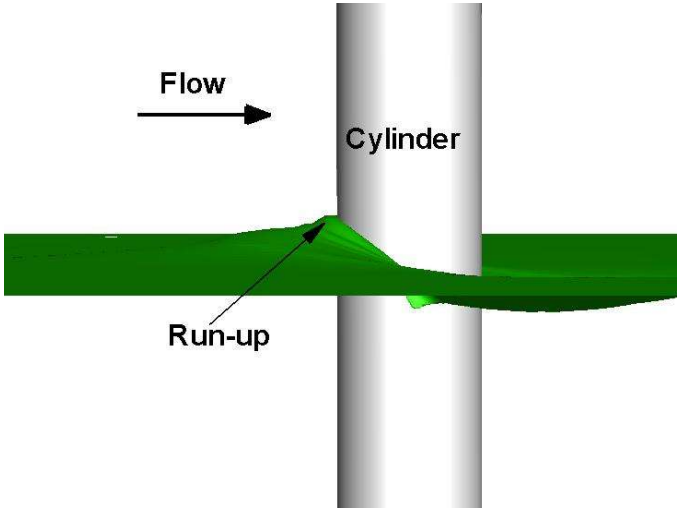
Movie

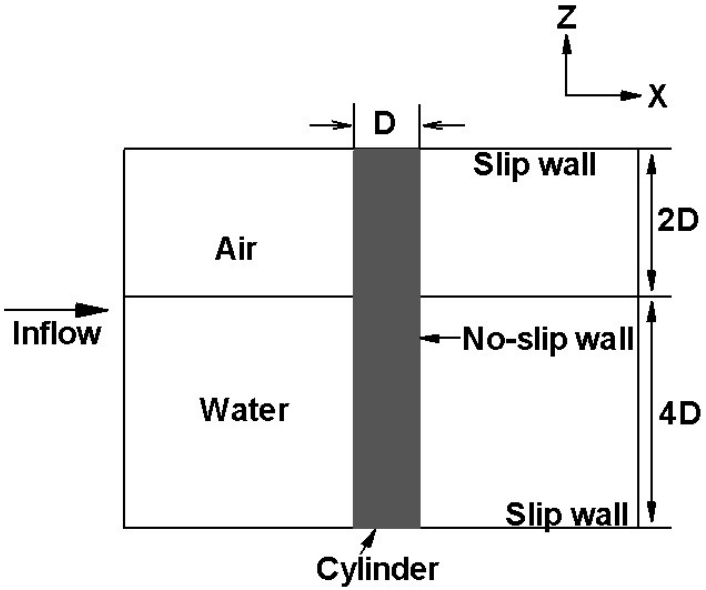
Previous research - Surface-piercing cylinder

Surface-piercing cylinder flow

- Noslip wall BC at cylinder
- Top/Bottom slip wall BC
- Constant velocity at inlet
- Convective BC at outlet







Previous research - Surface-piercing cylinder

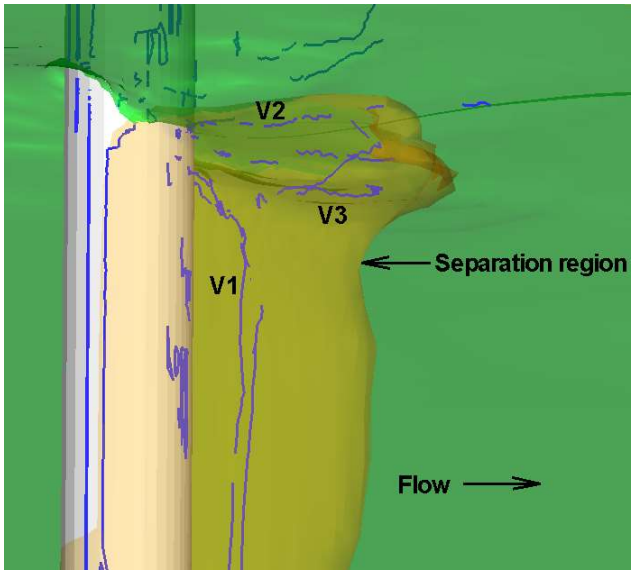
Case	Grid ($N_r \times N_\theta \times N_z$)	Re	Fr	Max y^+
1	4.2M, 256×128×128	55,800	0.2	0.96
2	4.2M, 256×128×128	123,000	0.44	0.96
3	4.2M, 256×128×128	27,000	0.84	0.96
4	4.2M, 256×128×128	234,000	0.84	0.96
5	4.7M, 272×136×128	346,000	1.24	»1
6	2.3M, 192×96×128 (C)	458,000	1.64	1.38
7	4.7M, 272×136×128 (M)	458,000	1.64	0.96
8	9.4M, 384×192×128 (F)	458,000	1.64	0.54
9	4.2M, 256×128×128	458,000	0.84	0.96
10	4.2M, 256×128×128	458,000	1.24	0.96

Movie

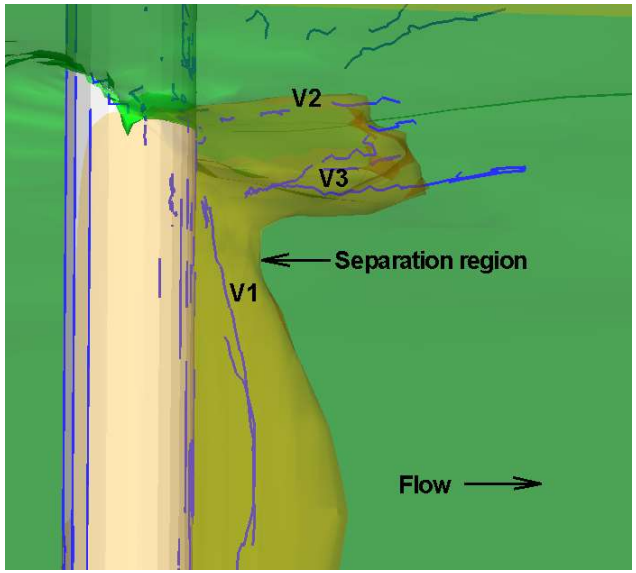
- Various Re (from sub-critical to critical) and Fr (from sub-critical to super-critical)
- Ratio of Re and Fr is 2.79×10^5 for the present simulations based on Chaplin and Teigen. (2003)
- Case 3,4, and 9 used for Re effect and case 8, 9, and 10 used for Fr effect
- Note: single-phase cases with Re=234,000, 458,000 has been simulated for correct separation point since case 4 and 9 has not enough resolutions in the deep flow
- Case 5 has not enough resolution near the cylinder wall (y^+»1) and will be corrected

Previous research - Surface-piercing cylinder

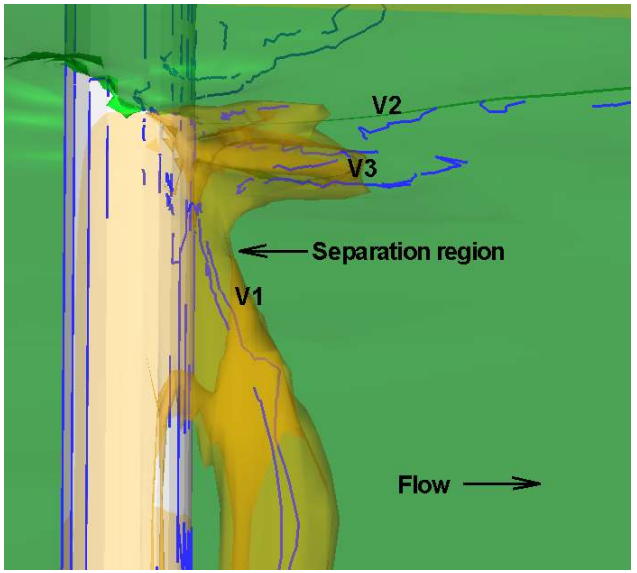
Mean separation pattern with vortex core line



Fr = 0.84 and Re = 27,000



Fr = 0.84 and Re = 234,000

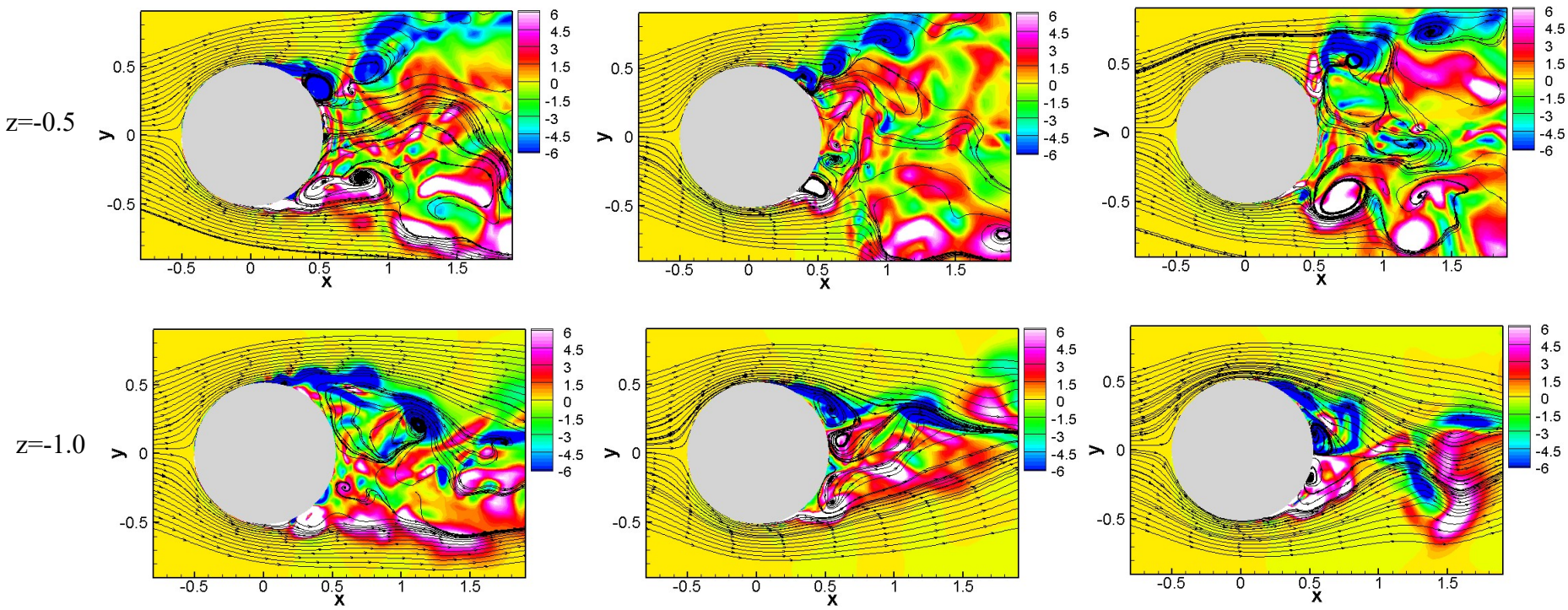


Fr = 0.84 and Re = 458,000

- Immediately below the interface, the separation region is reduced for all Re and "neck" shape is observed.
- The thickness of the neck is remarkably smaller with the critical Re at about z=-1
- At Re=458,000, it is interesting that the separated region is detached from the cylinder wall in deep water likely due to the separation bubble.

Previous research - Surface-piercing cylinder

Instantaneous vertical vorticity with streamlines



Fr = 0.84 and Re = 27,000

Fr = 0.84 and Re = 234,000

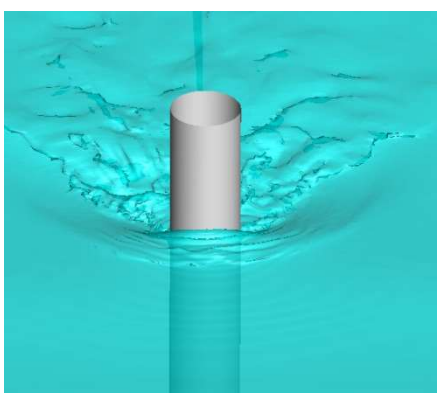
Fr = 0.84 and Re = 458,000

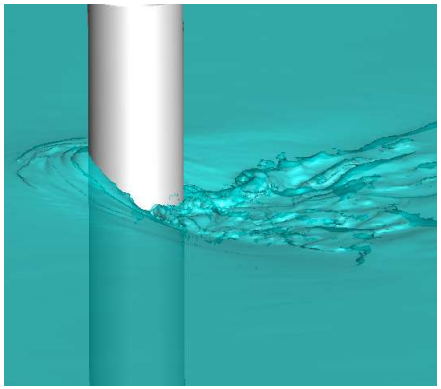
- Near the interface at z=-0.5 the attenuated vortex shedding. i.e. no organized periodic vortex is observed for all Re regimes due to the streamwise vorticity generated near the separated region.
- However, the separation point is much delayed with an increase in Re.
- At z=-1 where the “neck” shape is formed, the narrowing of the wake is observed for the critical Re (234,000, 458,000).

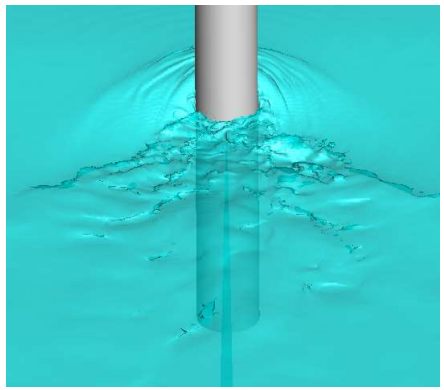
Previous research - Surface-piercing cylinder

Wave field

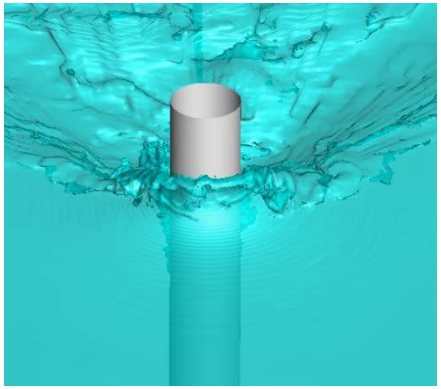
Fr = 0.84 and Re = 458,000

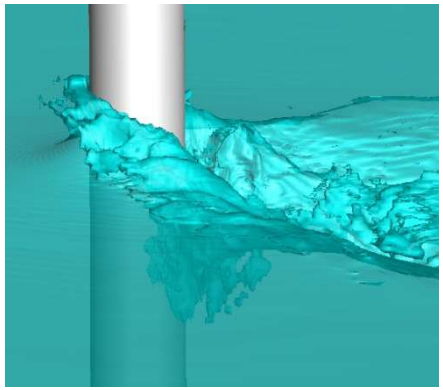


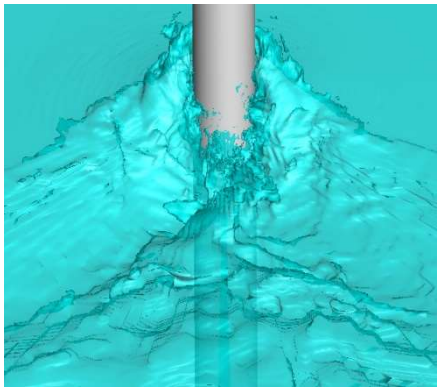




Fr = 1.64 and Re = 458,000







Front

Side

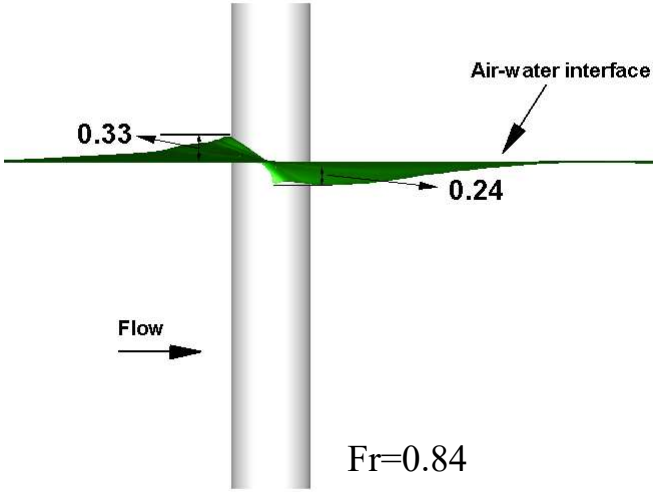
Wake

•For subcritical Fr (= 0.84), relatively smaller bow wave are observed in front of the cylinder with Kelvin waves behind the cylinder. There are some interface roughness and turbulence in the wake region behind the cylinder.

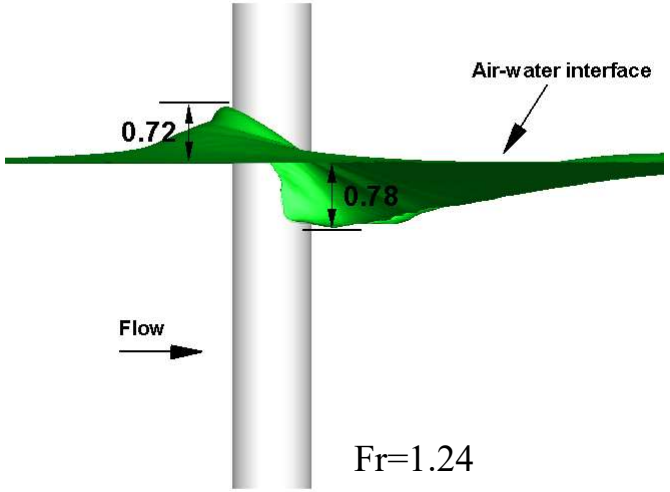
•For Fr = 1.64, the bow wave increases remarkably with the larger wake region and it also breaks with similar patterns of plunging breakers. Much more small air-water interface structures including splashes and bubbles are observed behind the cylinder.

Previous research - Surface-piercing cylinder

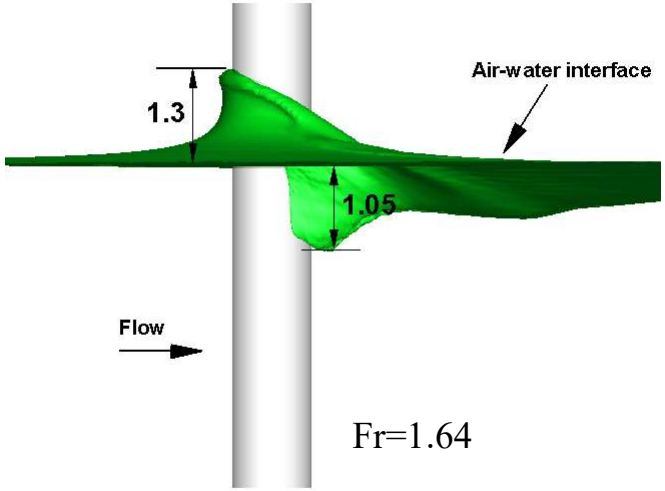
Wave Run-up



Fr=0.84



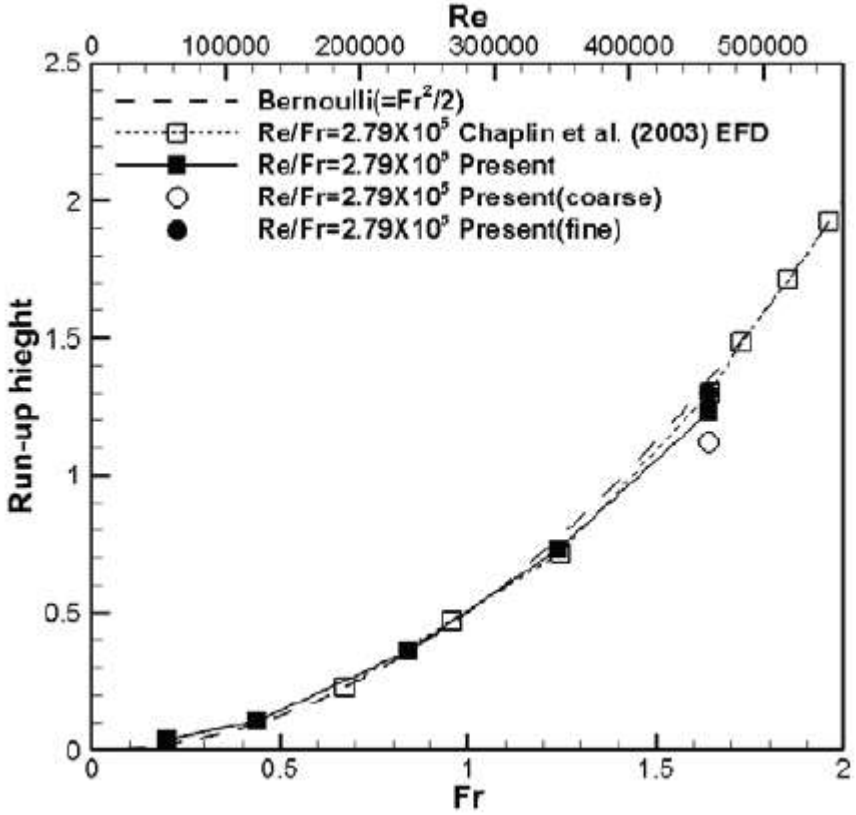
Fr=1.24

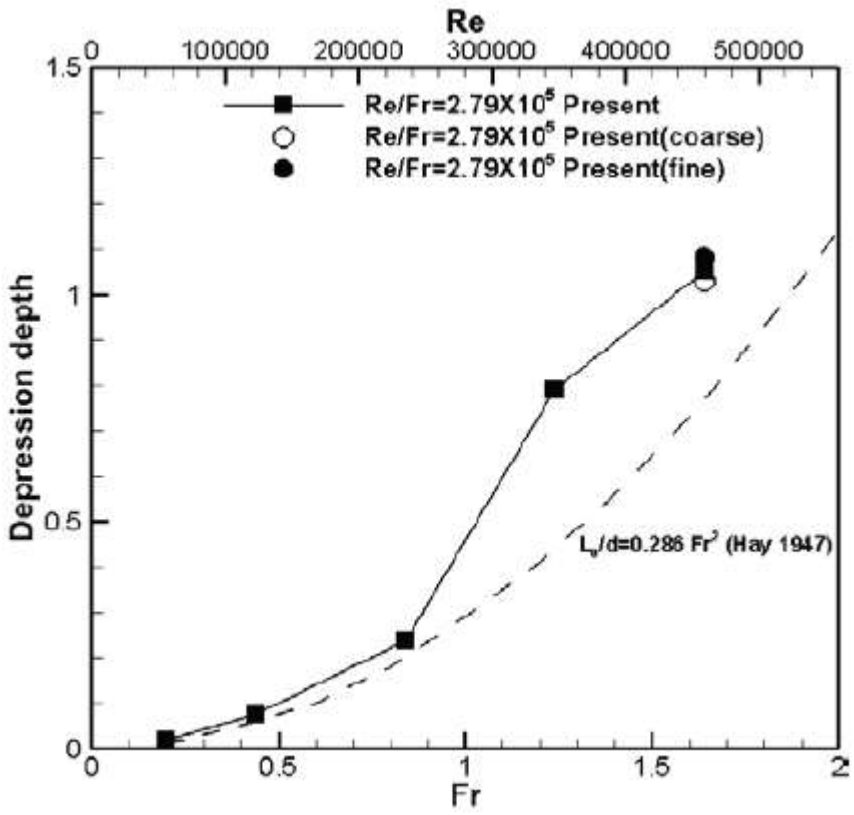


Fr=1.64

Previous research - Surface-piercing cylinder

Wave Run-up

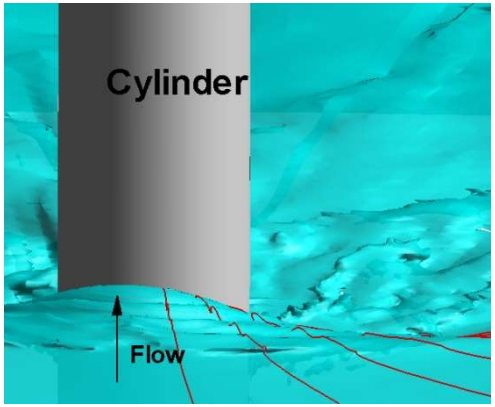




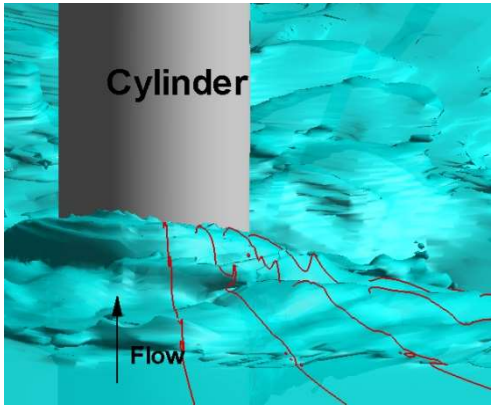
- Chaplin et al. (2003) and present simulations used a ratio of Re/Fr as 2.79×10^5
- Present results have good agreement with Bernoulli's equation (= $Fr^2/2$) and other studies

Previous research - Surface-piercing cylinder

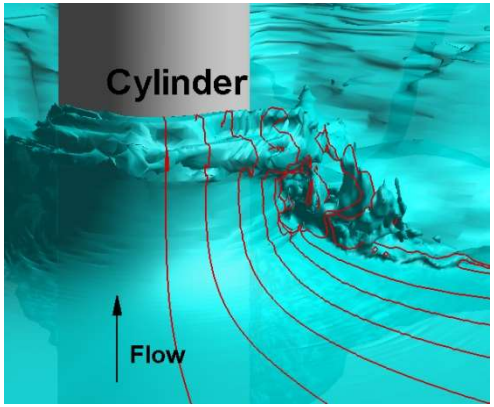
Iso-surface of air-water interface with the slices of the wave profile cut



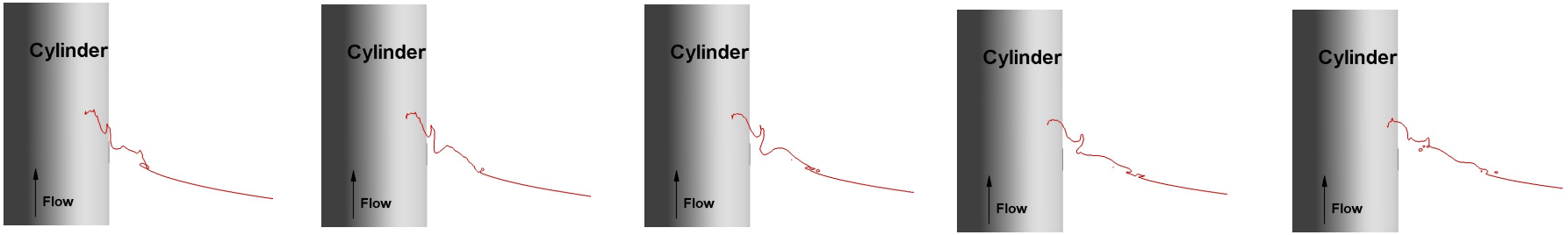
Fr = 0.84 and Re = 458,000



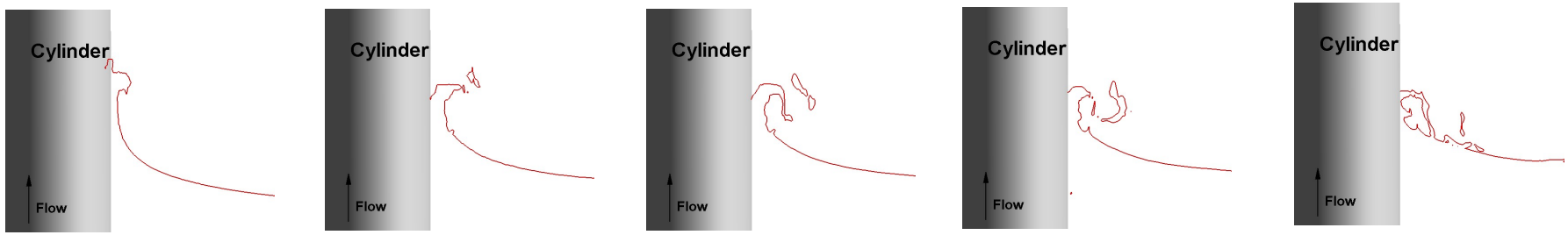
Fr = 1.24 and Re = 458,000



Fr = 1.64 and Re = 458,000



Fr = 1.24 and Re = 458,000

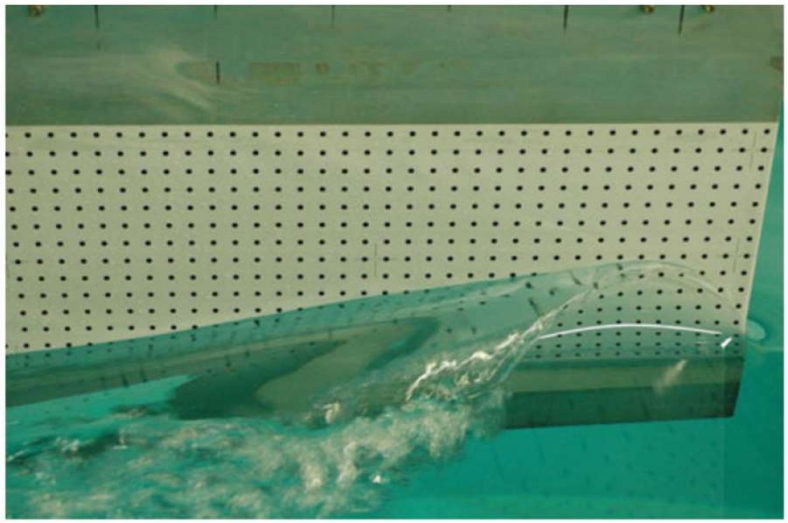


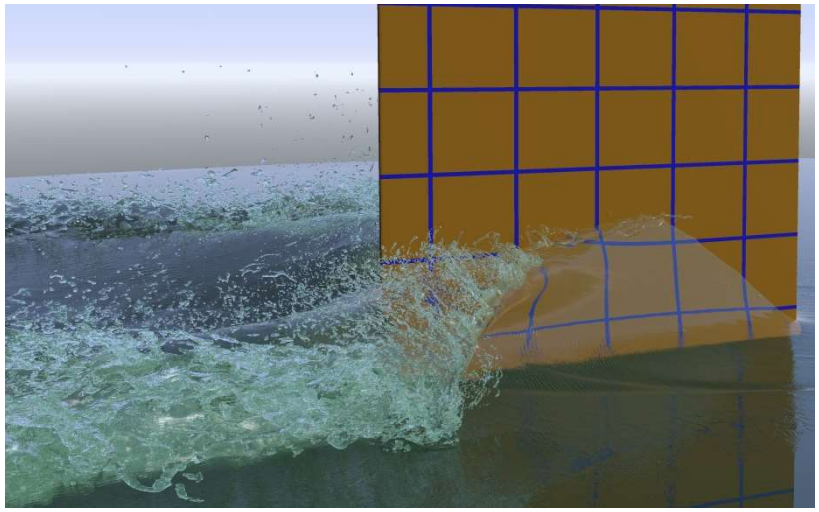
Fr = 1.64 and Re = 458,000

Previous research - Bow wave breaking

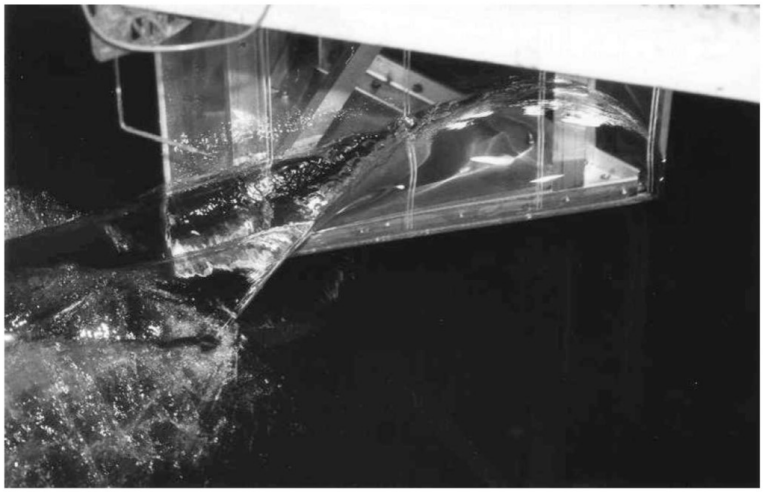
Bow wave breaking

Orthogonal Curvilinear Grid: 1536×768×848 (1 Billion); minimum Δh = 0.125 mm. Run on **1024 cores**

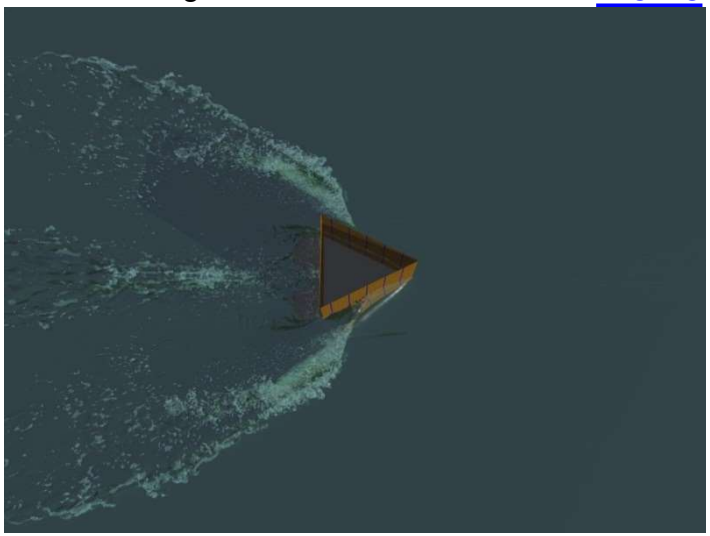




Left: Experiment (Noblesse *et. al,* 2008); Right: simulation. **Movie**

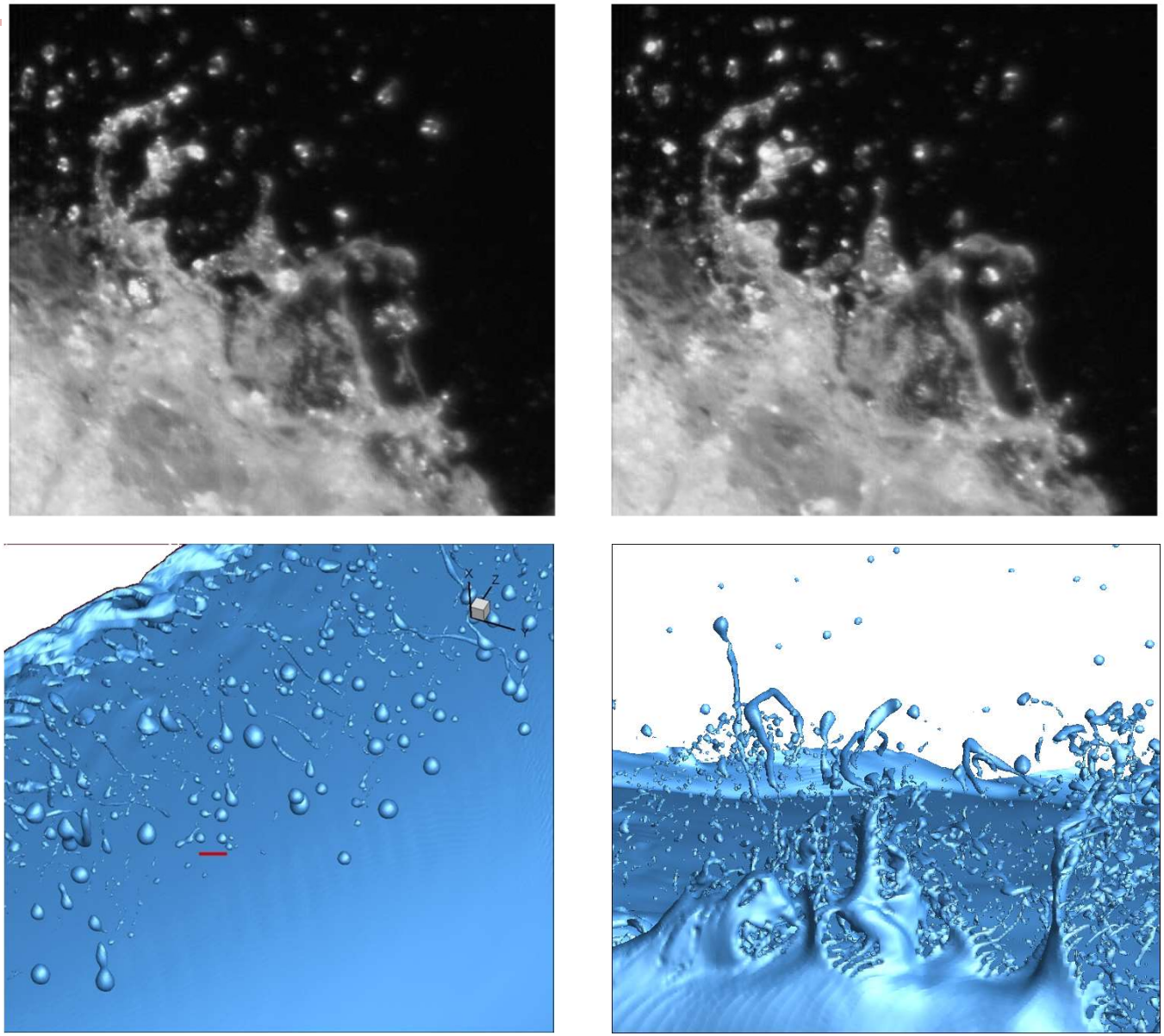


Wave profiles from experiment (Waniewski et. al, 2002)



Top view, rendered using POV-ray.

Previous research – Bow wave breaking



The minimum droplet is around 0.8 mm in diameter.

Close-up view of the wave sheet breakup
Top: Experiment (Karion et al., 2004); Bottom: Simulations.

II. Ship Flows

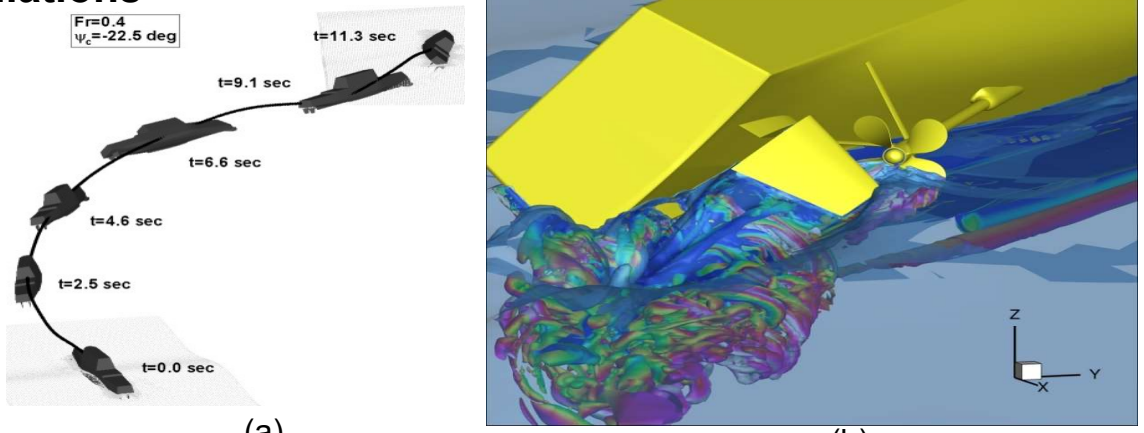
Previous research - Maneuvering

Course keeping in calm water and waves

a) Broaching and capsize simulations

i. Grid size 3.7M-21M
ii. Body force and actual prop.
iii. Validations for motions and instability map

Movies:
Broaching
Close-up



(a) (b)

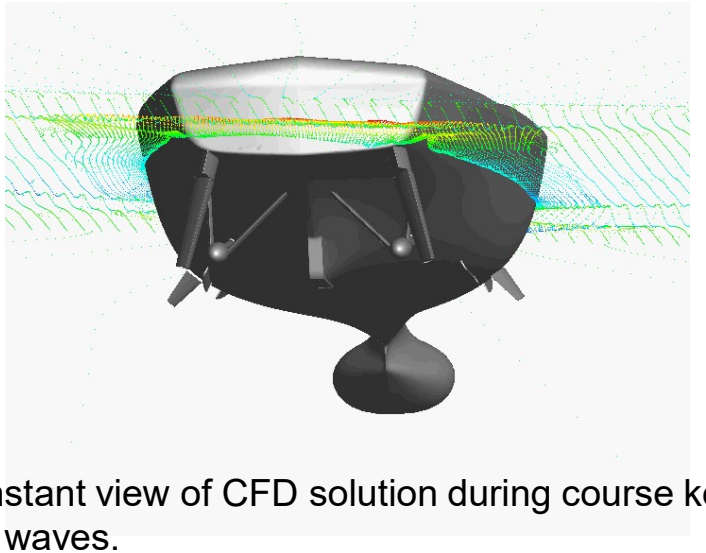
Broaching simulation of ONR Tumblehome : (a) body force propeller; (b) actual propeller

Previous research - Free running simulations

b) Course keeping with active fins for 5415M in calm water and regular waves

i. Grid size 6.3-18.6 M
ii. Body force prop.
iii. Active rudder and fin
iv. Validations for 6DOF motions and loads

Movies:
Forced roll

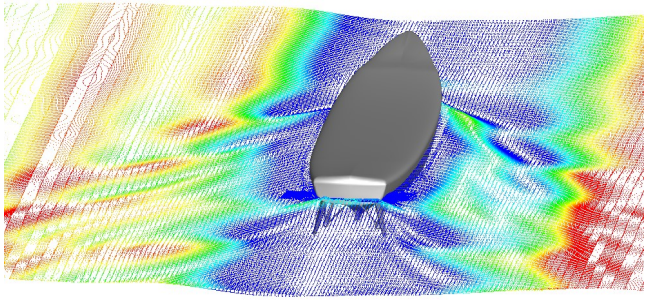


The instant view of CFD solution during course keeping in beam waves.

c) Course keeping with active fins for 5415M in irregular waves

i. Grid size 24 M
ii. Body force prop.
iii. JONSWAP wave spectrum

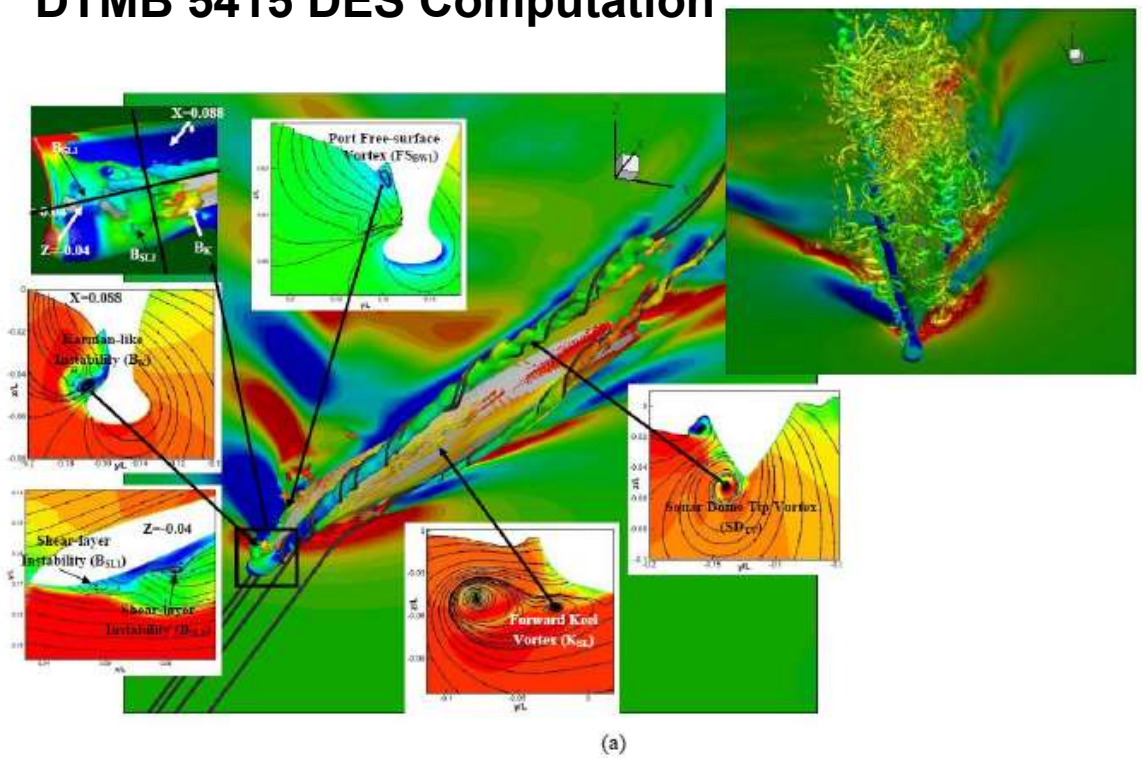
Movies:
Irregular beam waves



CFD simulation of course keeping in irregular beam waves with JONSWAP spectrum.

Previous research - Turbulent modeling

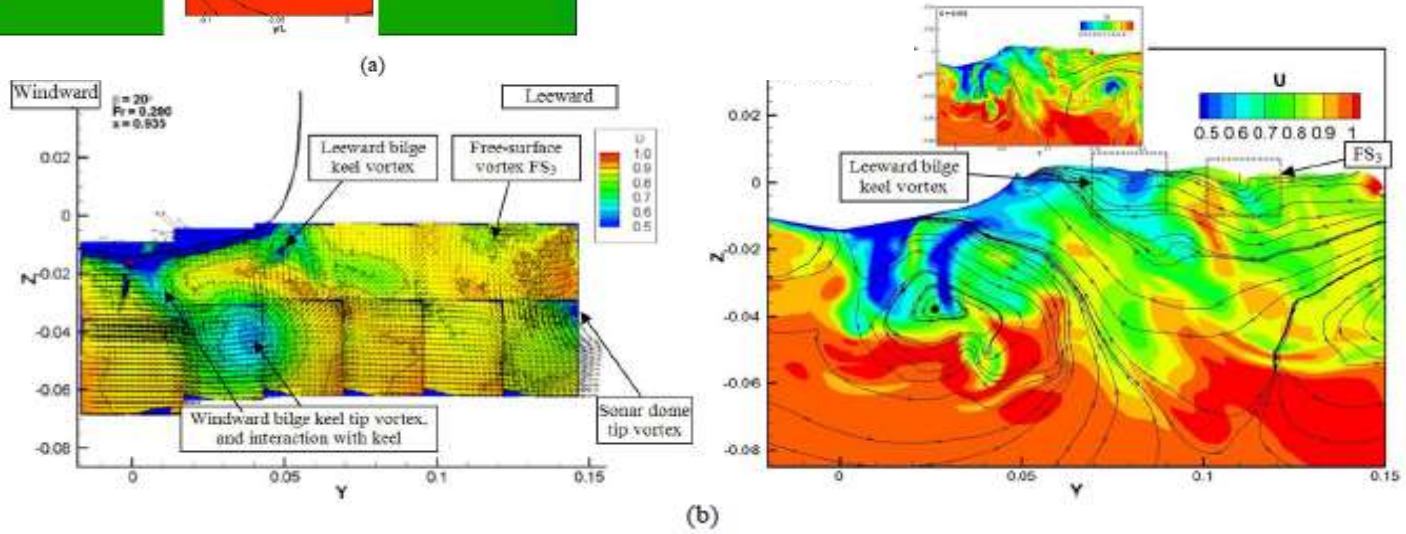
DTMB 5415 DES Computation



(a)

(a) Large scale vortical structures and instabilities identified for 5415 at β = 20. (b) Initial comparison of streamwise velocity contour and cross flow vectors and wake at X=0.935.

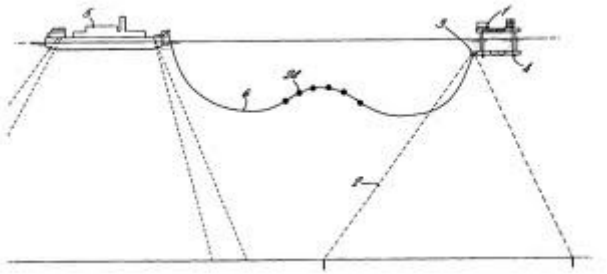
Movie: 5415 β=20º, 50M grid



(b)

Previous research - working projects in SHI

1. Wave load analysis on offshore structures
2. Sloshing analysis
3. Mooring analysis
   - CPF Semi-submergible
   - SHELL FLNG
   - TOTAL FSO
   - PETRONAS FLNG
   - EGINA FPSO





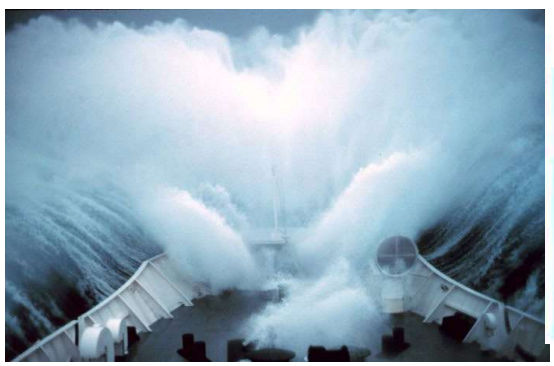
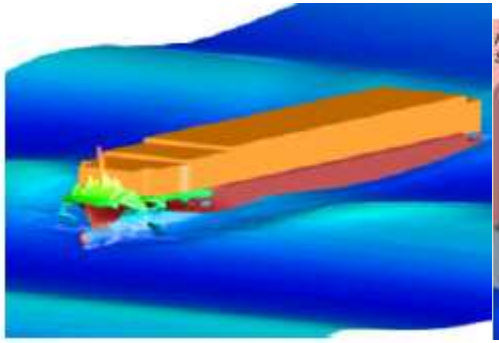
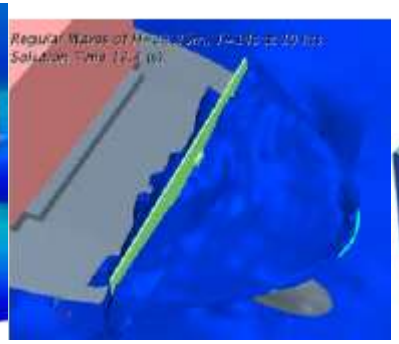
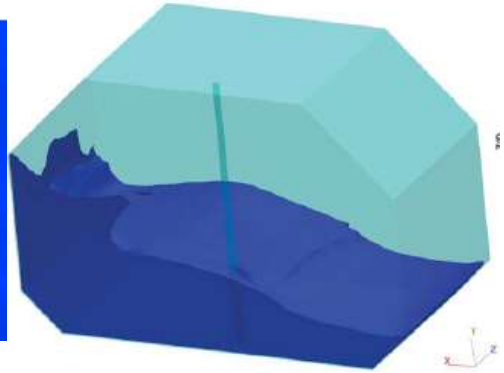




Previous research - working projects in SHI

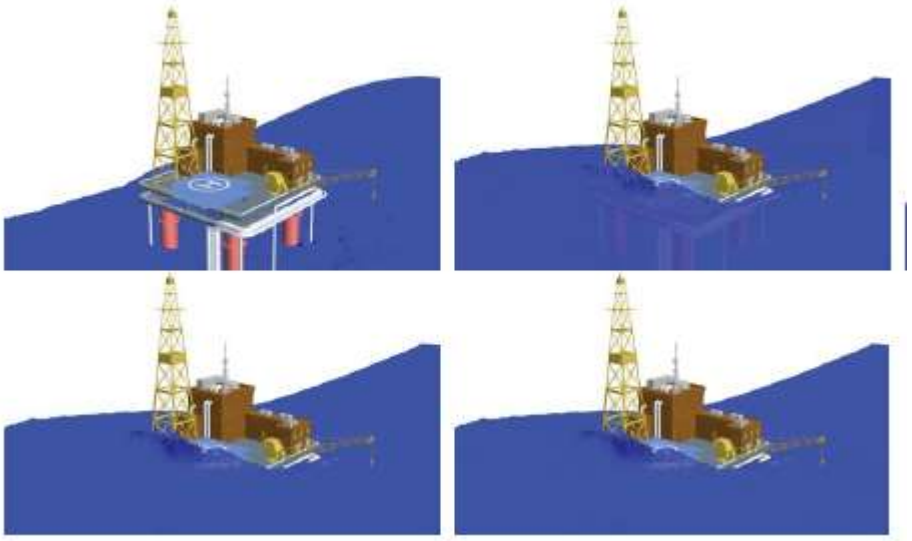
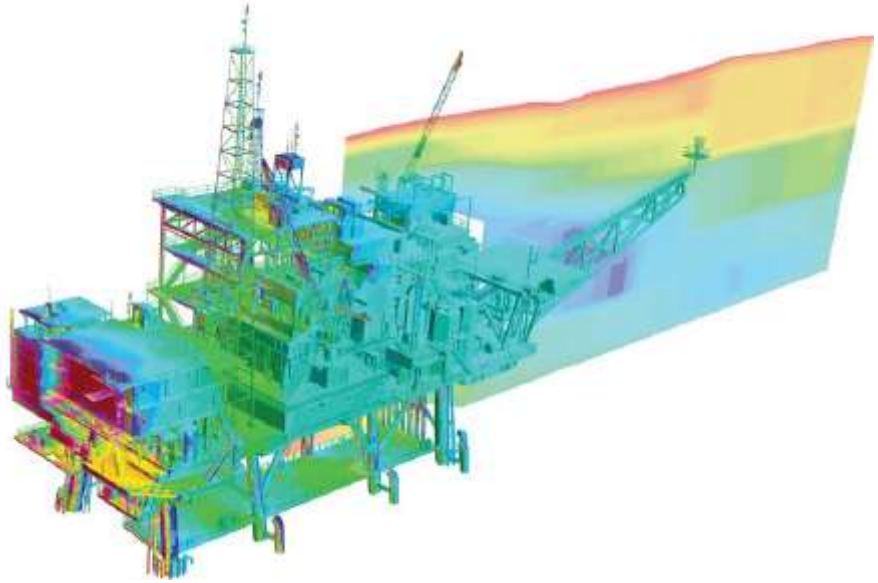
1. Impact wave load analysis on offshore structures (CFD)

- Numerical analysis: Slamming; Green-water; Sloshing

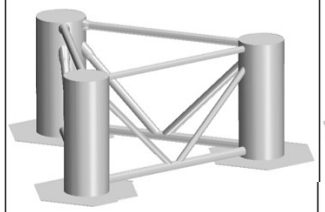
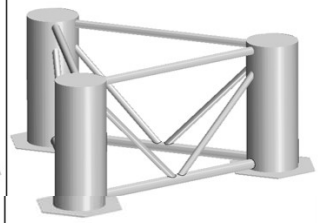
2. Non-linear wave load on offshore structures (CFD)

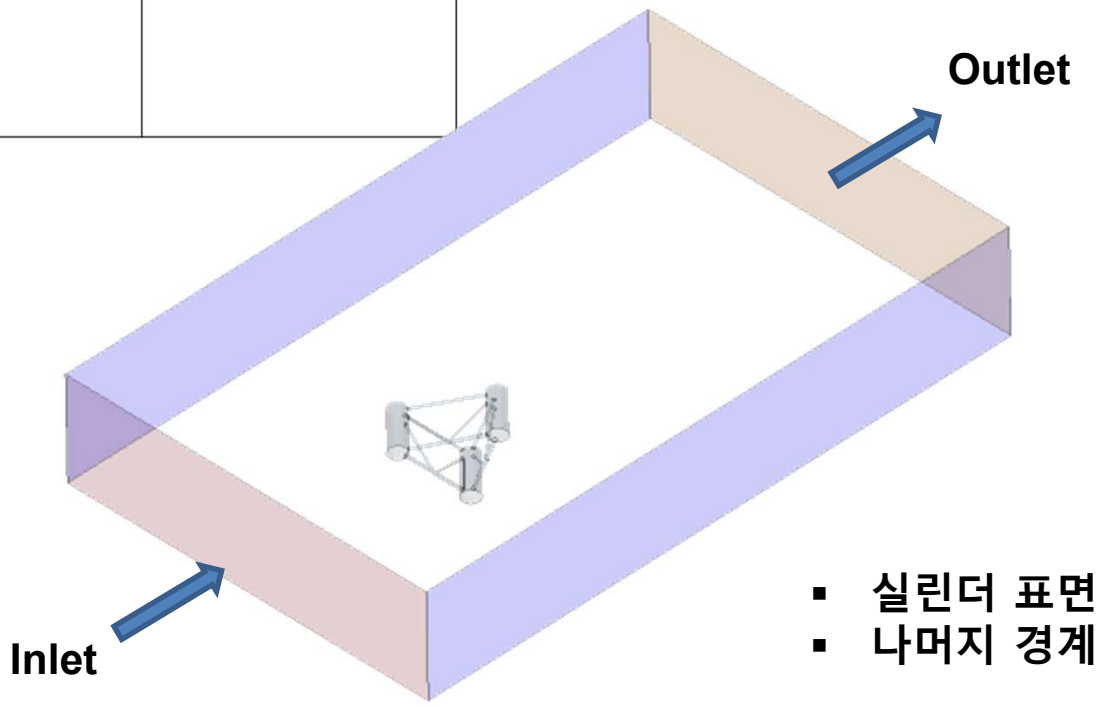
- Semi-submergible towing problem

III. Current/future Research

Current/future research – Wind turbine

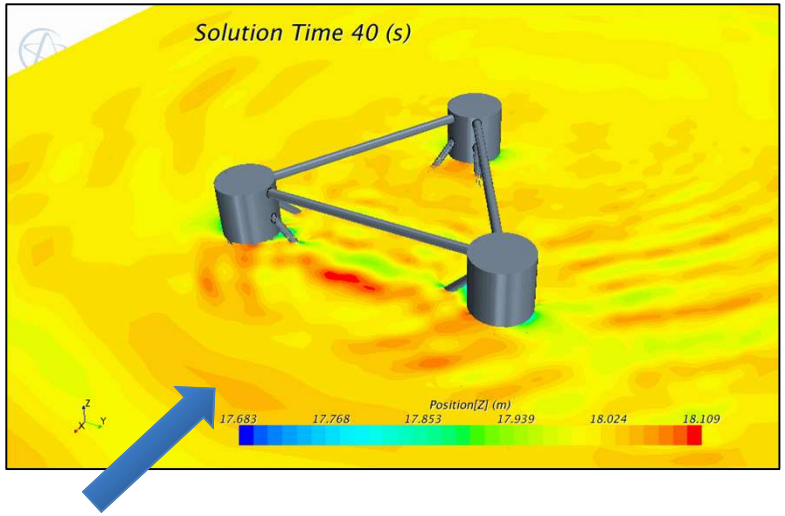
6MW (with tower)	6MW (w/o tower)	8MW (w/o tower)
		



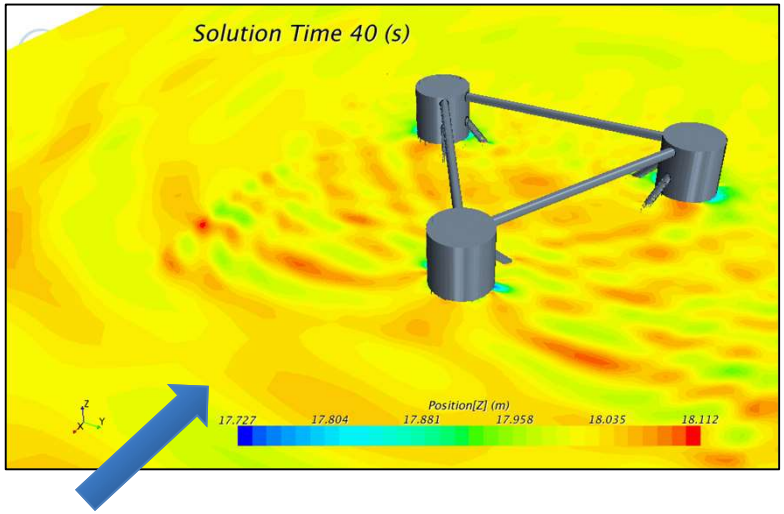
- 실린더 표면: No-slip
- 나머지 경계면: Symmetry

Current/future research – Wind turbine

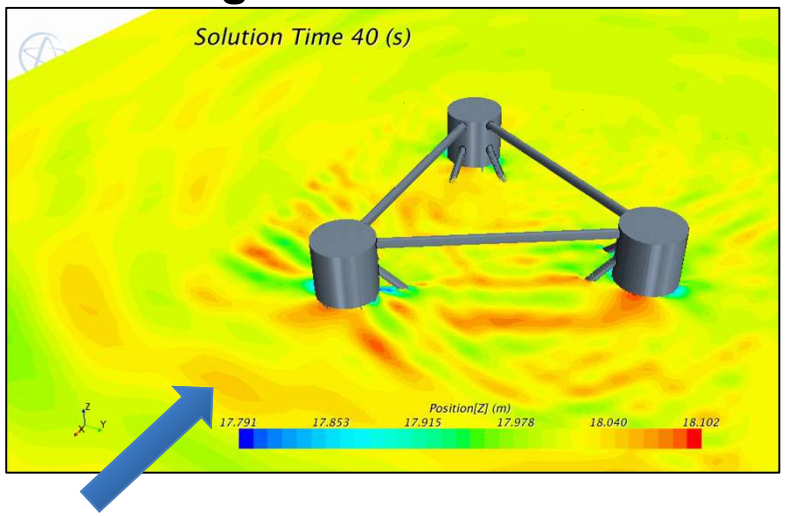
- 0 deg.



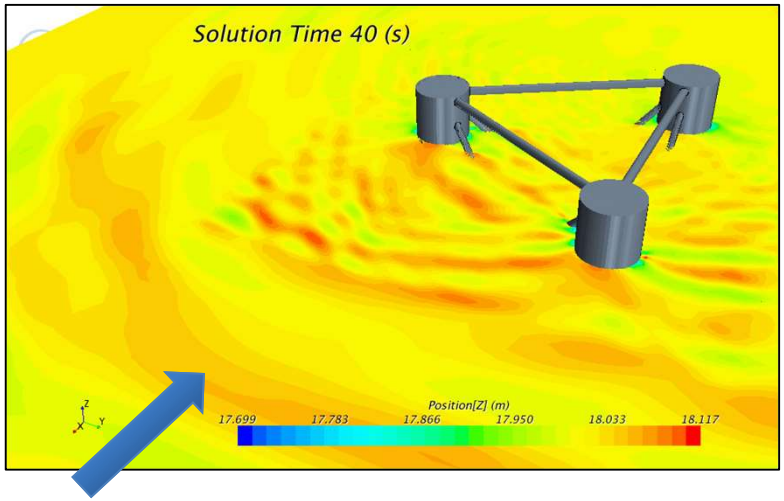
- 60 deg.



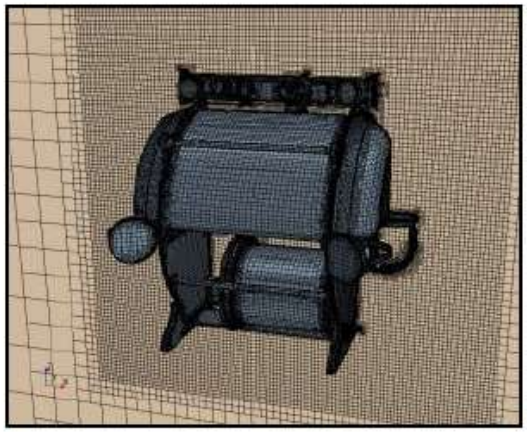
- 30 deg.

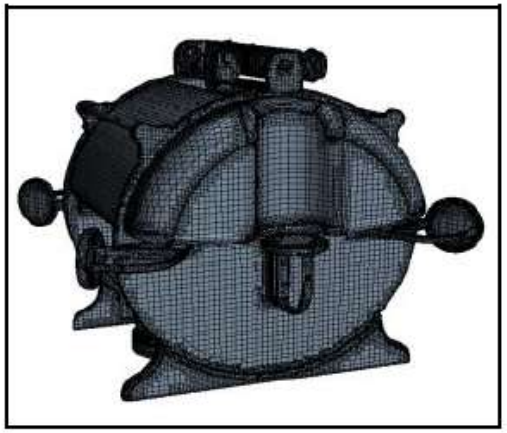


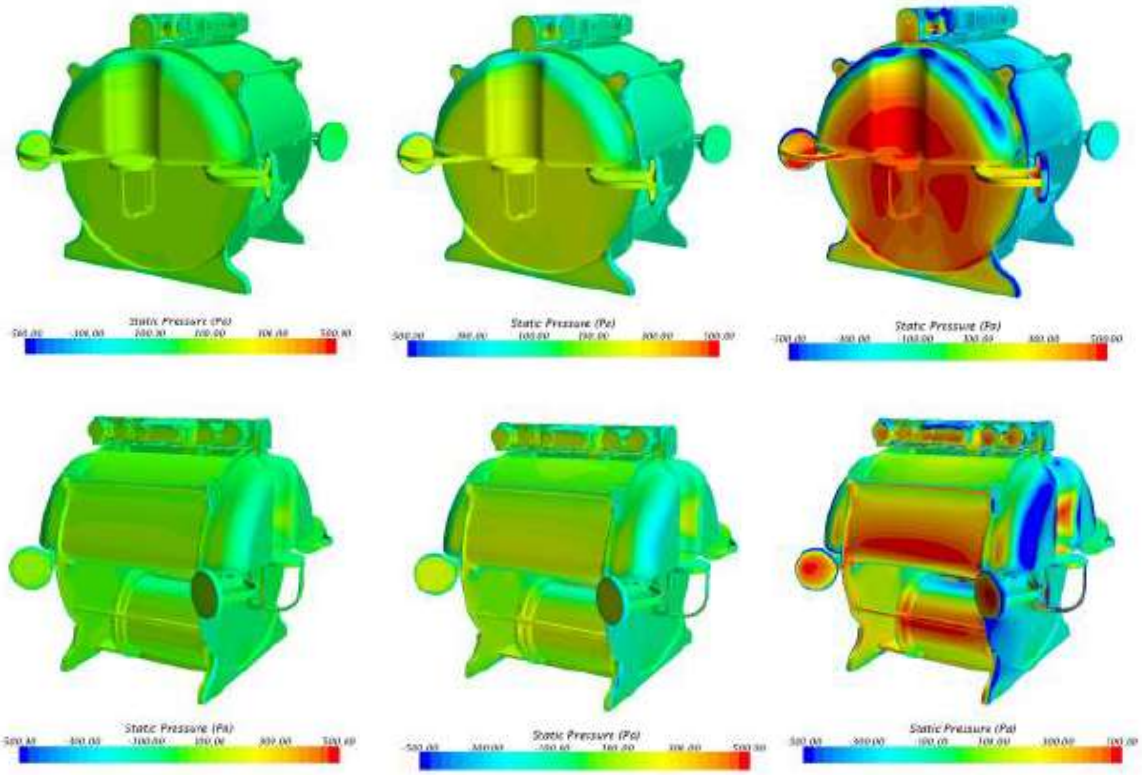
- 90 deg.



Current/future research – Underwater vehicle







Current/future research – Loop-shape wind turbine

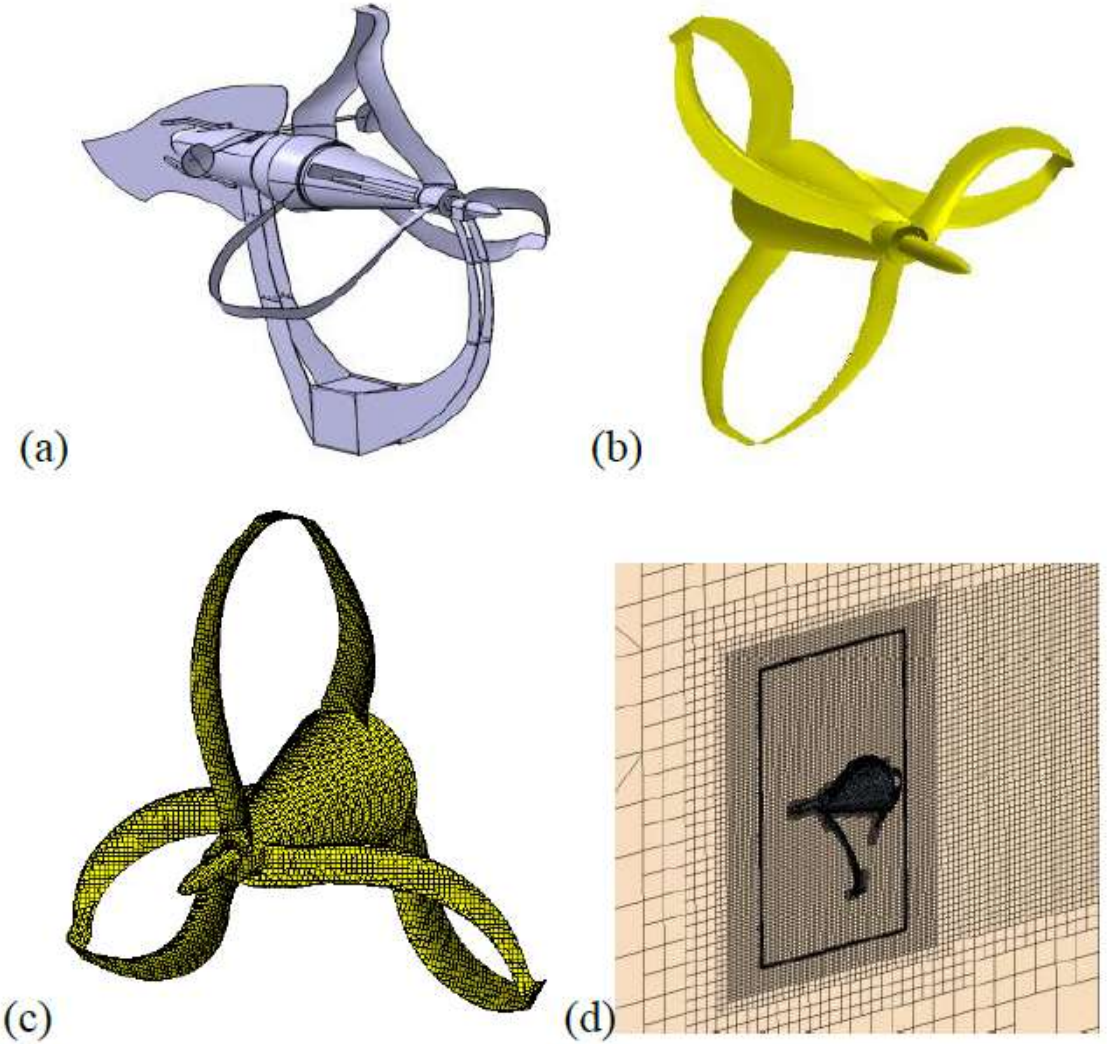


Figure 1. Loop-shape wind turbine model with grid system; (a) real model, (b) modified model, (c) wind turbine with grid, (d) wind turbine in computational domain

Current/future research – Loop-shape wind turbine

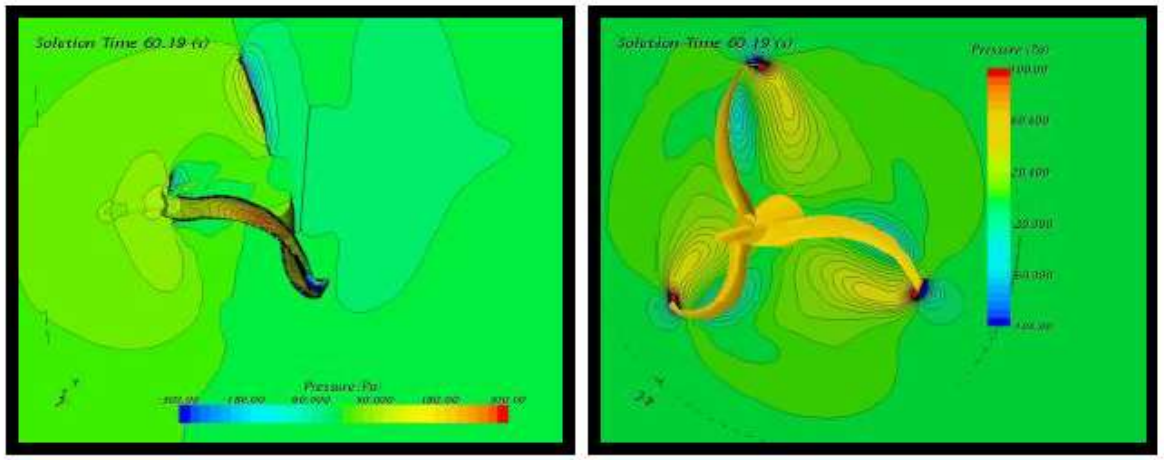


Figure 2. Pressure distributions for v=13m/s; front (left), back (right)

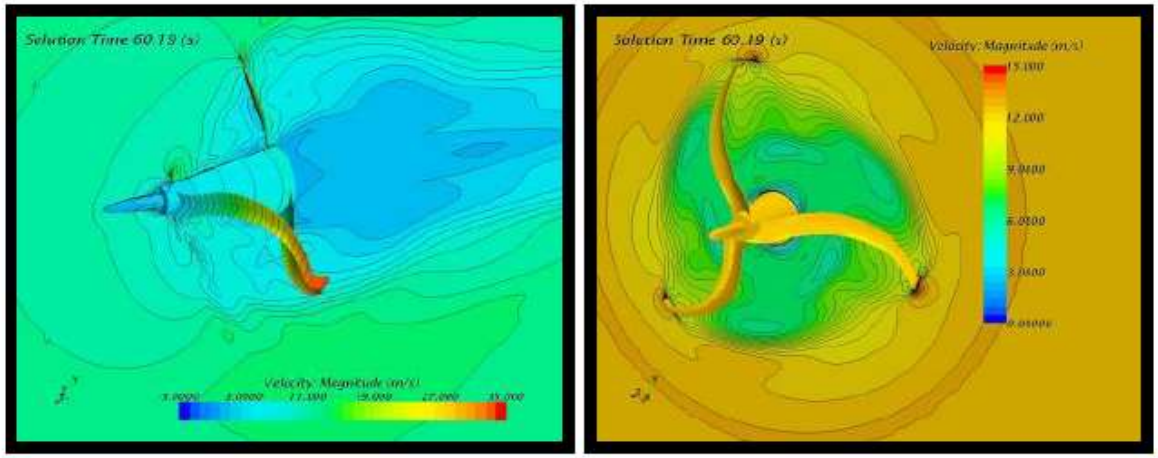
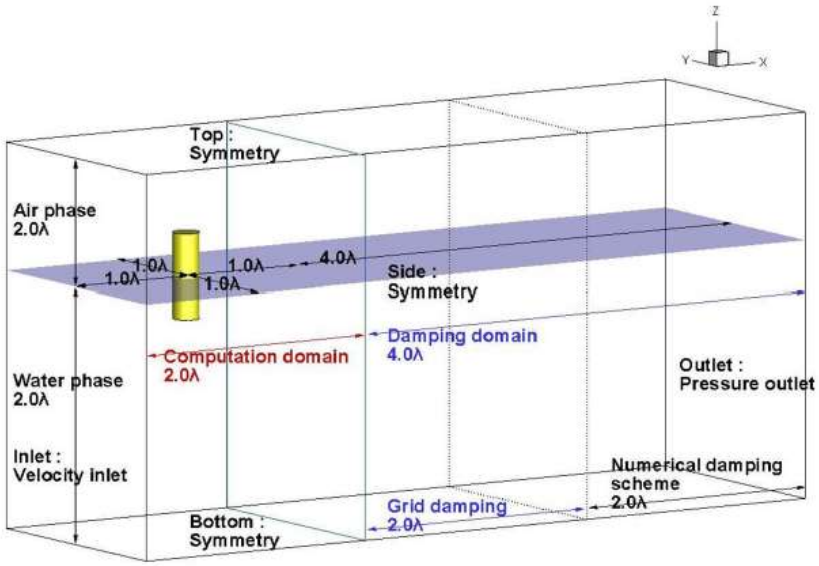
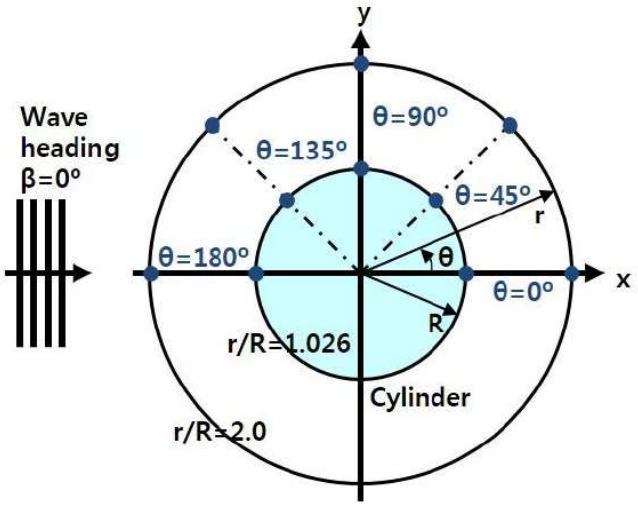
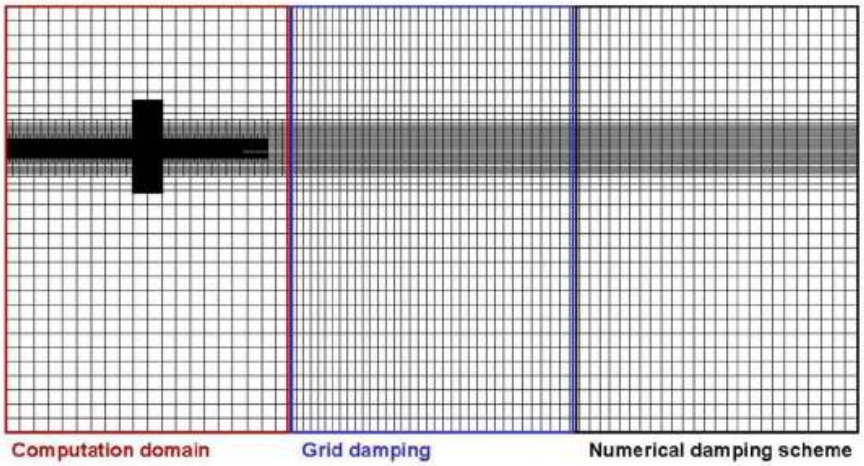


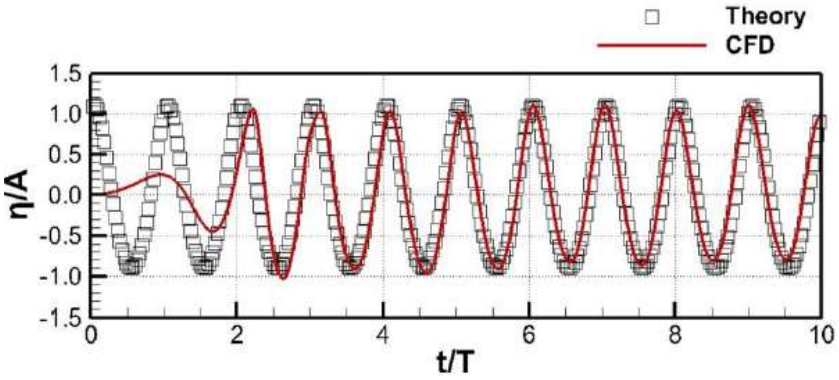
Figure 3. Velocity distributions for v=13m/s; front (left), back (right)

Current/future research – Wave run-up in regular wave

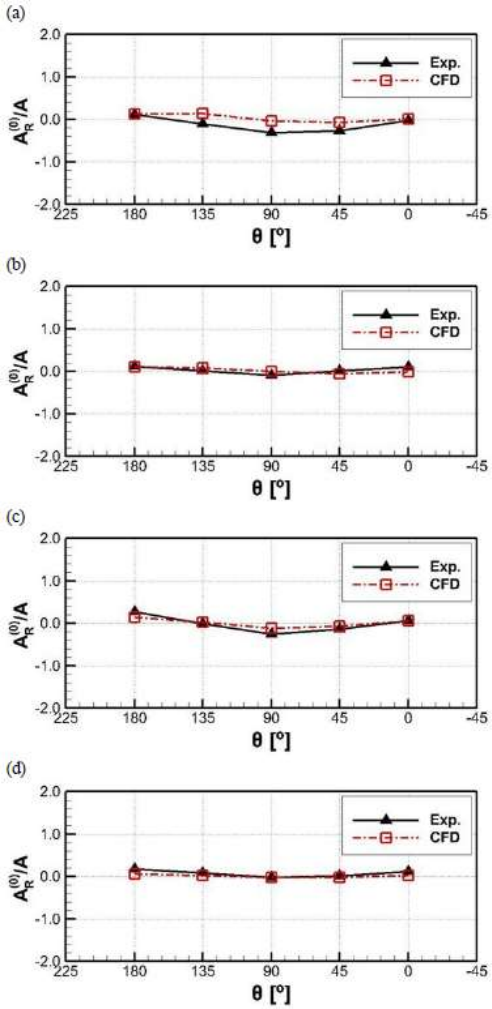




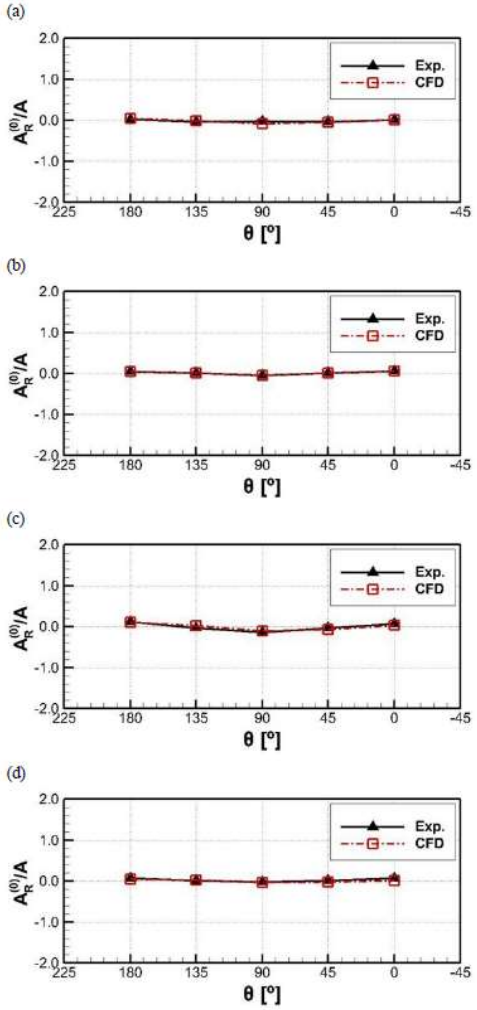




Current/future research – Wave run-up in regular wave



Mean values of wave elevation for single cylinder; (a) Wave probe at the cylinder surface, λ/D=7.9, H/λ=1/30, (b) Wave probe off the cylinder, λ/D=7.9, H/λ=1/30, (c) Wave probe at the cylinder surface, λ/D=7.9, H/λ=1/16, (d) Wave probe off the cylinder, λ/D=7.9, H/λ=1/16.



Mean values of wave elevation for single cylinder; (a) Wave probe at the cylinder surface, λ/D=14.0, H/λ=1/30, (b) Wave probe off the cylinder, λ/D=14.0, H/λ=1/30, (c) Wave probe at the cylinder surface, λ/D=14.0, H/λ=1/16, (d) Wave probe off the cylinder, λ/D=14.0, H/λ=1/16.

Q & A

Thank you