The Specialist Committee on

CFD in Marine Hydrodynamics

Final Report

CFD committee members & meetings

- *Chairman*: Emilio F. **Campana** — *INSEAN-CNR, Italy*
- *Secretary*: Takanori **Hino** — *Yokohama National University, Japan*
  - Peter **Bull** — *QinetiQ, UK*
  - Pablo M. **Carrica** — *IIHR, University of Iowa, USA*
  - Jin **Kim** — *MOERI, South Korea*
  - Sung-Eun **Kim** — *NSWC-CD, USA*
  - Da-Qing **Li** — *SSPA, Sweden*
  - Ning **Ma** — *Shanghai Jiao Tong University, China*
  - Ilkka **Saisto** — *VTT, Finland*
  - Bram **Starke** — *MARIN, The Netherlands*

I. 7 ~ 8 January 2009 — Roma, Italy
II. 8 ~ 9 September 2009 — Iowa City (IA) USA
III. 14 ~ 15 June 2010 — Gothenburg, Sweden
IV. 7 / 11 December 2010 — Gothenburg, Sweden

Structure of the CFD report

QUESTIONNAIRE

PHYSICAL MODELLING

- Free surface
- Turbulence
- Cavitation
- Propulsor

NUMERICAL MODELLING

- Solution algorithms
- Space-time discretization
- Free surface modelling
- Grid generation
- Solution adaptation
- 6DoF and motions
- Verification and validation
- High Perf. Computing

TRENDS IN CFD FOR NAVAL ARCHITECTURE APPLICATIONS

- Resistance
- Propulsors
- Propulsion
- Manoeuvring
- Seakeeping
- Ocean Engineering
- Simulation Based Design

Structure of the CFD report

QUESTIONNAIRE

PHYSICAL MODELLING

- Free surface
- Turbulence
- Cavitation
- Propulsor

NUMERICAL MODELLING

- Solution algorithms
- Space-time discretization
- Free surface modelling
- Grid generation
- Solution adaptation
- 6DoF and motions
- Verification and validation
- High Perf. Computing

TRENDS IN CFD FOR NAVAL ARCHITECTURE APPLICATIONS

- Resistance
- Propulsors
- Propulsion
- Manoeuvring
- Seakeeping
- Ocean Engineering
- Simulation Based Design

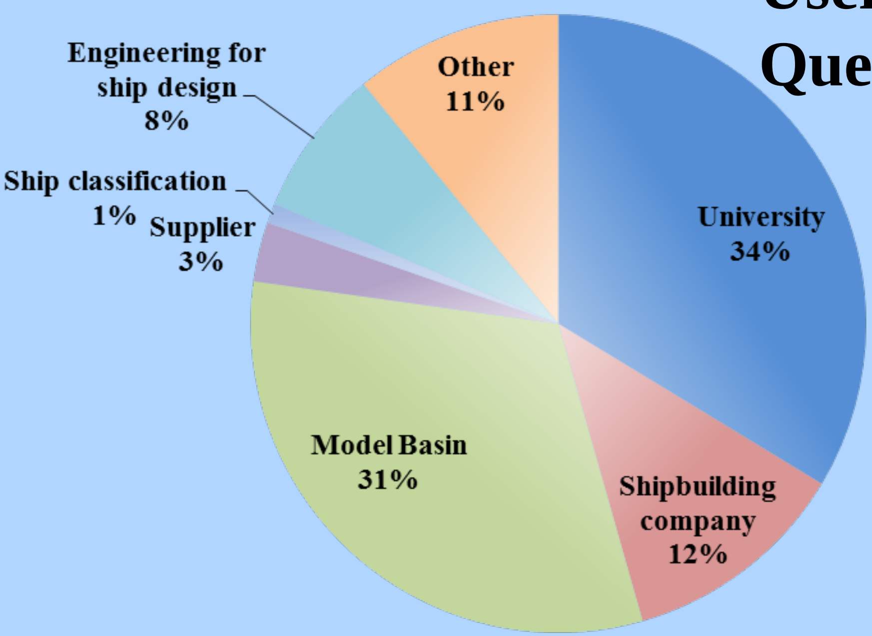
The questionnaire

194 answers (58% ITTC members)

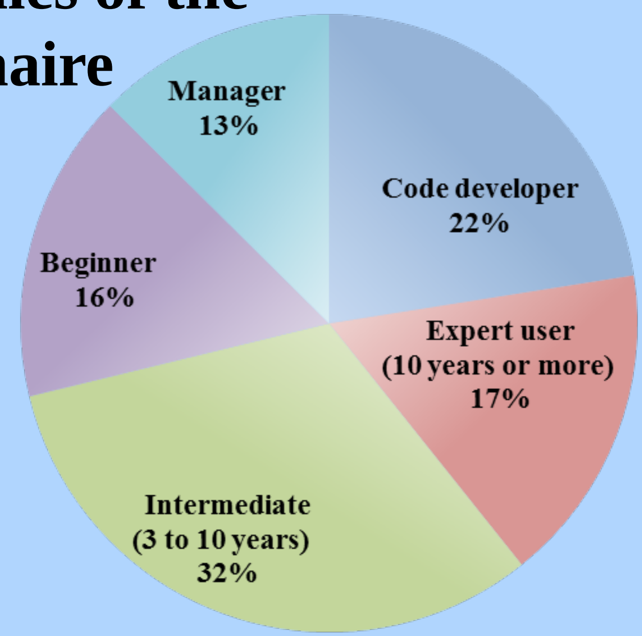
Europe (*45.6%*)
Asia (*30.6%*),
USA (*19.7%*)

ToR: Identify CFD elements of importance to the ITTC from a *user's point of view*, including applicability, accuracy, reliability, time and cost.

User Profiles of the Questionnaire

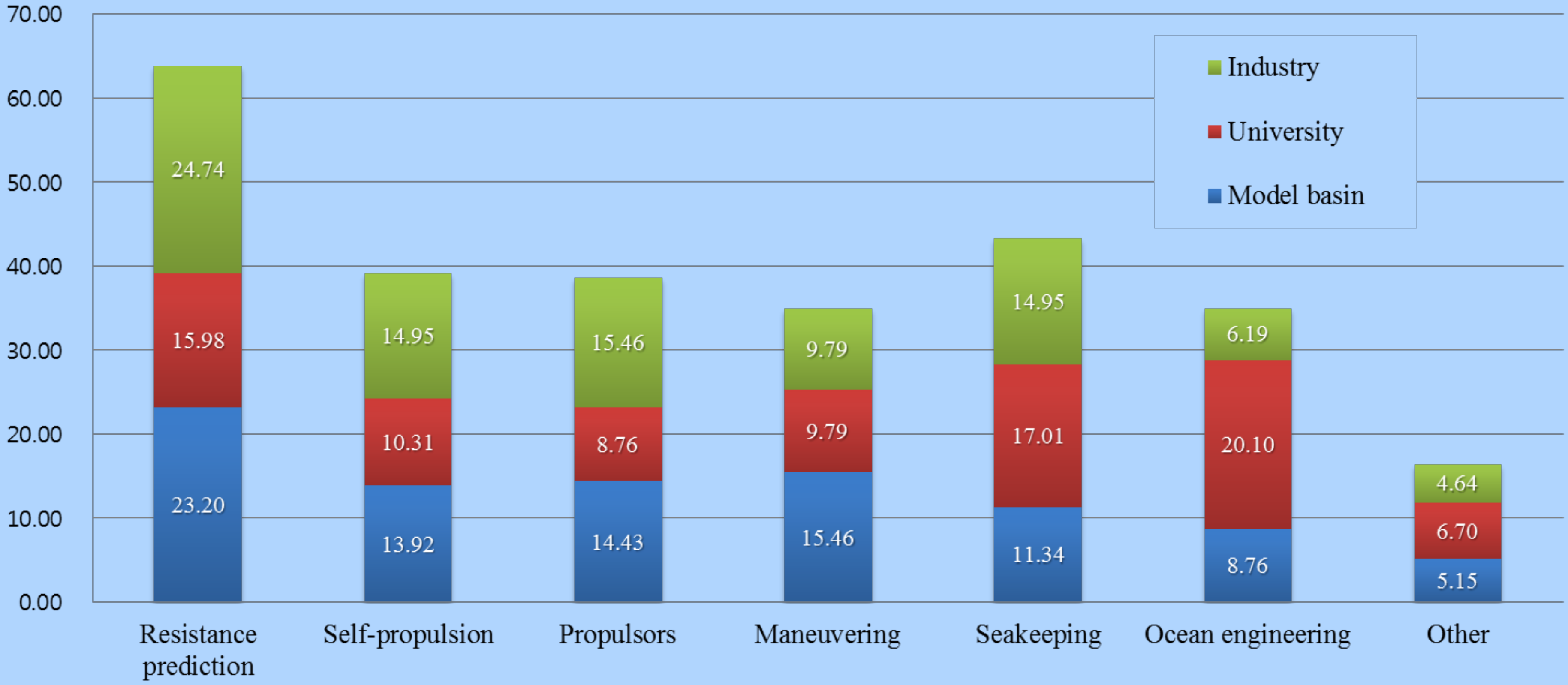


Institutional Distribution

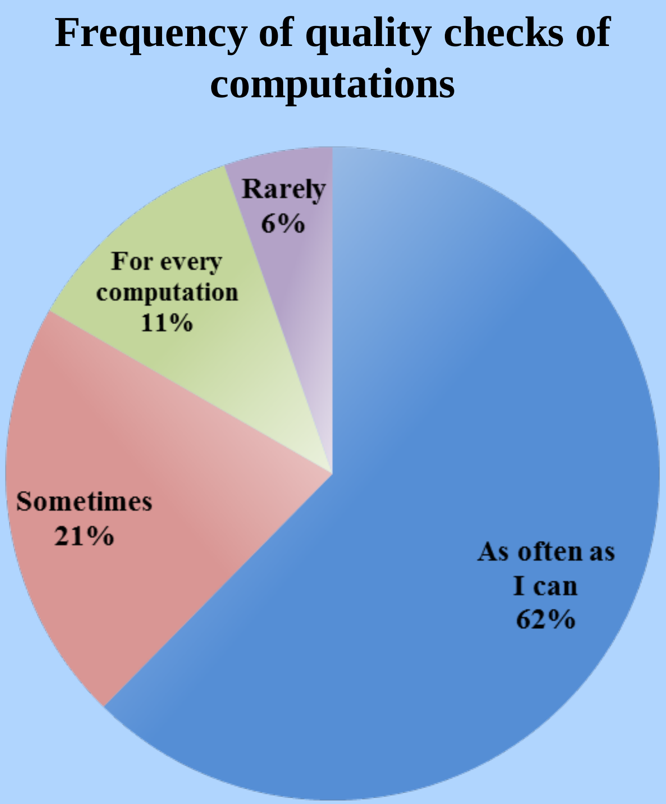


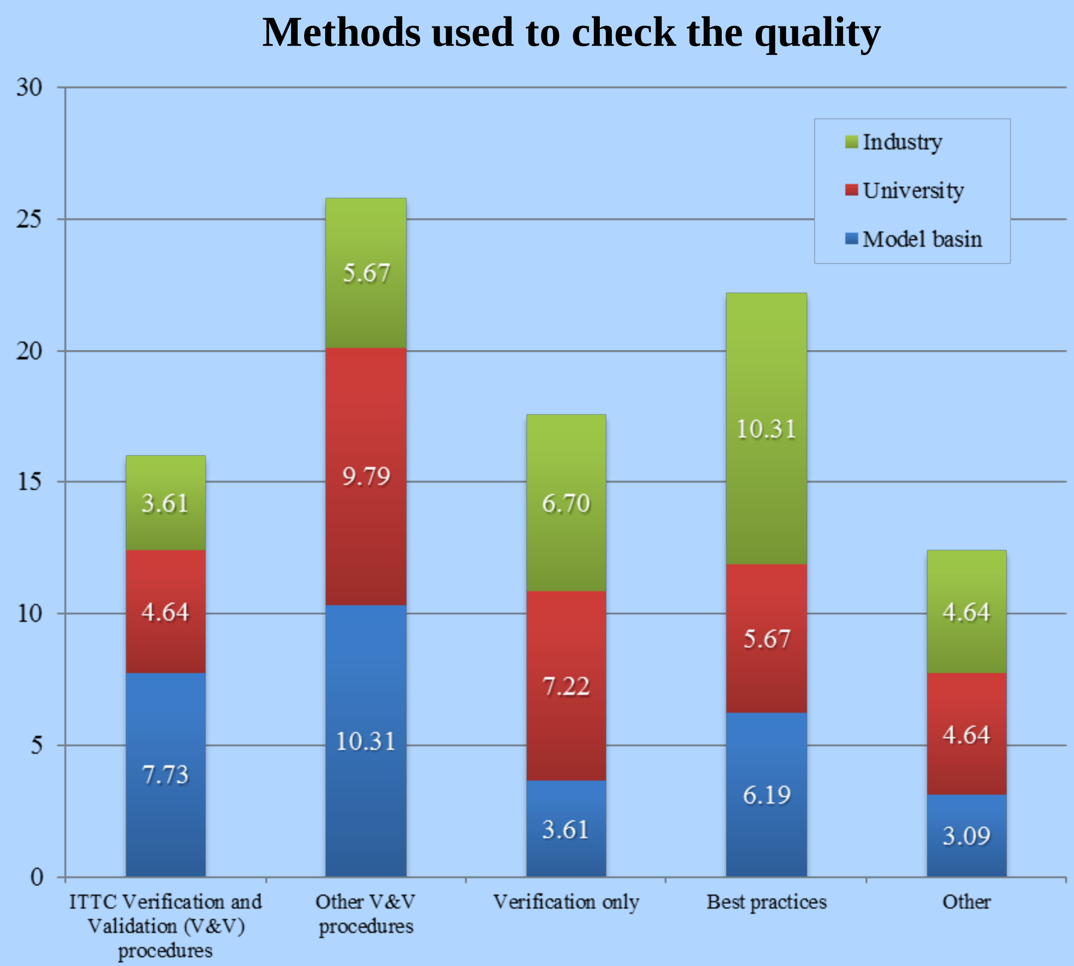
Experience in CFD Applications

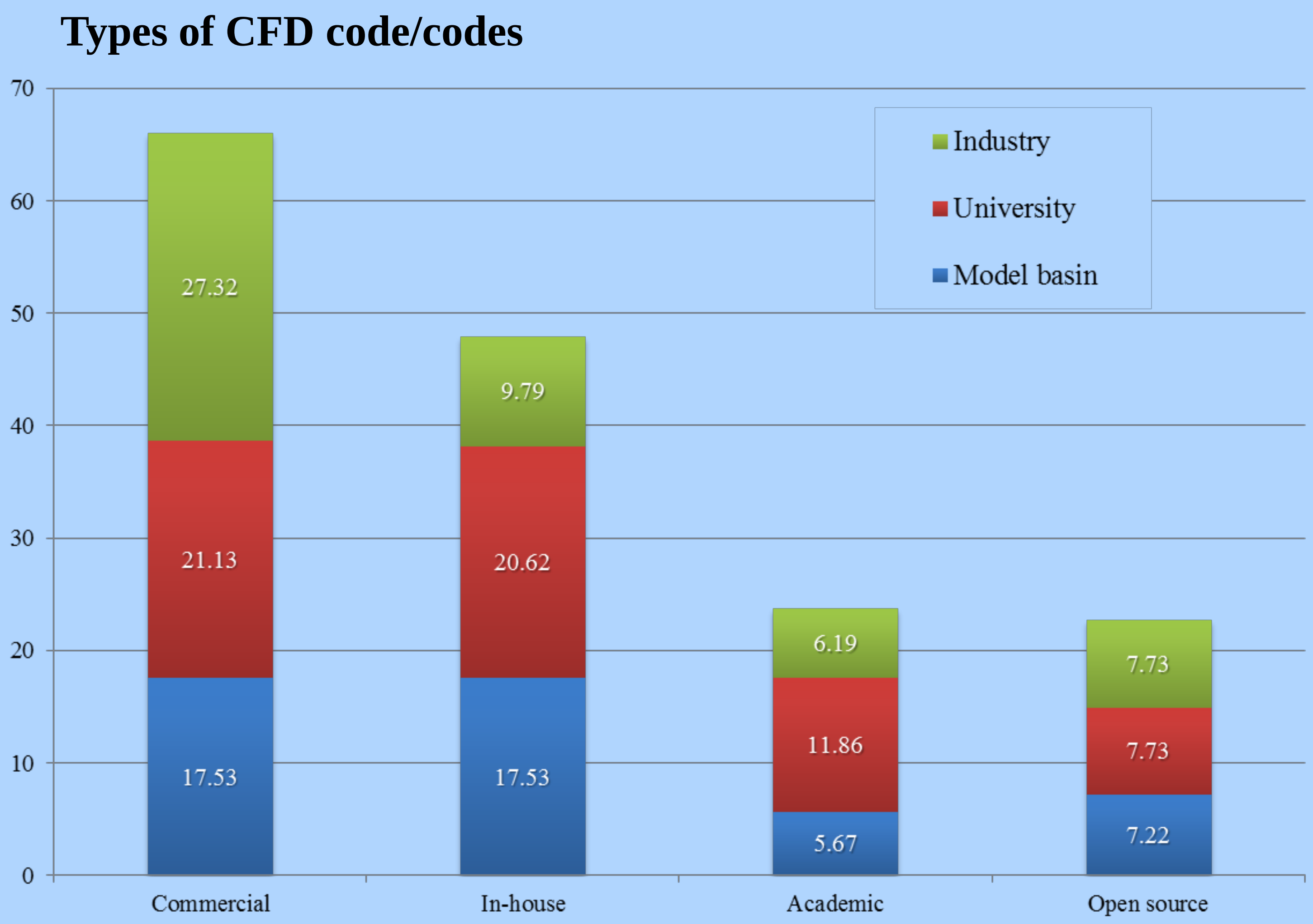
CFD applications in Marine Hydrodynamics (Multi-Choices)



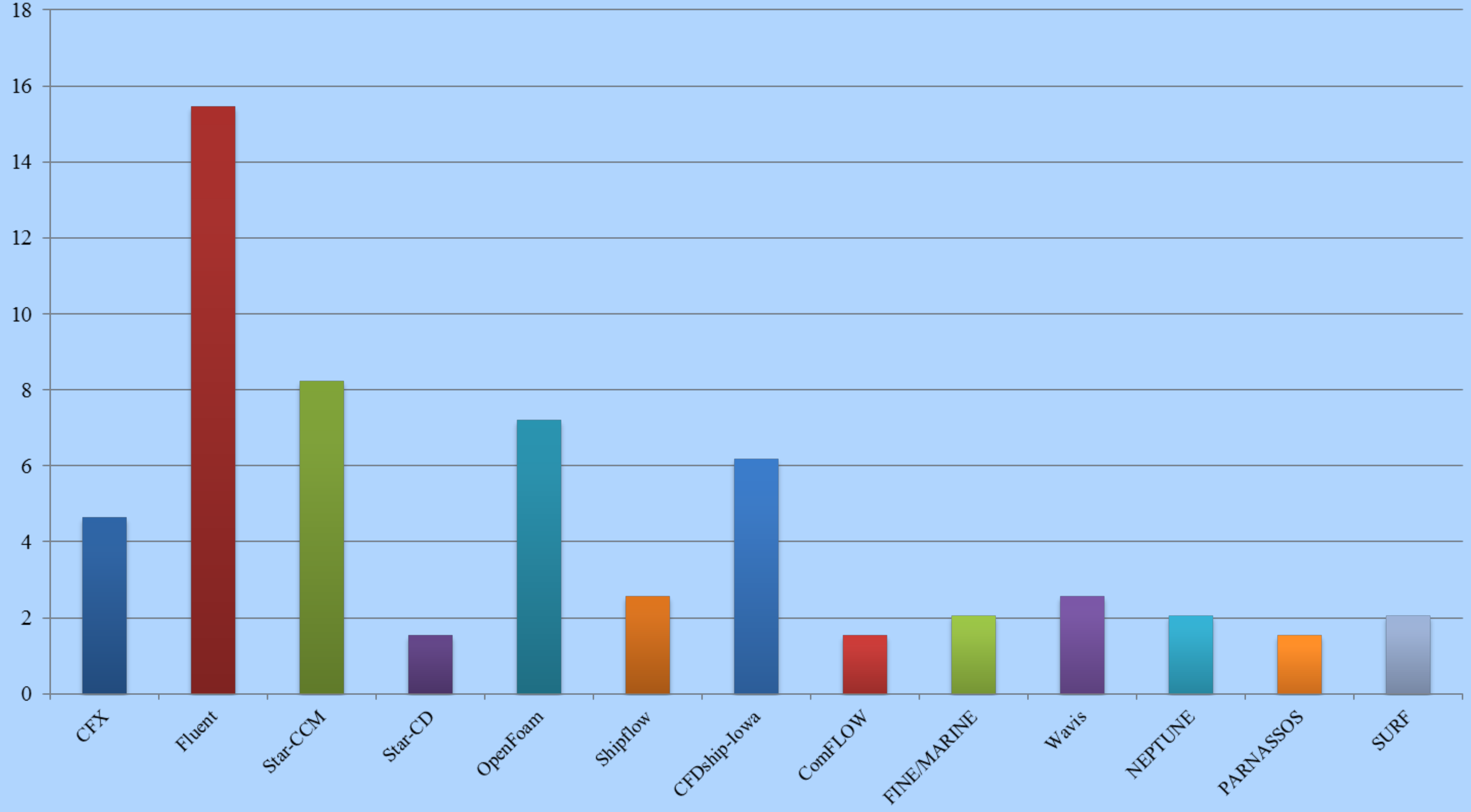
Quality Check (V&V)



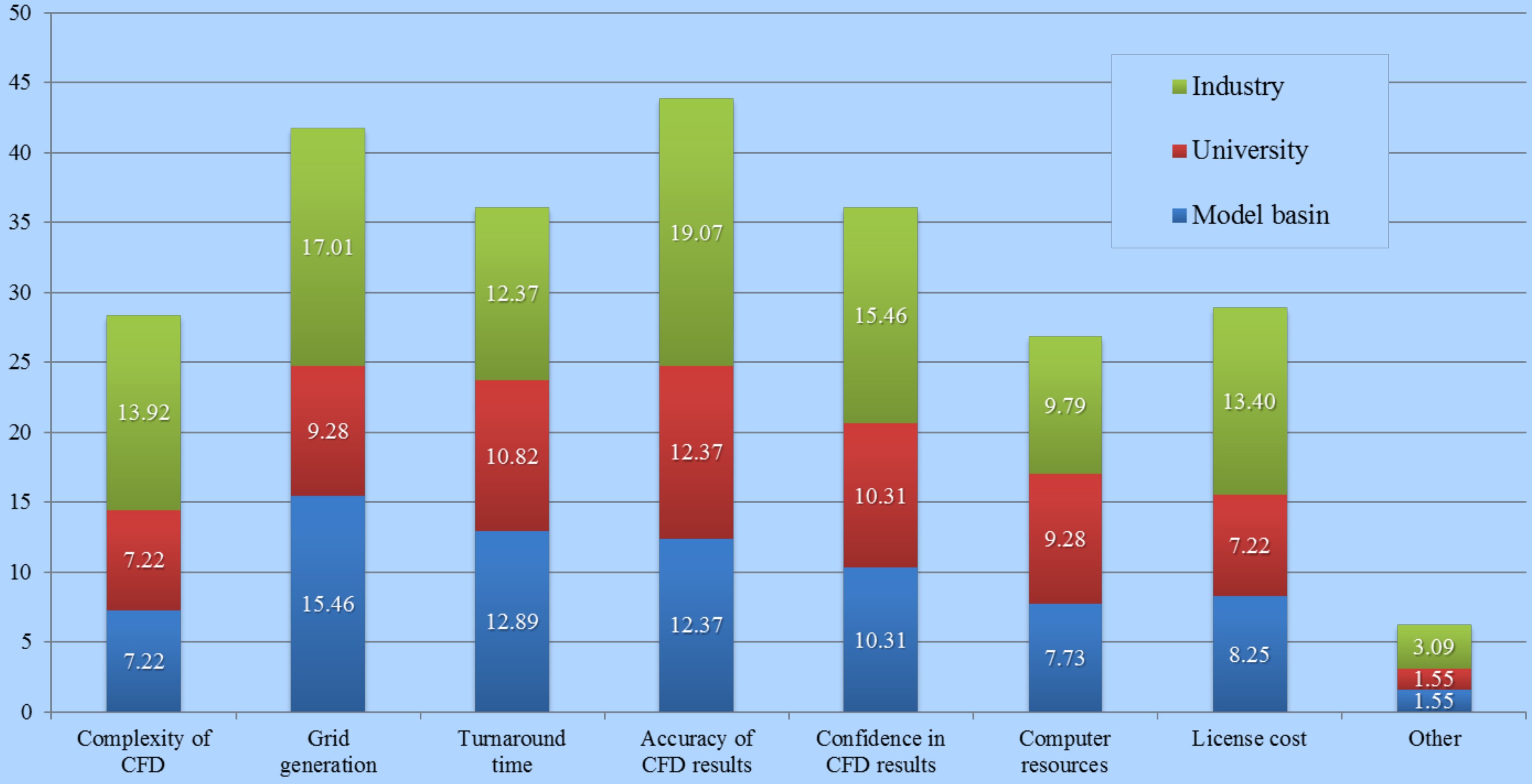



Types of CFD code/codes
70
60
50
40
30
20
10
0
Industry
University
Model basin
27.32
21.13
17.53
9.79
20.62
17.53
6.19
11.86
5.67
7.73
7.73
7.22
Commercial
In-house
Academic
Open source

CFD code/codes – Kind of Codes (42 codes)



The main difficulty in using CFD for work



Structure of the CFD report

QUESTIONNAIRE

PHYSICAL MODELLING

Free surface
Turbulence
Cavitation
Propulsor

The area of CFD where different numerical techniques for physical processes are described as 'models' for the particular process.

Free surface modelling

Free Surface Boundary Conditions to bulk flow (NS) eqs.

- ***Kinematic*** condition : Fluid particles remain on the surface.
  Define the shape of the surface.

- ***Dynamic*** condition : Stresses are continuous across the surface.
  Surface tension neglected.
  Determine pressure and velocity gradients.

Additional modeling:

Wave breaking

Cointe & Tulin ’94

Rhee & Stern ‘02

Muscari & Di Mascio ’04

Incident & ambient waves

Benchmark data from CFD workshops

Gothenburg 2010

Tokyo 2005

Turbulence modelling

Turbulence modeling aims to model the effect turbulent motion of the flow has on the mean flow.

A vast number of turbulence models has been proposed over the last decades, <u>but no ‘universally’ valid model exists.</u>

Thus one is forced to choose the best model available for a specific application.

There is a <u>hierarchy</u> of turbulence models, with <u>increasing complexity and expected physical accuracy</u>.

- **1** and **2**-equation models of increasing complexity *(e.g. from Spalart-Allmaras to k- ε and k- ω and modifications)*

- **Reynolds stress** models, that can give accurate predictions of e.g. resistance, wake fields and the possible occurrence of flow separation.

These models are, however, known to fail in largely separated flows, and there more complex (and time consuming) transient models have to be used

- *Large Eddy Simulation **(LES),***
- *Detached Eddy Simulation **(DES)*** ,
- *Delayed Detached Eddy Simulation **(DDES)***

Cavitation modelling

- Interface Tracking: a distinct interface to separate vapor from fluid domain, determined by kinematic and pressure conditions. (For steady attached sheet cavitation and inviscid flow)
- Discrete bubble dynamics: cavitation as an interaction between bubble nuclei and pressure field variation. Bubble size governed by Rayleigh-Plesset Eq. (For inception, travelling bubble, nuclei effects. A Lagrangian-Eulerian approach)
- Interface capturing: assumes the flow is a mixture of multi-phases. Uses a flow solver and a cavitation model to determine the vapor volume of fraction.
  - **Approach 1**: Homogeneous equilibrium mixture (HEM) of 2-3 component phases in one-fluid => one-set of RANS Eqs.
    (the most popular approach for unsteady cavity, cloud shedding and collapse)
  - **Approach 2**: Non-equilibrium mixture of *n*-phases
    => Each phase is solved by its own set of N-S Eqs with additional transfer terms to account for phase transition and interaction (For study of dynamic interaction between phases, surface tension etc)

Cavitation models for interface capturing

1. Barotropic isothermal models, **ρ**(*mix density*) = ***f*** (**p** *static pressure*)
   – e.g. Dalannoy & Kueny (1990)
2. Transport equation-based model (TEM or VOF) using source terms to account for mass transfer between phases,
   – Model relating source terms to bubble dynamics, e.g. Kubota et al. (1992), Sauer et al. (2000), Singhal et al.(2002)
   – Model using fully empirical source terms, e.g. Merkle (1998), Kunz et al. (2000)
3. Thermodynamic equilibrium models using Equations of State (EOS) (for liquid, vapor and mixture phases respectively)
   - e.g. Saurel et al. (1999), Schmidt et al. (2006, 2009), Koop (2009)

Cavitation Erosion prediction

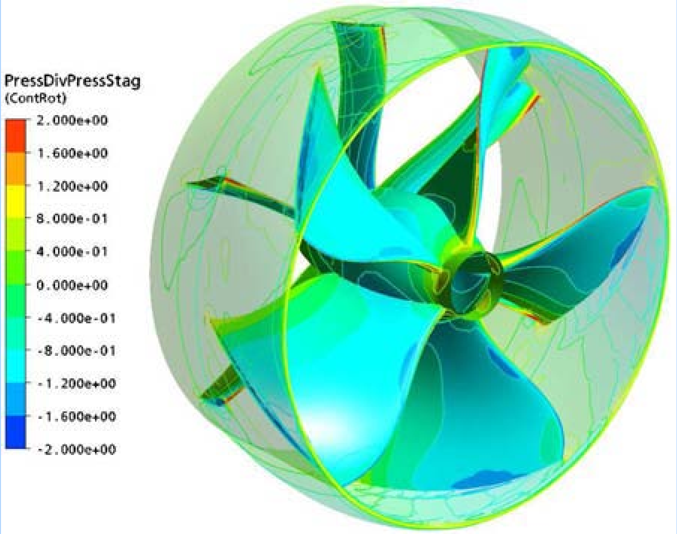
No major novelties since 25th ITTC

1. Micro-scale bubble dynamics to estimate the impact pressure, Fukaya et al. (2006) and Ochiai et al. (2009)
2. Model the relationship between the fluctuation of the void fraction and the occurrence of erosion, Dular et al. (2006)

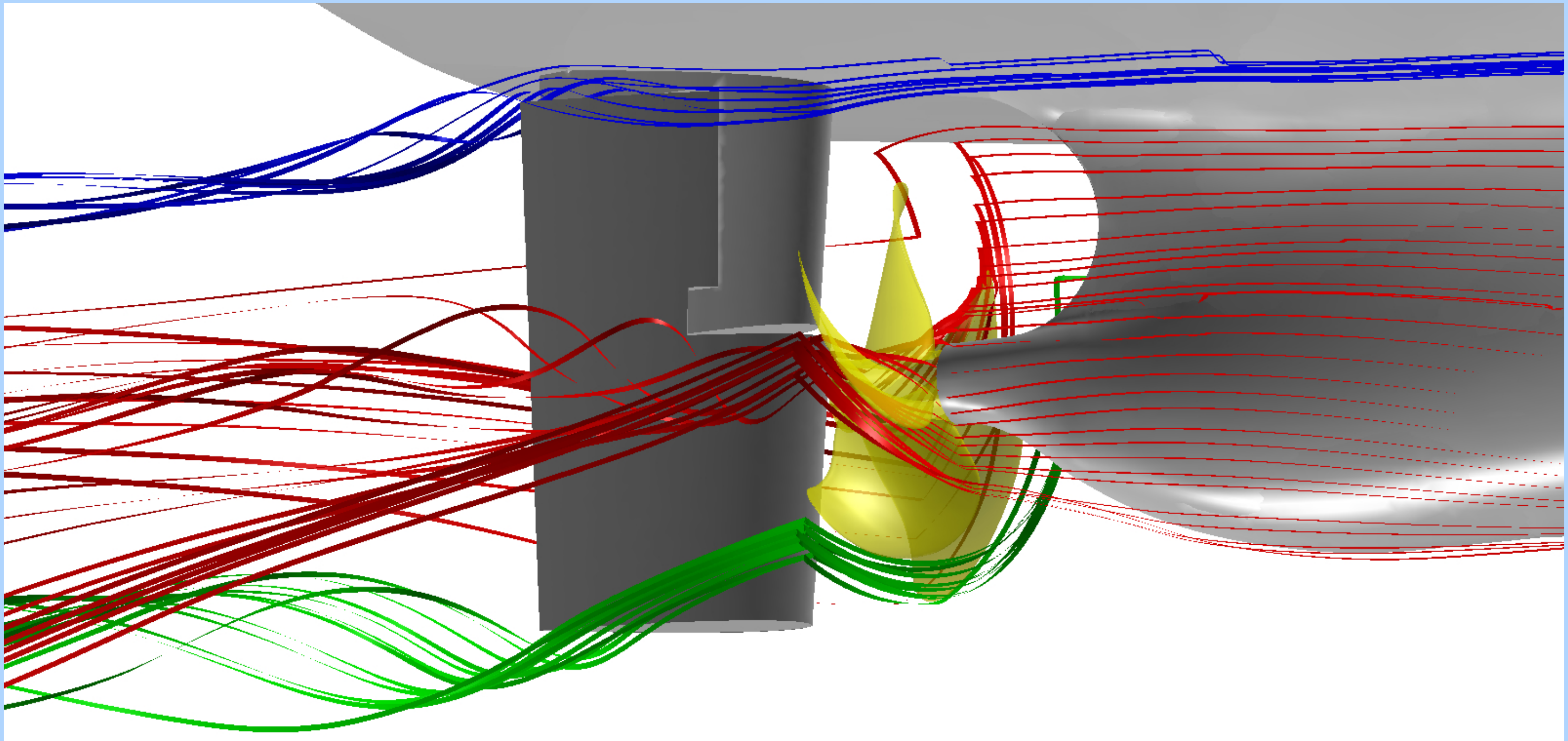
Propulsor modelling

Geometric models

- Shape of the propeller defined by a local grid
- Rotation defined by rotating reference frames
- 'Open water' propellers use a single blade with periodic conditions
- Transient interactions between the ship hull and the propeller update the rotational position every time step
  - Surface interpolation on sliding interfaces
  - Volume interpolation between overlapping grids
- Simplifications for steady state
  - Freeze the propeller at a given position
  - Circumferentially average around the propeller



Propulsion example



KCS hull with time frozen propeller and rudder deflection

Propulsor modelling

Approach 2: Body Force

- Influence of the propeller modelled by body forces or momentum sources
- Body forces constructed to integrate to the required thrust and torque
- Range of methods to define the thrust and torque distributions
  - Algebraic polynomials
  - Lifting line
  - Boundary element
  - RANS
- Coupling between the propeller and the ship hull evaluated as the ‘effective wake’

Structure of the CFD report

QUESTIONNAIRE

PHYSICAL MODELLING

- Free surface
- Turbulence
- Cavitation
- Propulsor

NUMERICAL MODELLING

- Solution algorithms
- Space-time discretization
- Free surface modelling
- Grid generation
- Solution adaptation

6DoF and motions

Verification and validation

High Perf. Computing

Solution Algorithms

For ship hydrodynamics applications, the fluid (fresh or sea water) is considered "incompressible". A special treatment of continuity (mass conservation) equation is needed.

Artificial Compressibility Method

-Incompressibility enforced using the concept of "artificial compressiblity" (Chorin, 1967) that can be viewed as a special case of pre-conditioned compressiblle flow formulation
-Can take advantage of solution algorithms developed for compressible gas dynamics
-SURF (Hino, 1998), Tenasi (Briley et al., 2006) are examples of this method.
-Both steady and unsteady formulations are available.

Projection Method

-Pressure used as a constraint to enforce "divergence-free" velocity field
-Involves a "projection" of velocity field on to a divergence-free vector-space giving a pressure equation (Harlow and Welch, 1965).
-SIMPLE family, PI
-Typically uses a sequential (segregated) solution process
-Coupled solvers based on projection method exist
-CFDSHIP-IOWA, NAvyFOAM, and the majority of commercial codes

Other Methods

-Fully coupled formulation

Space-Time Discretization

One of the major issues in CFD. It determines not only accuracy but also stability (robustness) of numerical solutions

Spatial Discretization

-Finite-volume method (FVM) is the most widely used.

-Modern FVMs can take arbitrary polyhedral unstructured grids

-2nd-order FVM is the workhorse for industrial applications,

-Higher-order FVM/FDM exists and show a better spatial accuracy yet limited to structured grids

-Interface-capturing using VOF requires a special advection scheme.

Temporal Discretization

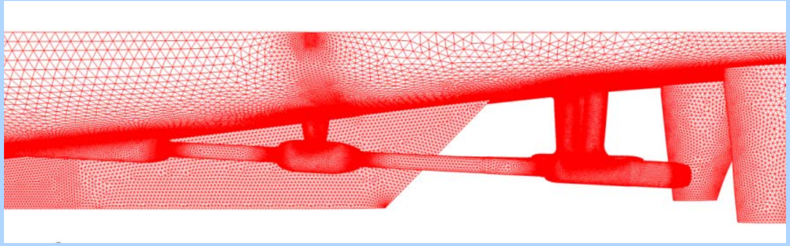
-Implicit time-marching is widely used for ship hydrodynamics due to large time-step size it allows

-Use of explicit time-marching scheme is rational only for LES and DNS

-First-order backward Euler scheme is often used when pursuing steady-state solutions

-The survey of G2010 workshop shows that, for time-accurate solutions, second-order schemes (Crank-Nicolson, three-level backward Euler) are the popular choices.

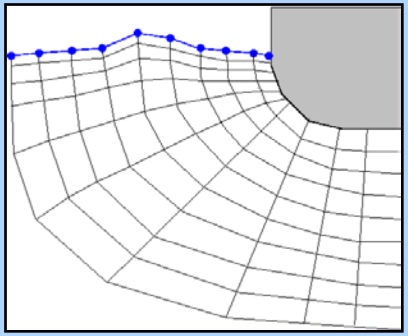
-4th -order Runge-Kutta scheme has been seen but rarely used.



Free surface numerical modelling

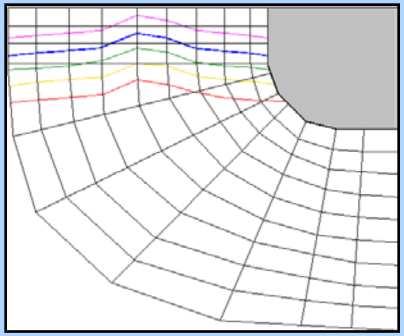
- **Interface Fitting**

  Grid lines fitted to surface

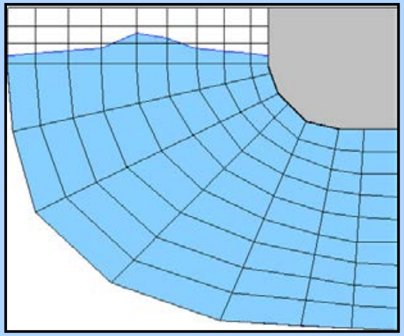


- **Interface Capturing**

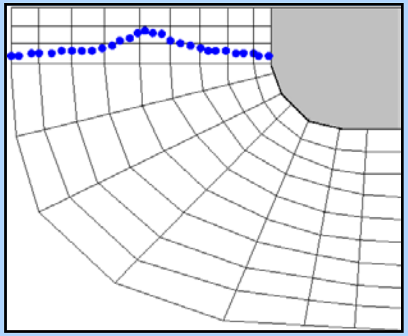
  Levelset method | VOF method | MAC method



distance fn.



volume fraction



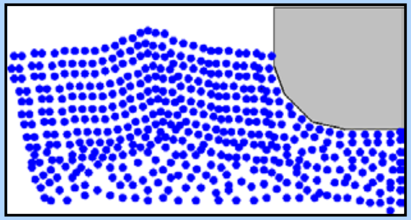
marker particles

Two-phase flow approach : solve both water and air

One-phase flow approach : solve water only

Free surface numerical modelling

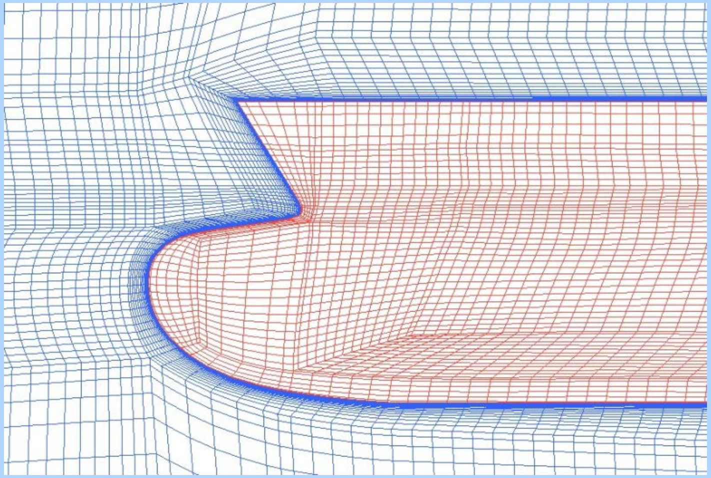
- Patricle method
  - SPH
  - MPS

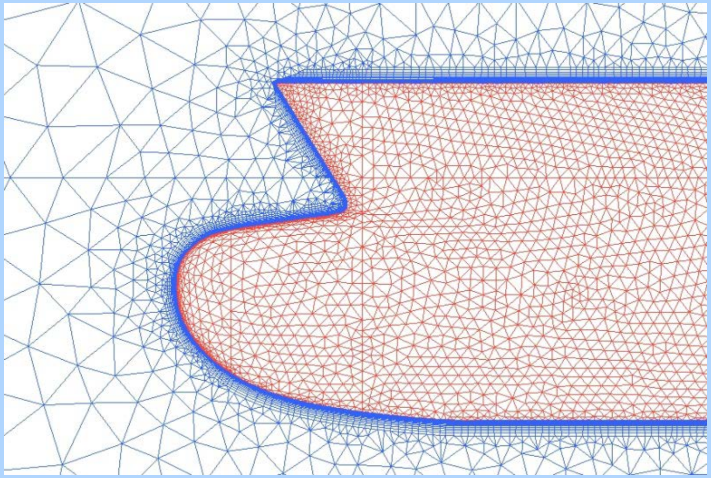


Methods	Interface Fitting	Interface Capturing	Particles
Advantages	Accurate BC	Large deformation	No grid required
Disadvantages	Re-gridding required	Approximated BC, Specialized Scheme	Force estimation

Grid generation

- Computational cells to resolve fluid flow parameters
  - Tetrahedral, prism, hexahedral and polyhedral
- Methods to define the computational cells
  - Cartesian
    - Octree based cut cell
    - Inflation layers to capture boundary layer
  - Structured body fitted
    - Single block, multi block
    - Smoothing techniques to improve 'grid quality'
  - Unstructured
    - Octree, Delaunay and advancing front point insertion methods
    - Inflation layers to capture boundary layer
- Interpolate between different grids for non-conforming grid points
  - Volume interpolation using overset and overlapping
  - Surface interpolation using interfaces





Solution adaptation

- Localised refinement to obtain a more accurate fluid flow
  - *h*-refinement modifies the grid
  - *p*-refinement modifies the solution process
- Adaptation markers used to identify regions in space (and time) where flow solution requires refinement
  - Geometric description
  - Solution markers
  - Solution gradients
  - Error estimators
- Grid refinement
  - Grid point insertion - Increases the number of cells in the adaptation region to reduce the flow errors
  - Grid point movement - Reduces the grid spacing in the adaptation region to reduce the flow errors
- Solution refinement
  - Increases the order of accuracy of the local solution algorithm – 1st- 2nd order numerical scheme increased to 3rd,4th or 5th order numerical scheme

6 DoF and motions

Motions are needed to compute:

- Attitude (sinkage and trim)
- Self-propulsion (in some cases)
- Seakeeping
- Pitch and heave
- Stability
- Roll decay
- Maneuvering
- Ship-ship interaction, etc.

HIGHLY DESIRABLE CFD CODE CAPABILITY

Motions and 6DoF approaches:

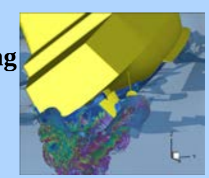
- Ship or earth fixed coordinate frame

- Direct or fluid momentum balance force integration

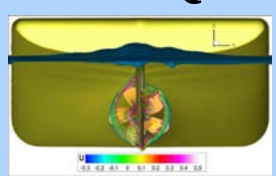
- Grid motions: fixed grids, deformable grids, Sliding grids, overset, regridding, local grid refinement, immersed boundary

- 6DoF solver: Euler angles or Quaternions, Implicit or explicit

ONR Tumblehome broaching



KCS self-propulsion



Verification and Validation

Basic assumption: have a set of CFD solutions that are in or enough close to the **asymptotic range**. Then use methods based on Richardson extrapolation for the spatial discretization error (modeling the error as a low order polynomial in the discretization parameter)

Problems:

(i) all the solutions must be close to the asymptotic range (otherwise the estimated order of accuracy p_{RE} approaches the theoretical order p_{th} with oscillations) and
(ii) require **3 or more refined high-quality grids** (often too expensive for industrial applications).
(iii) Oscillatory convergence, for which Richardson extrapolation cannot be used
(iv) Complex geometries, e.g. prohibitively high grid resolution requirements
(v) Overlapping and unstructured grids.

Alternatives

Method of Manufactured Solutions (MMS)

Difficulties in setting up manufactured solutions for turbulence quantities in 1- and 2-equation eddy viscosity models

High Performance Computing

Ship computations getting bigger

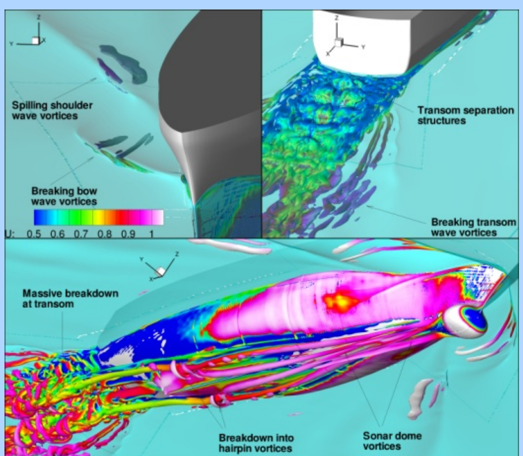
-10~30 million grid points standard

-300+ million on curvilinear grids have been performed

- 10+ billion on Cartesian grids have been demonstrated

DTMB 5512 Forward speed diffraction with 115 million points

Animation DTMB 5512 pitching and heaving in regular waves (70 million points)



Weak scalability achievable for incompressible codes => billions of points foreseeable in the near future

- OK for analyzing flow physics, local problems

- Too expensive, complex and slow for naval architecture design

Strong scalability more difficult

- Superfast computations with relatively small problems (~3 million points) unlikely in the near future

- Modest speed ups can be expected, about an order of magnitude every 5 years.

Structure of the CFD report

QUESTIONNAIRE

PHYSICAL MODELLING

- Free surface
- Turbulence
- Cavitation
- Propulsor

NUMERICAL MODELLING

- Solution algorithms
- Space-time discretization
- Free surface modelling
- Grid generation
- Solution adaptation
- 6DoF and motions
- Verification and validation
- High Perf. Computing

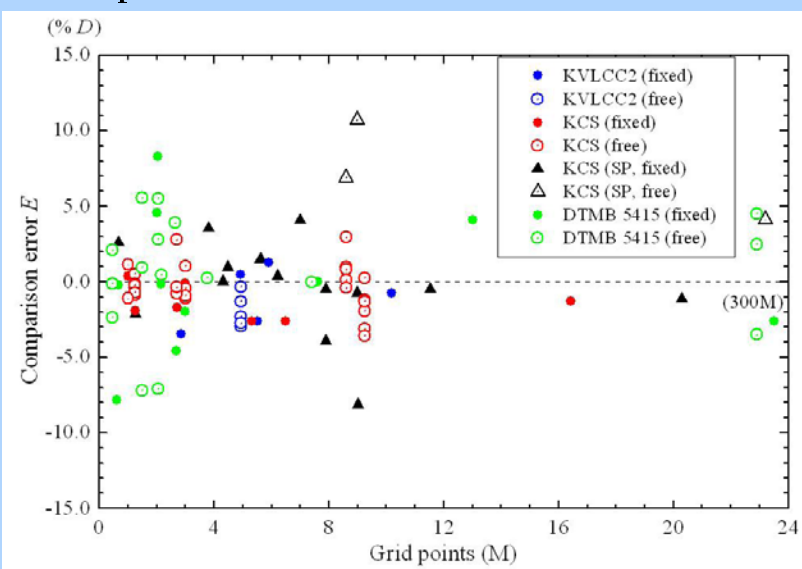
TRENDS IN CFD FOR NAVAL ARCHITECTURE APPLICATIONS

- Resistance
- Propulsors
- Propulsion
- Manoeuvring
- Seakeeping
- Ocean Engineering
- Simulation Based Design

Resistance

Overall Assessment of Resistance Prediction capability from G2010

- The survey showed that the statistical variance (scatter) of all submitted predictions is considerably smaller than those reported from the previous workshops.



From the G2010 - the preliminary report (Larsson et al.)

-The majority of the (better) predictions seem to be within a few per cents from the measurement for all cases (KVLCC2, KCS, DTMB 5415) when adequate mesh resolution are used.

Trends in Resistance Prediction

-High-performance computing with several to tens and hundreds of millions of elements

-Advances in gridding techniques such as arbitrary polyhedral (unstructured) mesh, adaptive mesh refinement (AMR), and overset grids

-Increasingly popular use of FVM on unstructured grids

-Two-equation based EVM turbulence modeling, with further improvements shown by EARSM and RSTM

-An increasing number of contributions resolving viscous sublayer

-Industry (shipyards) seems to have benefitted from wall function approach.

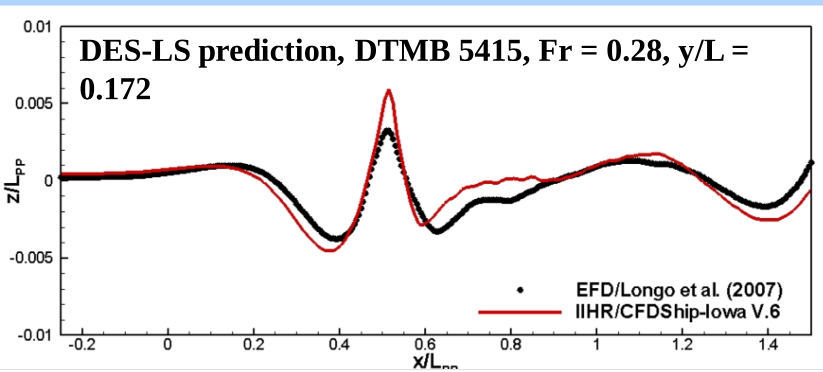
26th International Towing Tank Conference
Rio de Janeiro, Brazil, 28 August - 3 September, 2011

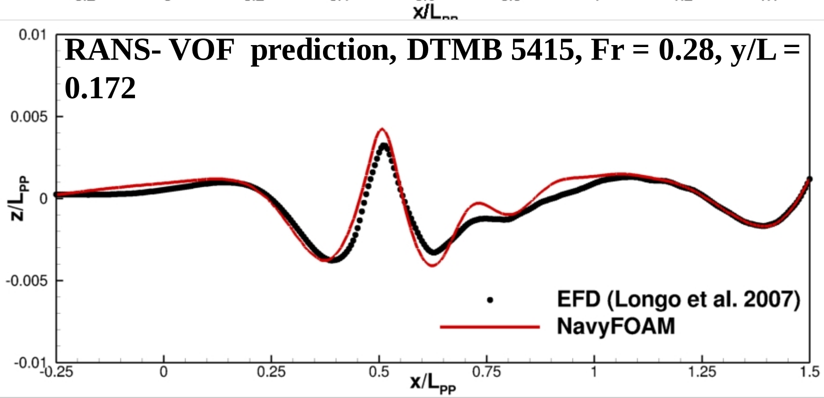
ITTC 2011 Rio de Janeiro

Wave Pattern

-At G2010 workshop, contributions are equally split between volume-of-fluid (VOF) and level-set (LS) methods.

-The quality of VOF predictions seems largely on par with that of LS predictions.





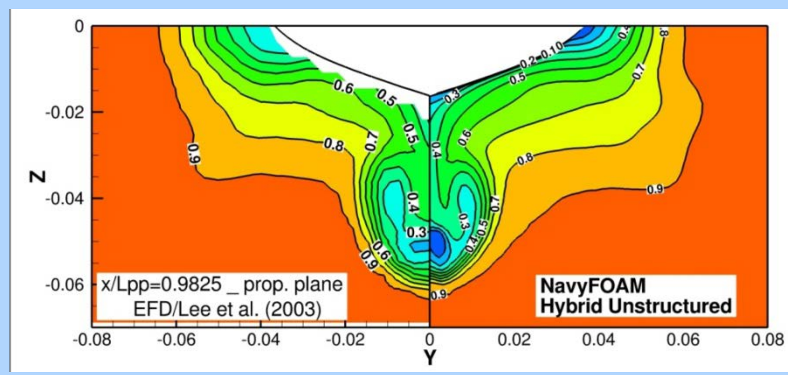
Local Flow Fields

-G2010 workshop showed improvements over the previous years with a smaller scatter among the results for wake predictions (e.g., KVLCC2)

-The main driver of the improvements is the ever-increasing grid resolution and use of advanced turbulence models (EARSM, RSTM)

-LES and DES haven't really shown advantage other than the predicted features are all grossly exaggerated.

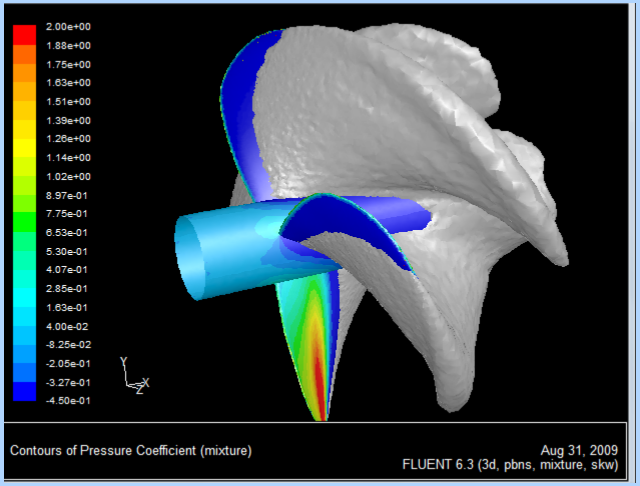
-Efficacy of unstructured grids has been demonstrated.

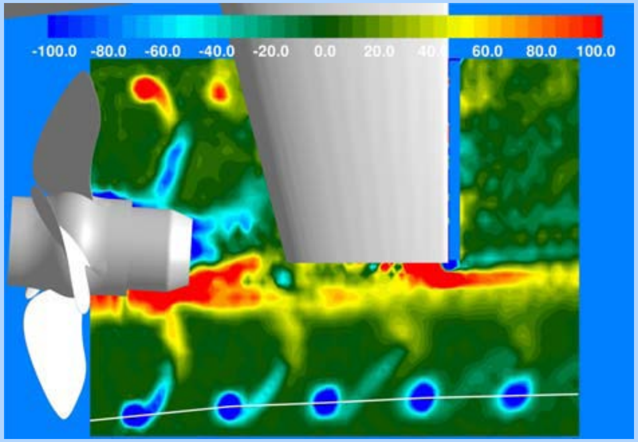


RANS prediction on a unstructured grid of the contours of axial velocity at a stern plane for KVLCC2.

Propulsors

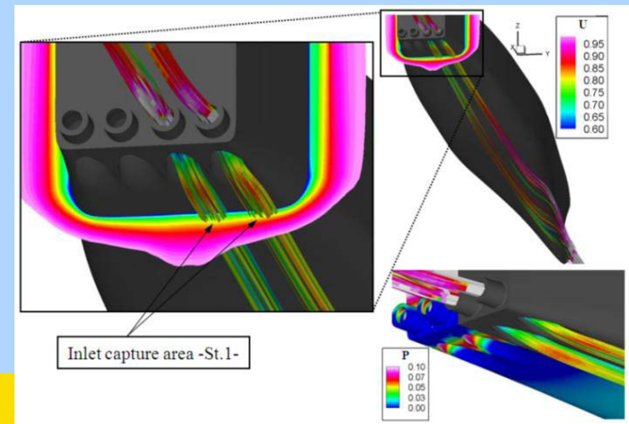
- Open water propellers
  - Basic thrust and torque performance characteristics regarded as routine and well established
  - Scale effects due to Reynolds number and transition are being investigated
  - Cavitation performance ongoing research
- Operating propellers behind ship and with shaft/brackets and rudders
  - Propulsion characteristics provided using momentum sources/actuator discs
  - Propeller geometry using rotating reference frames and grid overlaps or interfaces
  - Influence of wake equalisation devices and vortex generators to improve propeller inflow

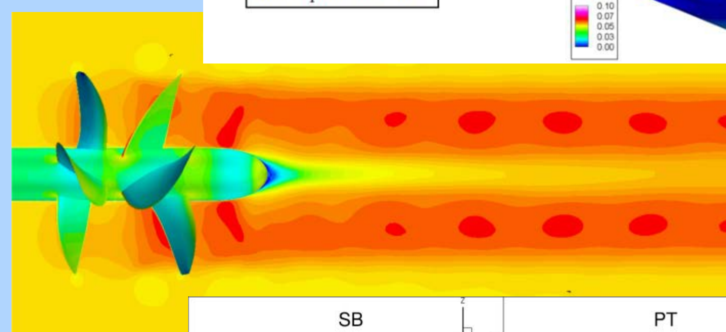


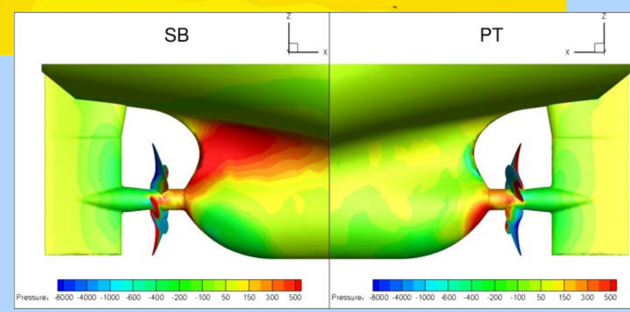


Propulsors

- Waterjets
  - Axial WJ1 and 2 being used to provide detailed validation cases
  - Design of high speed craft
  - Design of propeller, duct and stator
- Podded propulsors
  - Design of fillets and support struts
- Ducted propulsors
  - Development of nozzle designs and associated Reynolds number scaling
  - Bow thrusters
- Interaction effects between ship hull and appendages
  - Influence of wake equalisation devices and vortex generators to improve propeller inflow
  - Propulsion and appendage configurations





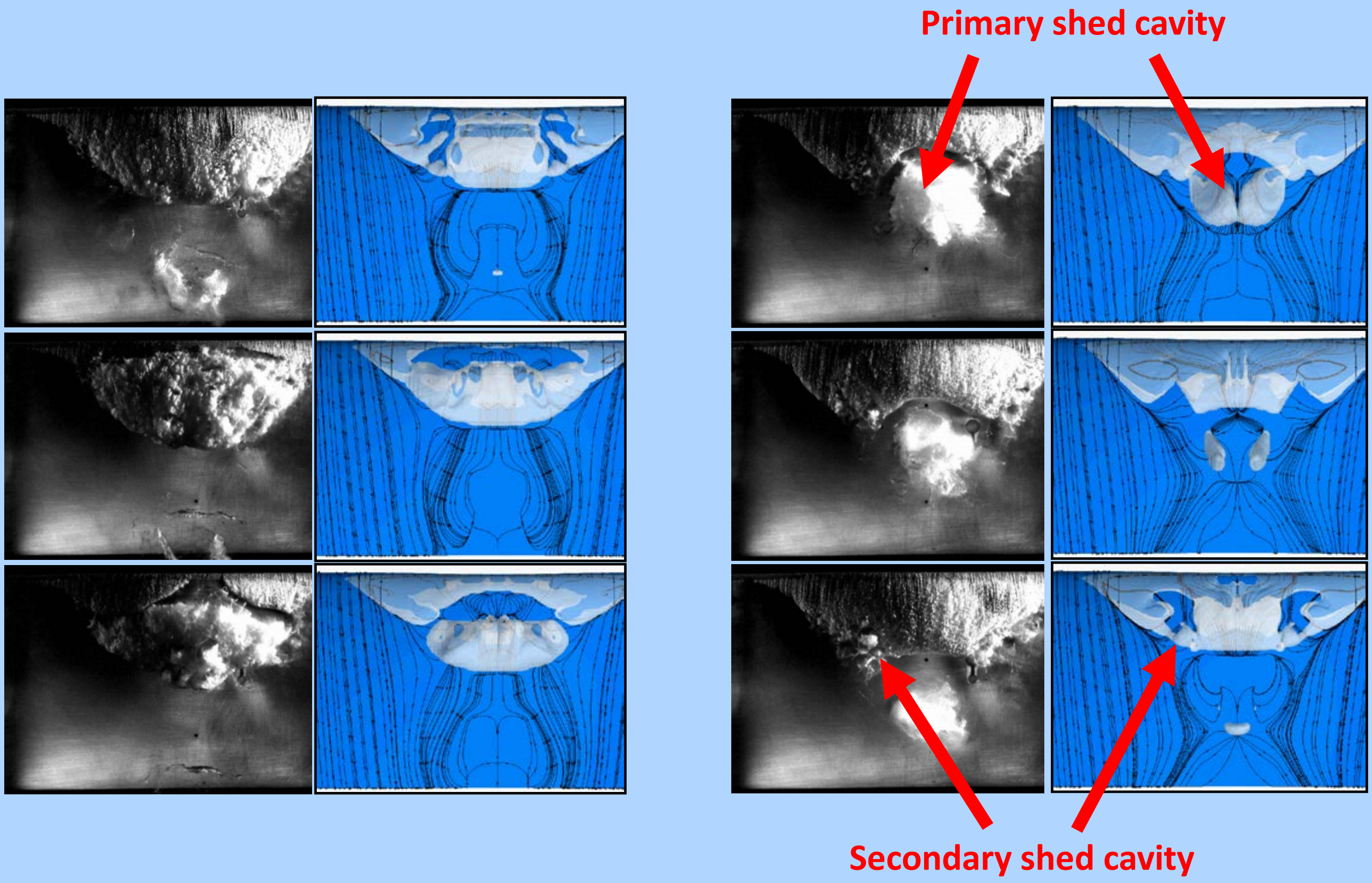


Cavitation

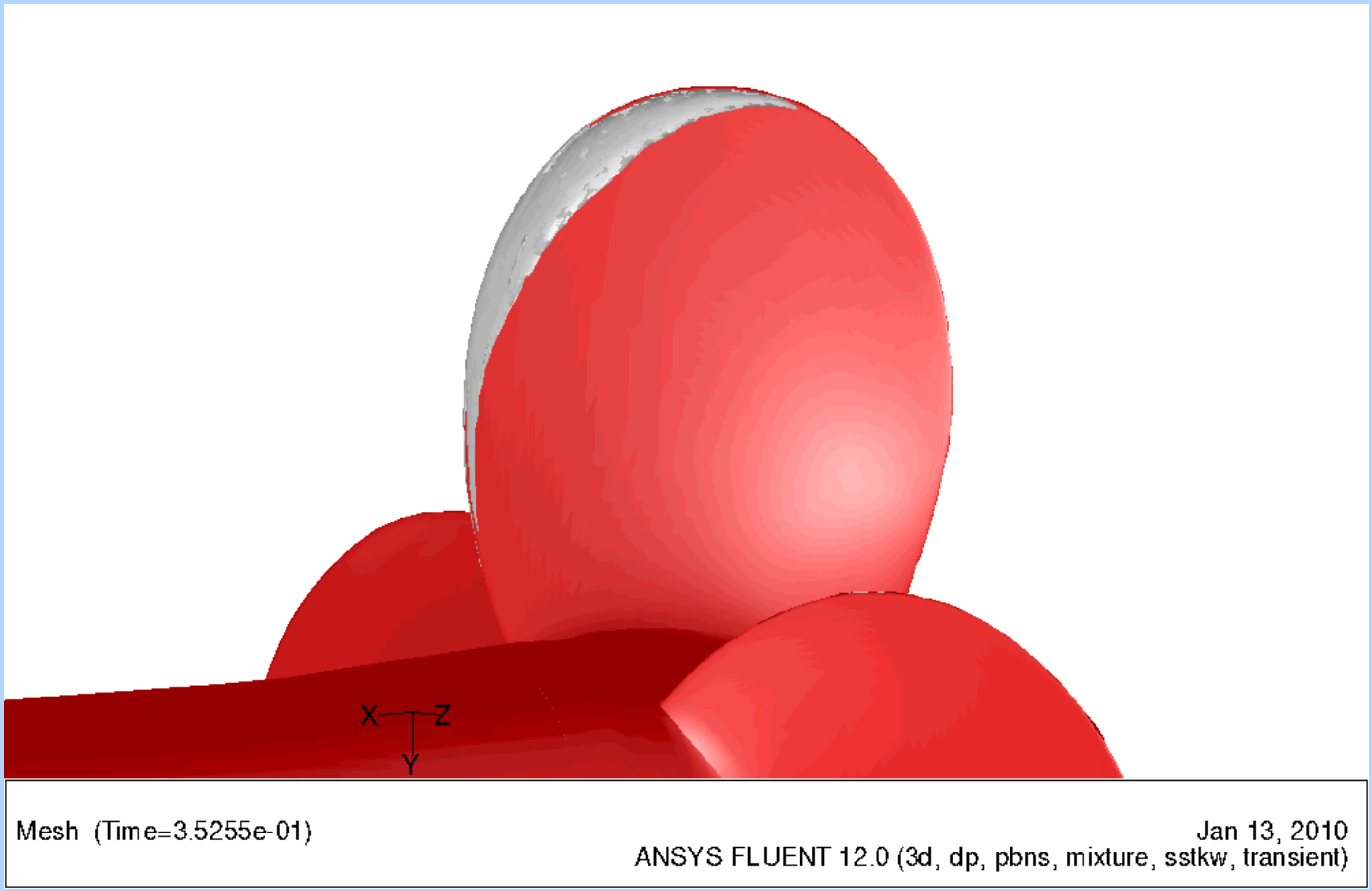
Current status / capabilities

1. Cavitation inception
2. Tip vortex cavitation and scaling – *grid resolution critical*
3. Travelling bubble cavitation
4. Stable sheet cavitation
5. Unsteady sheet cavitation – *cavity extent, re-entrant jets, break-off, shedding frequency*
6. Performance breakdown – *underpredicted* C_L & C_D at low σ
7. Pressure fluctuations – *underpredicted amplitude*
8. Collapse induced shock waves/pressure waves, pressure pulses and noise – *still a challenge*
9. Erosion – *Qualitative level for judgement of erosion risk. Quantitative method not established yet*

Example 1 – Delft twisted-11 hydrofoil (flow from top to bottom)

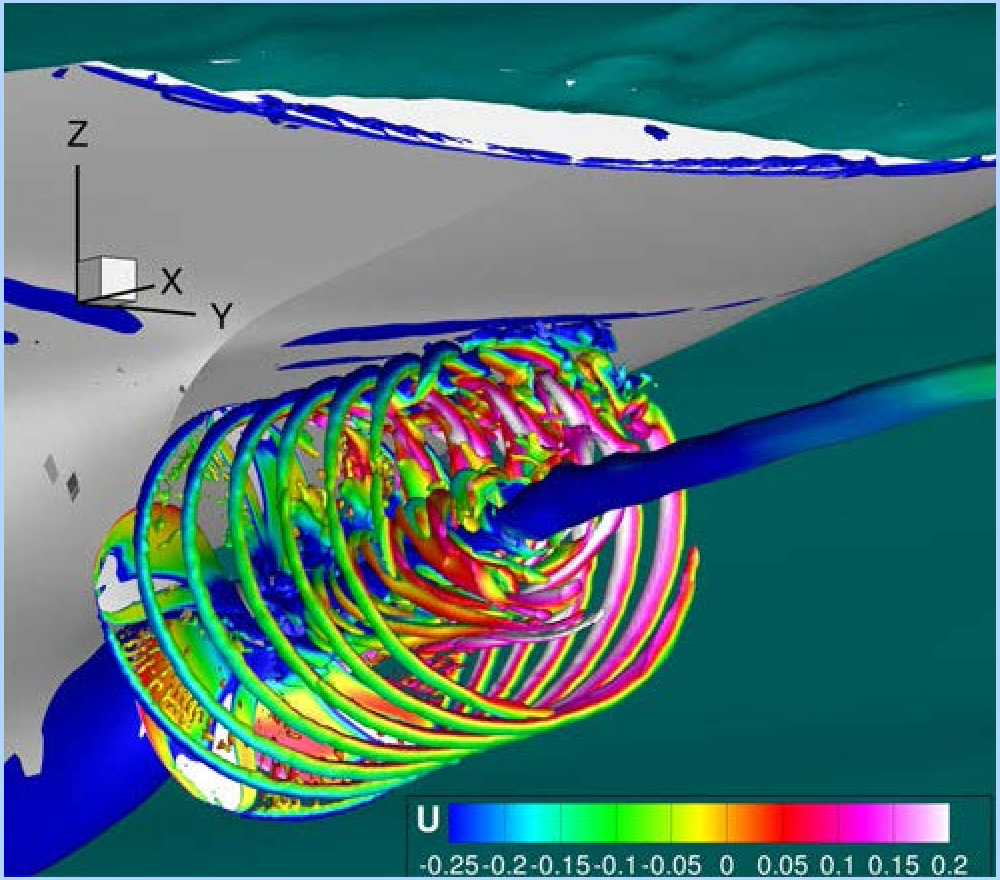


Example 2 – INSEAN E779A in an inhomogeneous wake



PROPULSION

- Propulsion computations are based on the double-model or free surface resistance computations.
- Both fully discretized (both hull and propeller) and body force approach are widely used.
- Most of the computations are done in model scale and standard skin friction corrections are used.
- CFD methods are close to be a every day tool for propulsion performance estimation.



Carrica et at 2010; Isosurfaces of axial velocity for KCS under self-propulsion

G2010 CFD WORKSHOP

- **Test case to compute self propulsion for KSC hull a) fixed model at ship point b) free model at model point**
- **17 different computations**
- **Given skin friction coefficient or given *n* to determine the self propulsion point forces were alternatives. => when the former approach was used the errors in K_T ,K_Q and n values were smaller**
- **Half used actual propeller computation and half different kind of body force method => clearly smaller scatter in K_Q values when actual propeller was used**

Case no.	Classification	Group	Prop. model	E%D			
				KT	KQ	n	RT(SP)-T
2.3a	Given n, actual propeller	CSSRC	A	0,06	-1,39	-	-8,03
		MARIC	A	4,12	-2,88	-	-4,12
		SNUTT	A	-1,94	-7,99	-	-3,43
		SSRC(1)	A	3,35	-0,35	-	-8,89
		TUHH-FDS&ANSYS	A	6,47	-0,42	-	-14,38
	Given SFC, actual propeller	CTO	A	11,65	1,77	-3,16	-
		IIHR	A	0,65	-2,81	-1,27	-
		SSRC(2)	A	-1,59	-3,82	-2,11	-
	Given SFC, modeled propeller	IIHR/SJTU	BP	2,4	1	0,7	-
		MARIN	Body f.	-4,7	-7,3	2,6	-
		MOERI	BS	1,76	0,66	-1,11	-
		NMRI	BX	-6,53	-16,32	5,68	-
		South/QinetiQ	BP	-18,92	-17,99	1,49	-
		SSPA	BL	-5,34	-6,26	2,34	-
2.3b	Given SFC, act. or mod. propeller	IIHR	A	6,7	5,1	-2	-
		MOERI	BS	12,13	12,55	-3,87	-
		SSPA	BL	-0,18	2,45	4,98	-

Summary of self-propulsion computations of KCS hull (Larsson et al 2010)

Maneuvering

Approach 1: Use CFD to compute derivatives for system-based methods

-Static "manoeuvres"

- Turning circles, pure drift, turn+drift
- Toxopeus (2006), Queutey and Vissoneau (2007), Bhushan *et al*. (2009), many others.

-Dynamic manoeuvres (PMM)

- Pure sway, pure yaw, sway or yaw+drift, constrained or free to pitch, heave and/or roll
- Broglia *et al*. (2006), Cura-Hochbaum (2006), Sakamoto *et al*. (2008), others.

Approach 2: Direct CFD simulation of free model manoeuvres

-Zig-zag

-Turning circle

-Spiral test

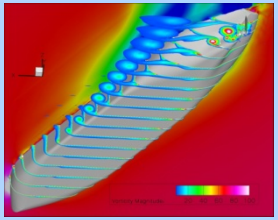
-Constant RPM or constant torque

-Requirements

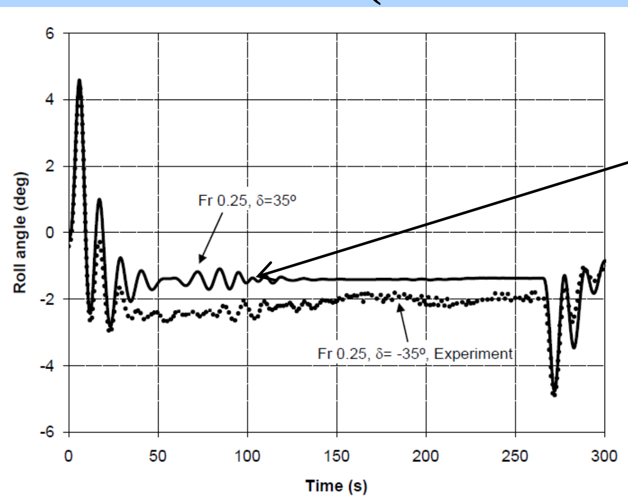
- Propulsion
- Moving rudders
- Controllers

- Xing-Kaeding and Jensen (2006), Muscari *et al*. (2008), Carrica *et al*. (2008a, 2008b)

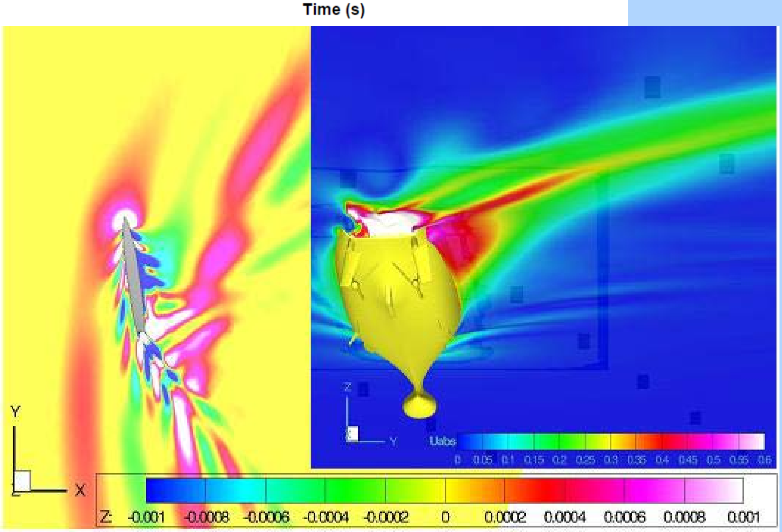
KVLCC1 in 30 degree rudder turning circle



Marin 7967 in turning maneuver hit by its own Kelvin wake (Carrica et al. 2008)



Wake reaches the ship



Conclusions

-Though it requires more resources and advanced code capabilities, CFD use for manoeuvring is becoming more commonplace, though validation of results and procedures has been more limited.

-CFD is adequate to obtain derivatives for system-based manoeuvring calculations

-Significant head has been made towards full CFD computations of maneuvers, though still very expensive

-Simulations with resolved propeller appear feasible and fairly accurate

Seakeeping

Capability requirements

- -Waves: regular, irregular
- -Motions
- -Wet deck
- -Slamming
- -Self-propulsion
- -Controllers

Towing tank simulations: Head or following waves, regular or irregular (long crested)

Wave basin simulations: regular or irregular waves, captive or free model

Seaway simulations: irregular, short-crested waves (Bretschneider, JONSWAP, Pierson-Moskowitz, etc.)

Pitch and heave: 2DoF, ship or carriage system, single wavelength, Fourier spectrum or Focused wave.

-First computations by Sato et al. (1999).

-10 computations presented in G2010 for pitch and heave of KCS and KVLCC

- Motions within 10% of data, added resistance is more challenging

Pitch, heave and surge: 3DoF, imposed force, RPS or speed.

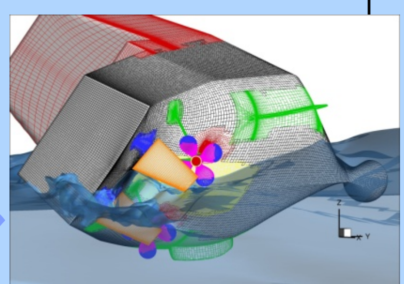
-G2010 attracted two submissions: El-Moctar et al. using Comet and Sadat-Hosseini et al. using CFDShip-Iowa

Free model: 6DoF, propulsion, controllers (autopilot), waves, wind

-Problems of interest: Dynamic stability, controllability, seakeeping

-The boundary with maneuverability becomes blurred

-Broaching:
P controller
PI v-gain controller



Conclusions

- More codes able to compute seakeeping problems
- Pitch and heave responses currently reasonably predicted with CFD (within 10% of EFD)
- For more complex problems there is limited experience and data, but the capability exists:
  • Pitch-heave-surge
  • Seakeeping in oblique waves (free or captive model)
  • Stability in waves, controllability, FSI
  • Ship-ship interaction in waves (free or captive model)

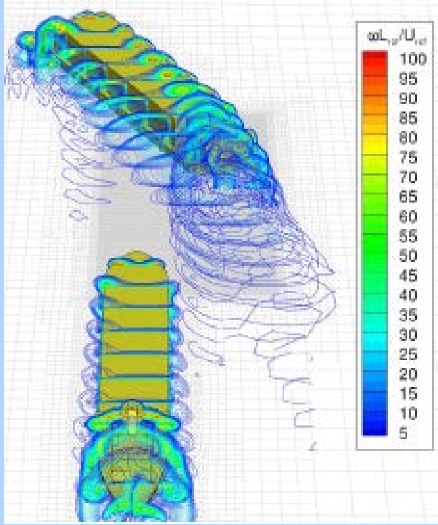
Ocean Engineering

Practical applications

1. Coupled wind-wave and wind Loads Simulation

-- a new area in which CFD has made significant progresses recently --

- ➢New method to capture the non-linear processes in **realistic ocean wave simulation** with the turbulent wind motions. (Shen et al. (2008))
- ➢ CFD Simulation for estimating **wind loads, wing wakes and shielding** effects. (Koop et al. (2010))



Wake (Non-dim. vorticity) of FPSO (Koop et. At 2010)

2. Wave/Fluid-structure interactions, including viscous effects

*-- a challenging problem in non-linear /breaking waves, **numerical techniques** are developed –*

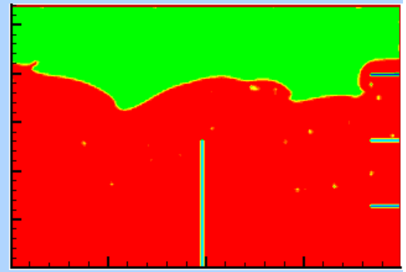
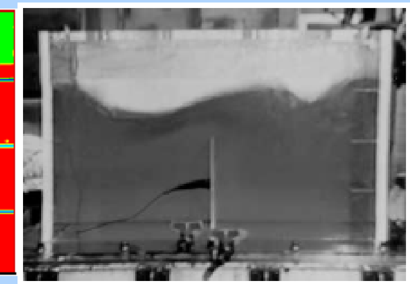
- ➢SWENSE (**Spectral Wave Explicit Navier-Stokes Equations**) approach (Monroy et al. (2009))
- ➢ Coupled Eulerian scheme with two Lagrangian particles (**SPH and free surface particle on Eulerian grids**) (Baso et al. (2010))

Ocean Engineering

3. Violent flows, slamming, sloshing, green water on deck, impact

-- *CFD is clearly a powerful tool for simulating violent flows, but* ***should be robust enough*** *for engng. prediction* --

- 3D CIP (constrained Interpolation Profile) for **water entry problems.** (Yang et al. (2010))
- RCIP scheme for predicting the **violent sloshing** (Hu et al. (2010))
- The modified VOF and Young's VOF for **sloshing** problem. (Wemmenhove et al. (2009) and Liu et al. (2010))

Experimental and calculated free surface profile due to sloshing (Liu et al. 2010, Kim et al. 2001)

Numerical methods and schemes

1. Hybrid methods for potential/viscous flow coupling

- Efficient VOF based RANS method (Woeckner et al. 2010): implicitly forced viscous RANS complying with a prescribed solution towards the far-field boundaries for **problem of motion in waves**.

Ocean Engineering

- Finite difference method (FDM) and smoothed particle hydrodynamics (SPH) (Kim Y (2007)) for **coupling problem** of sloshing and ship motions.

2. Verification, validation and uncertainty analysis

-- *Still few methods and experiments are applicable for actual engineering use* --

- **Sloshel Project** (Brosset et al. (2009), Maguire et al. (2009)) aiming to reproduce at full-scale wave impact condition due to sloshing.
- Method of **the Manufactured Solution** - a procedure for CFD code verification (Eca et al. (2010)).
- **ASME's procedure** for UA (Roache (2009), ASME Committee (2010)) : to estimate the modeling error including numerical, experimental and parameter uncertainties.

Conclusions

(1) The focuses of CFD application in OE are placed on problems of non-linearity, viscosity and FSI.

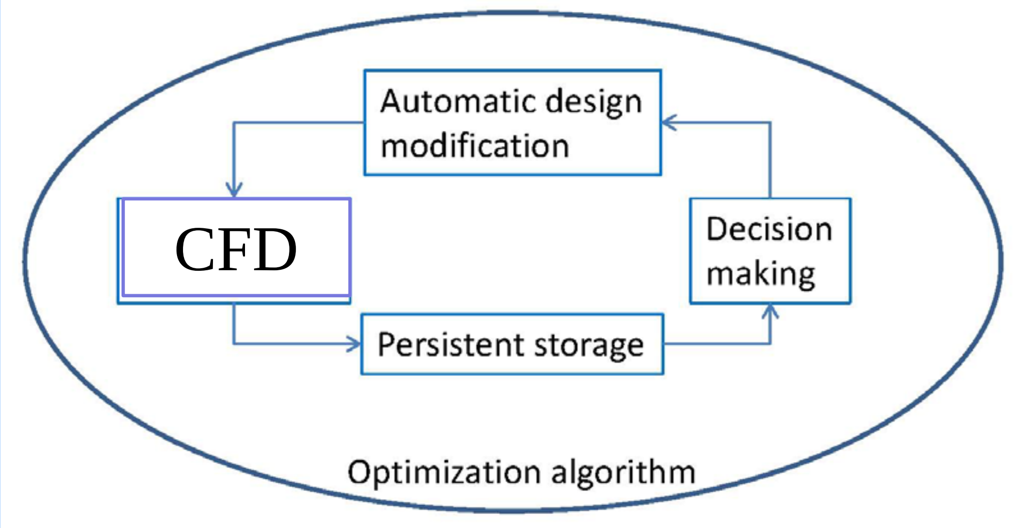
(2) Numerical methods and schemes which are the matter of concerning, had been significantly developed.

(3) Validation is an on-going activity that intends to estimate the modeling error. Benchmark and experimental data including full-scale are needed.

Simulation Based Design

SBD: general framework to integrate **Simulation**, **Optimization** and **Design**

- Simulation (CFD) tools evaluate **design performances,** feeding an **optimization algorithm** capable of finding the minimum of some user-defined objective functions
- **Geometry-modeling** method provides the link between the design variables and the deformation of the body
- **Persistent storage** accumulate trial solutions
- **Decision making** is necessary for multi-objective problems



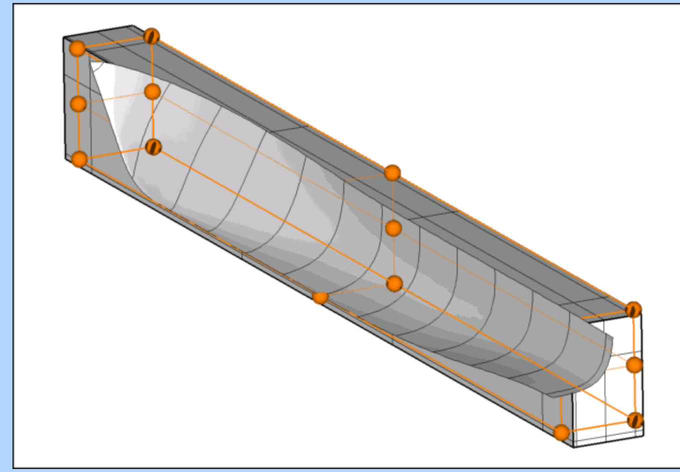
SBD is computationally expensive and needs accurate CFD solvers.

SBD allows for:

- Multiobjective Design
- Robust Design

Simulation Based Design

- Constrained, Continuous Optimization:
  - Gradient-based (local) vs. Derivative-free (global)
  - Single and multi-objective problems
- Automatic mesh and geometry deformation →



- Variable Fidelity methods
- Multidisciplinary Design Optimization
- Uncertainty Quantification and Robust Design Optimization

Consider an objective function *f (d,u)*, where
- ***d*** represents the **design variables** (**controlled by the designer**)
- ***u*** represents the **uncertainty** (**not controlled by the designer**),
- ***f(d; u)*** quantifies the **design performance's loss** **when condition *u* occurs** (with probability *p(u)*)

Replace some of the objective functions *f* with a more complex function *φ(d)*

$$\min_{d \in D} \varphi(d), \quad \varphi(d) = \int_U f(d;u)\,p(u)\,du,$$

Practical Guidelines for Ship CFD Applications

Goal : Separate analyses of the same problem, using the same model physics, should produce consistent results.

Aim: to encourage a common best practice.

Inevitably, the Guidelines cannot cover every aspect of CFD in detail →

Practical Guidelines for Ship CFD Applications

They are intended to offer roughly some of the most important general rules of advice that cover (hopefully!) most of the problems likely to be encountered.

As such, they constitute essential information for the novice user and might provide a basis for quality (and safety) management which rely on CFD.

We hope that they can also provide useful advice for the more experienced user

Practical Guidelines for Ship CFD Applications

The guidelines are written assuming the use of **surface capturing** methods, the method found in most commercial and academic CFD packages.

It also assumes that the solver is **grid-based**, as opposed to mesh-free methods.

We divide the CFD process into **pre-processing, computation, and post-processing** steps.

- ❑ **Pre-processing:** definition of the problem, grid generation and input setup
- ❑ **Computation**: preparing the computer to run the problem, and running.
- ❑ **Post-processing:** provide useful numbers and plots.

THE END