



Current status in CFD Resistance & Propulsion

- Application of CFD in the maritime and offshore industry
- Progress in Viscous Flow Calculation Methods
- Trends: from G2K to CFDWT'05
- Analysis and design



0. Validation of prediction techniques

This is not a typo

Need and importance of establishing credibility of CFD simulations and codes through verification and validation (V&V)

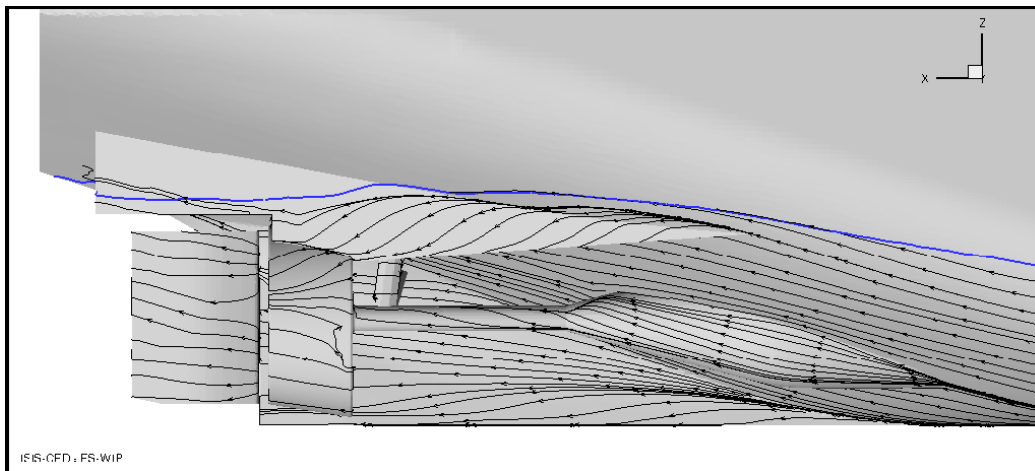
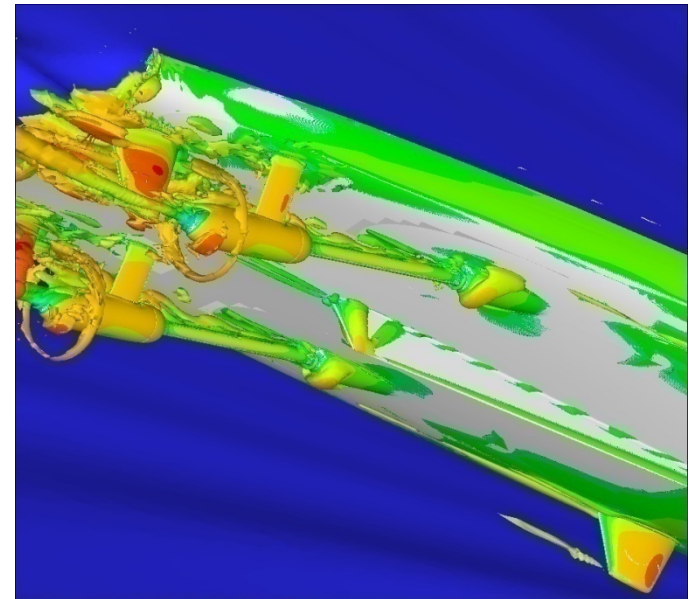
Resistance Committee report reviews recent activities in the field of Verification and Validation (V&V) considered to be of significance for the members of ITTC

Application of CFD in the maritime and offshore industry

- Inviscid methods still heavily used
 - Free- surface Panel Methods (linear – non linear)
- RANS model scale calculations
 - Large amount of hull forms
 - Increasingly sophisticated with actual geometry: appendages, bilge keels, shafts, struts, propulsors
- RANS full-scale calculations
 - Wall function w, w/o roughness
 - Becoming nearly as routine for realistic configurations as model scale predictions
 - Limited experimental data for comparison
- Sinkage and trim capability increased

RANS Practical Applications

Miller et al. (2006) Athena model scale prediction



Visonneau et al. (2006) Limiting wall streamlines of propelled hopper-dredger at full scale

Trends: from G2K to CFDWT'05

CFD WORKSHOP TOKYO 2005

Test Case 1.1



KRISO Container Ship

- Towing condition in still water
- Fixed(even keel)
- Without rudder

Froude number (F_n)	0.26
Reynolds number (R_n)	1.4×10^7



Test case #1.1 (11 participants) – **Resistance Coefficient**

Coefficient of variation V for the generic force coefficients $C_{(\cdot)}$: $V = (\sigma / C_{(\cdot)}) \cdot 100$
 being σ the standard deviation

	C_T	C_P	C_F
<i>Exp.</i>	3.56	---	2.832*
Mean	3.600	0.744	2.856
Std. Dev.	0.1501	0.0858	0.1895
V	4.17 %	11.53 %	6.64 %

*ITTC 57

G2K

V for C_T and C_F was found to be about (5%-8%).

Larger values were been obtained for C_P (20%).

CFDWT-05

V is decreased for all force coefficients

C_P still double C_T and C_F

C_P is particularly grid-dependent

Averaged simulation numerical uncertainty U_{SN} is about 2.1% (at G2K was 3.2%)

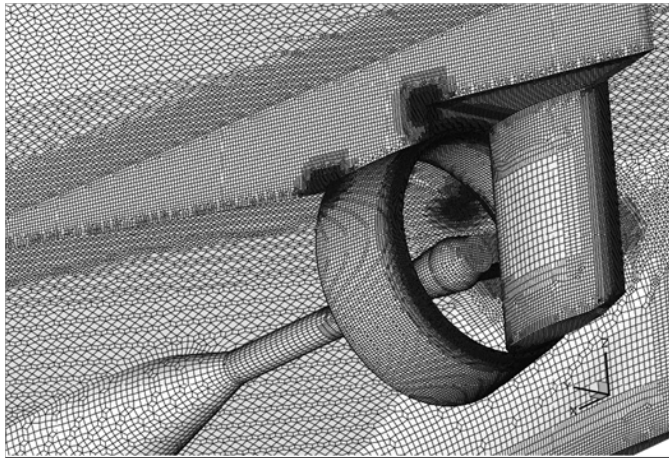
Averaged comparison error E (i.e. the difference between the experimental data and the value from the simulation) for C_T is 4.7% (at G2K was 4.8%)

Progress in Viscous Methods

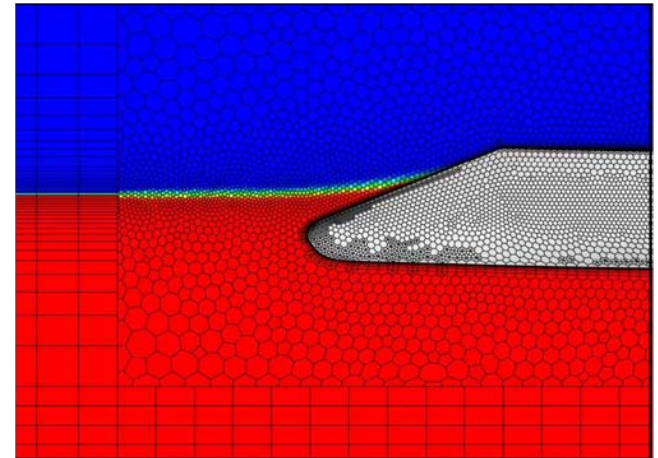
- Variety of grids and gridding techniques
 - Structured grids most heavily used
 - Good for bare hulls and some complicated geometries
 - Oversets being used more often for complicated geometries
 - Unstructured grids
 - Hexahedral, tetrahedral, and polyhedral
 - Tetrahedral and polyhedral need prism layers for boundary layer accuracy
 - **Cartesian with immersed boundary methods**
 - Gridding is trivial [$O(\text{Panel codes})$]
 - Boundary layer prediction still problematic

Gridding

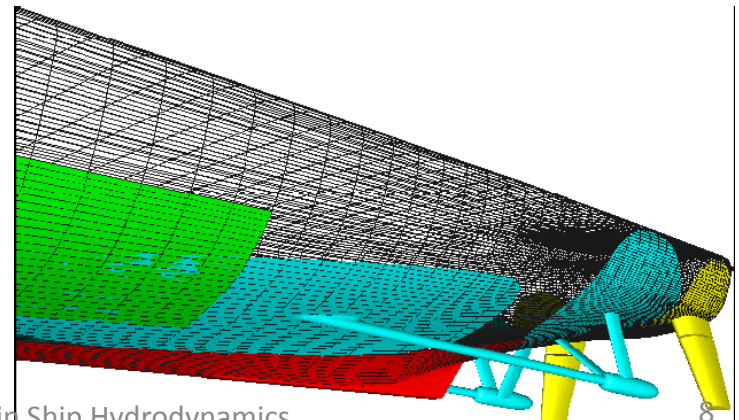
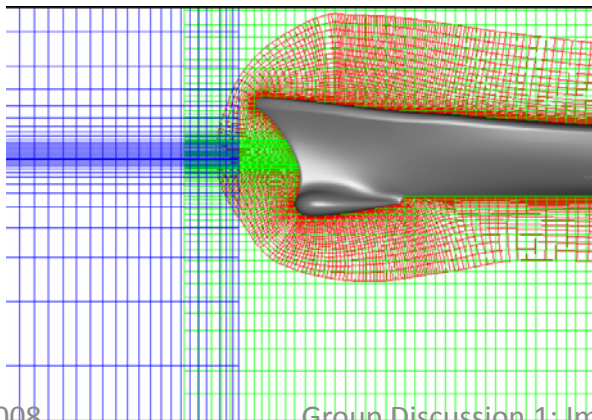
Visonneau et al. (2006) Stern region of hopper-dredger



Maki et al. (2007) Trimaran polyhedral grid



Noack (2007) Overset grids for combatant



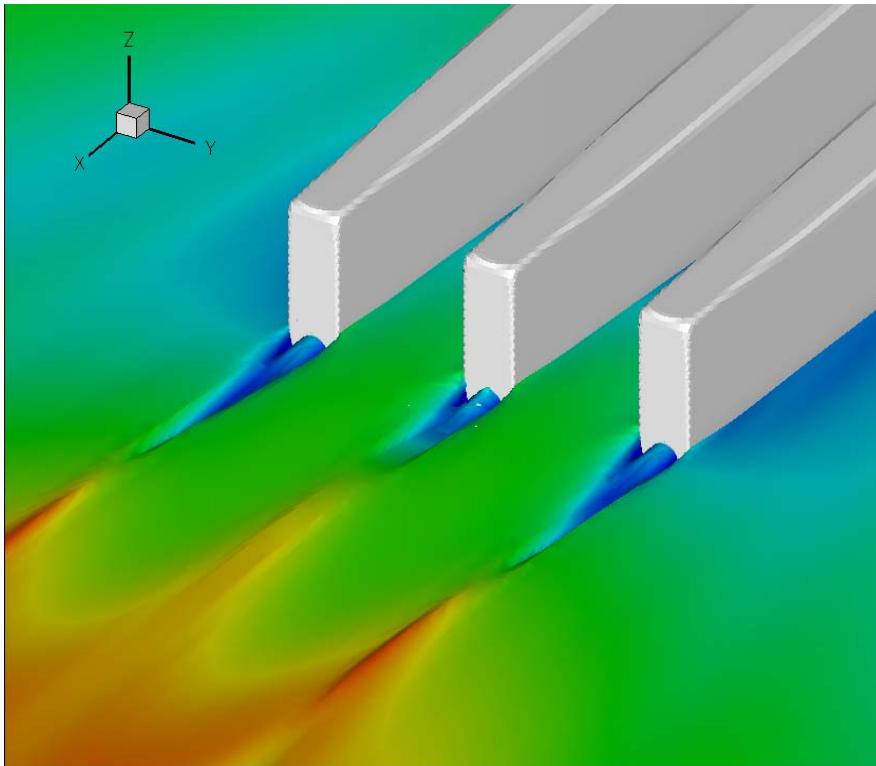
Progress in Viscous Methods

- Free surface treatment
 - Capturing methods have become routine (Volume of Fluid and Level Set) and used by the majority of groups
 - Can numerically handle very complex free surface
- Turbulence modeling
 - Largely one- and two-equations models in practice
 - Reynolds stress models by some groups for flow details
 - Large Eddy Simulations (LES) and Detached Eddy Simulation (DES) seeing more use, but still limited

New Applications

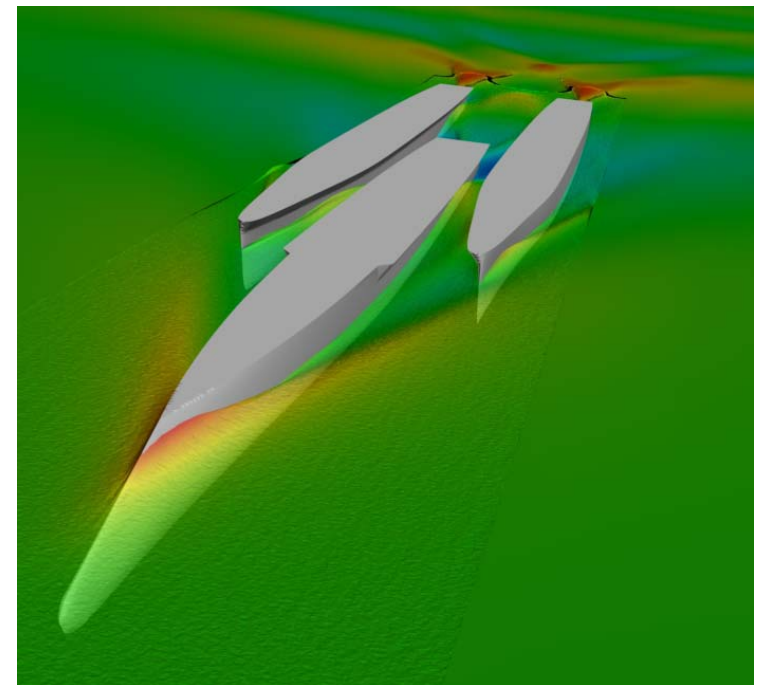
- Propulsor/Hull Interaction
 - Actuator disk models
 - Lifting surface/panel methods
 - Full rotating propeller
- Drag Reduction
 - Microbubble and polymer effects modeled
 - Mostly restricted to simple flows and modeling issues
- High Speed Vessels
 - High Froude number
 - Catamarans, trimarans, slender monohulls

High Speed Vessels

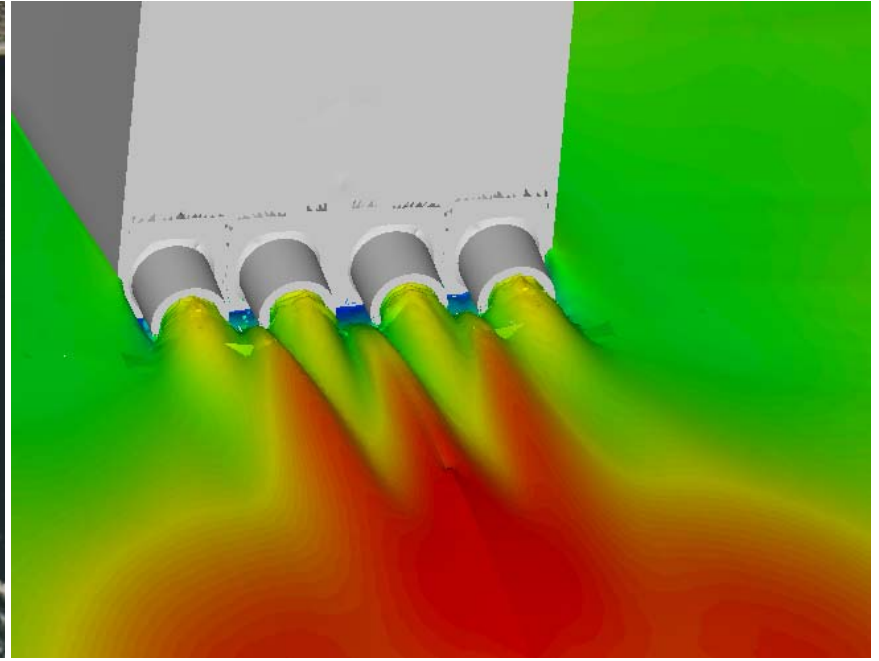


Stern et al. (2006) Trimaran free surface with waterjets

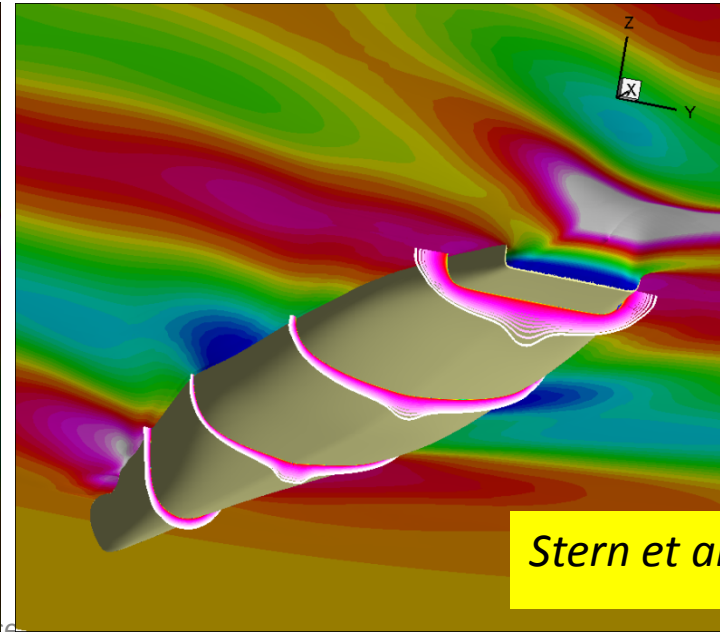
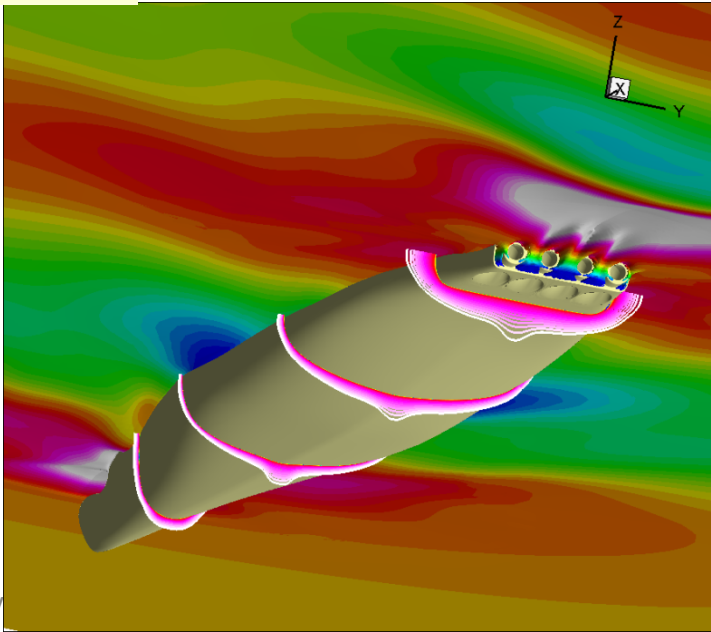
Maki et al. (2007) Trimaran free surface



Propulsion Committee
presentation



Jessup et al., 2008



Stern et al. 2008

2005 ONR Ship Wave Breaking Workshop & Review

Wilson W. et al, 26th SNH, Rome 2006.

Focused effort to assess CFD capability as applied to ship generated waves and wave breaking.

CFD solutions were generated for two full scale speeds (10.5 and 18 kn) and made by four separate groups, utilizing five CFD codes:

Das Boot / NFA / CFDSHIP-IOWA / Comet / Fluent

Physics: Potential flow, NS “no-viscous-flux” solver, RANSE solvers

Free Surface: Interface Tracking, Level Set, VoF

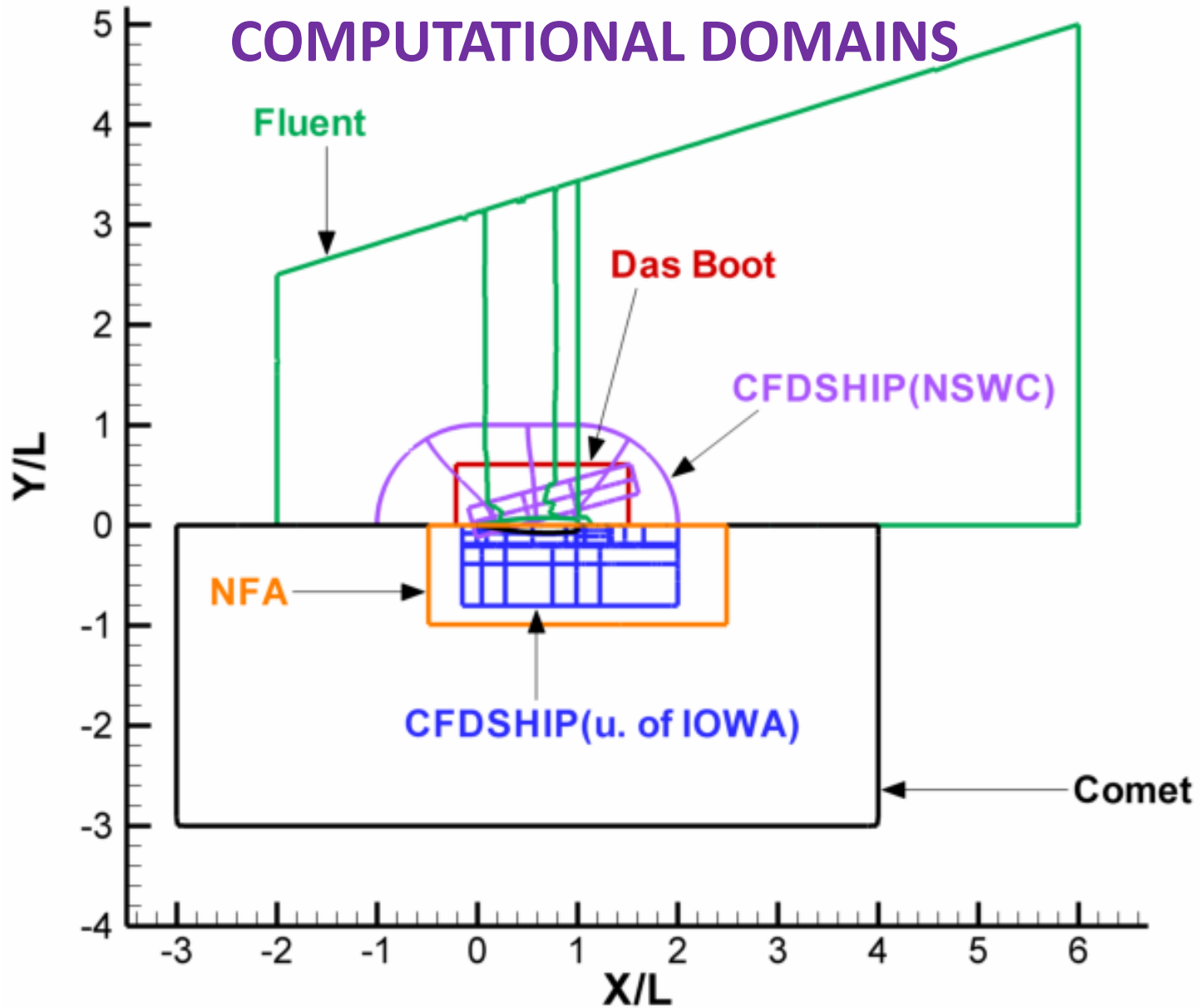
Turbulence closure: Blended $k-\omega$, Blended $k-\epsilon/k-\omega$, Realizable $k-\epsilon$, $k-\omega$ SST

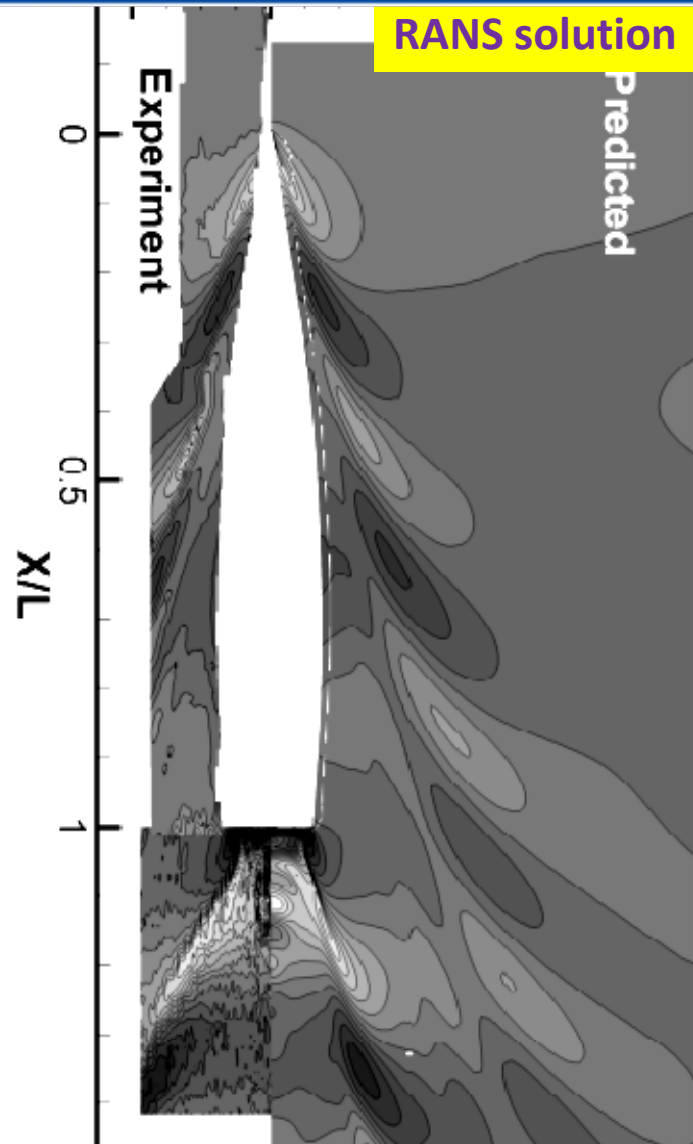
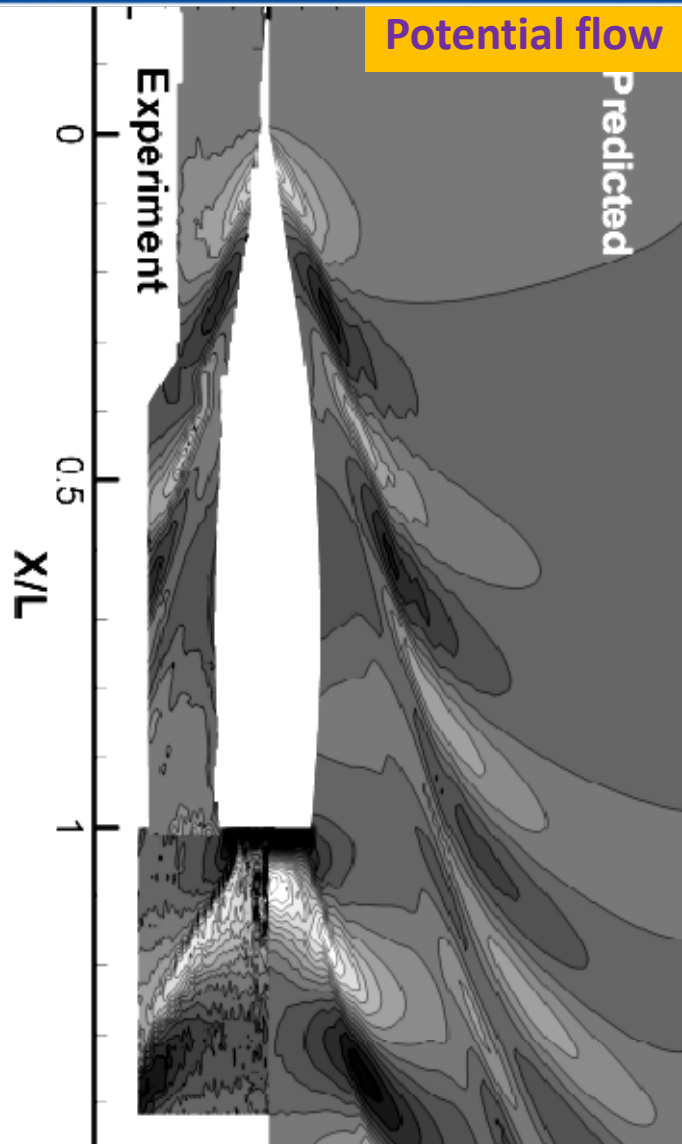
Seven separate solution sets were submitted for each of the test conditions

Although focus was on free surface, total resistance was also predicted by each code for two different ship speeds and compared with model test data.

All of the CFD predictions were performed in a “blind” manner, with the results provided prior to the experimental measurements being released

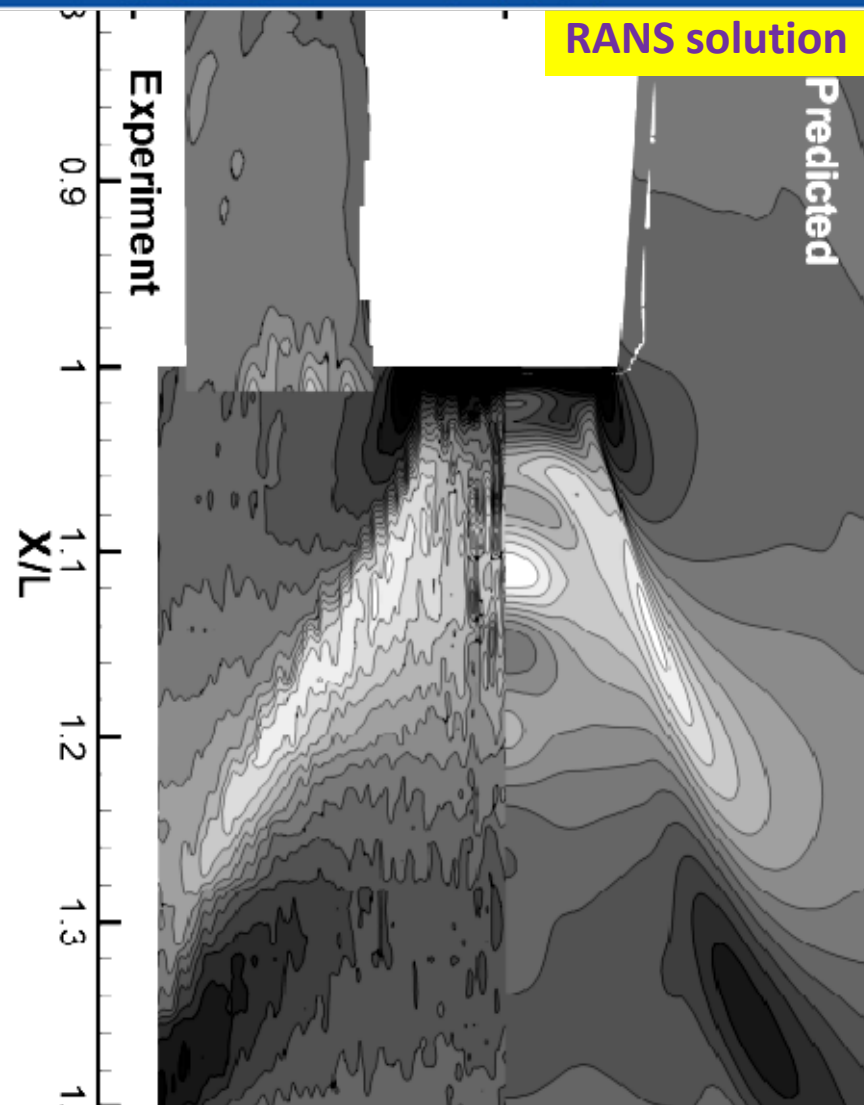
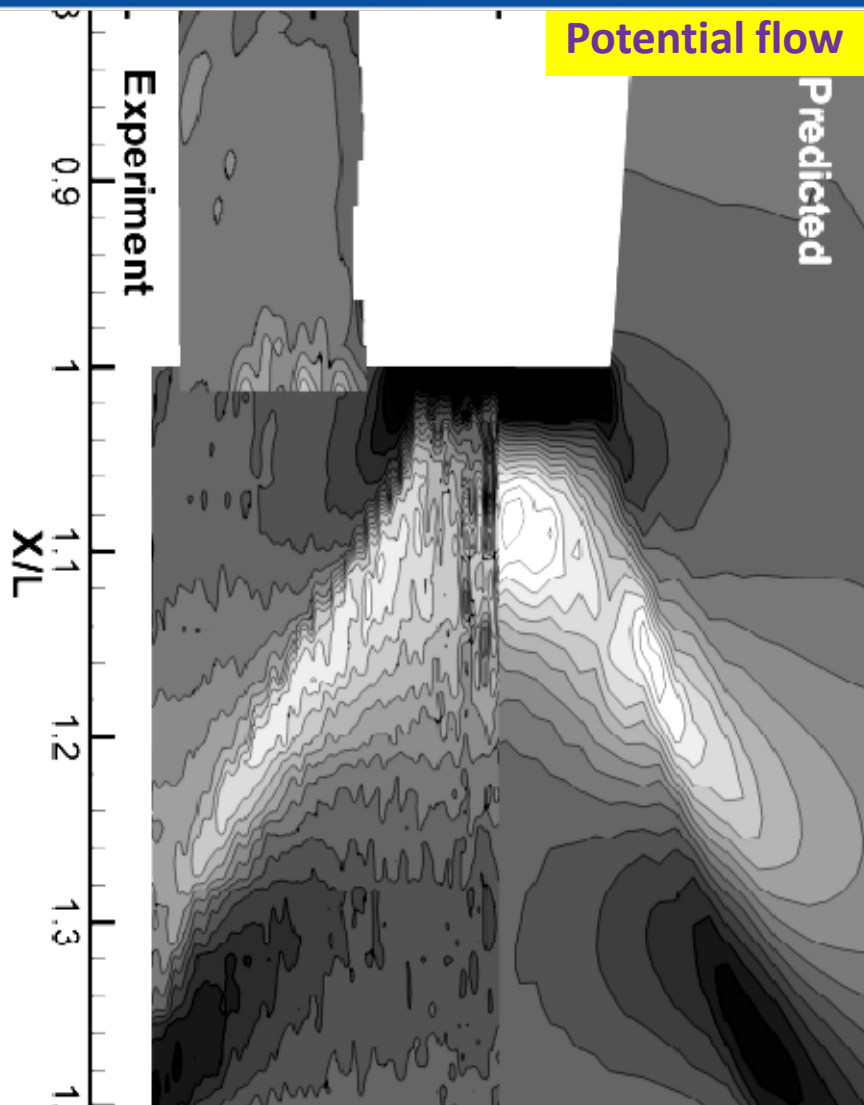
COMPUTATIONAL DOMAINS





Good prediction of the Kelvin wake

Good prediction of the Kelvin wake

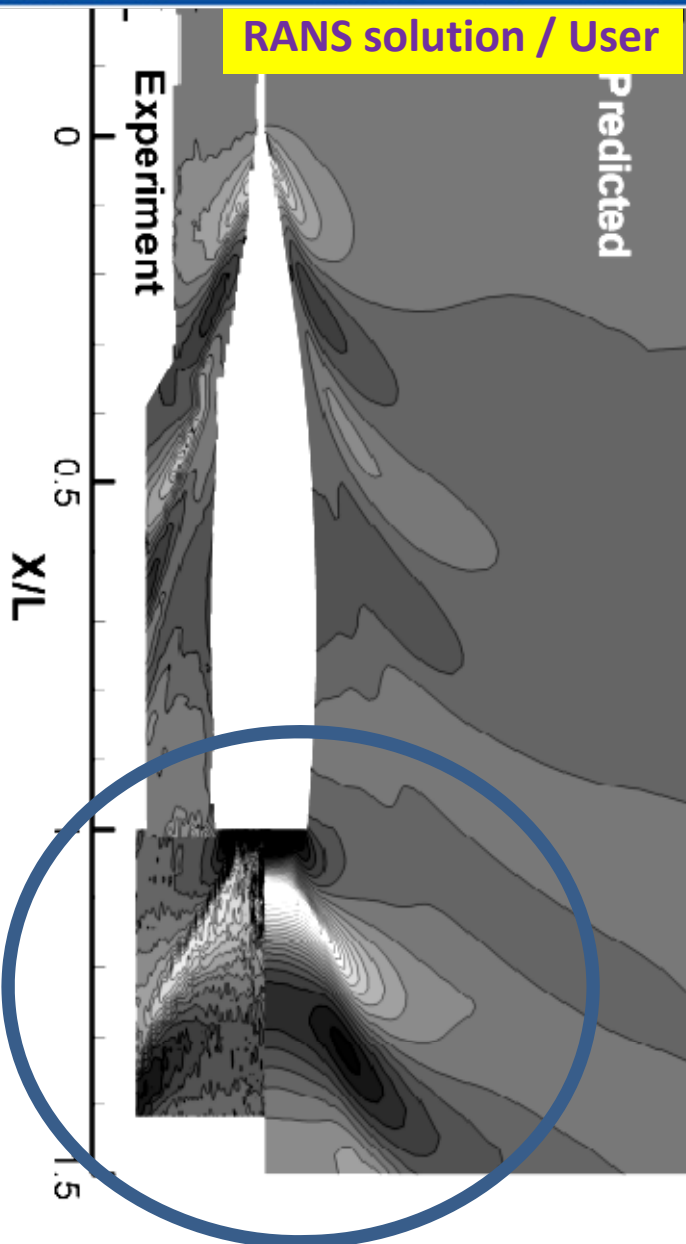


Good prediction of the wave trough aft of the transom. Wave heights aft of the stern slightly over-predicted and broader wave peak.

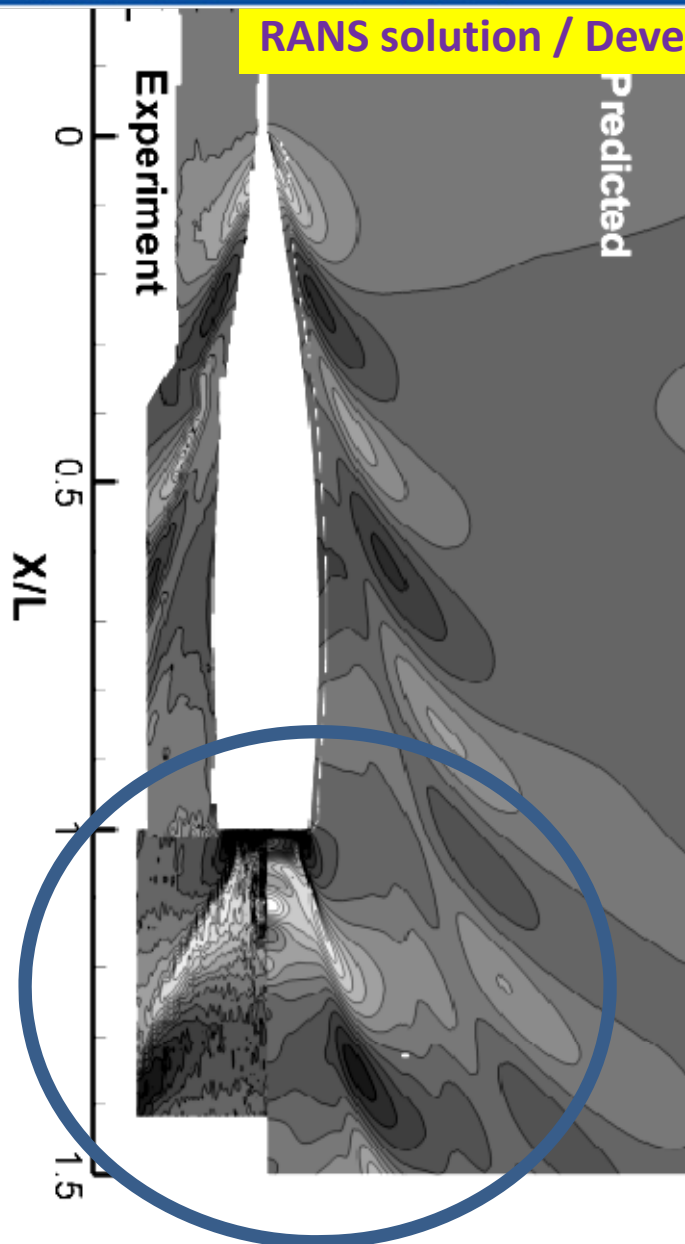
Good prediction of the wave heights and topology in the stern region.



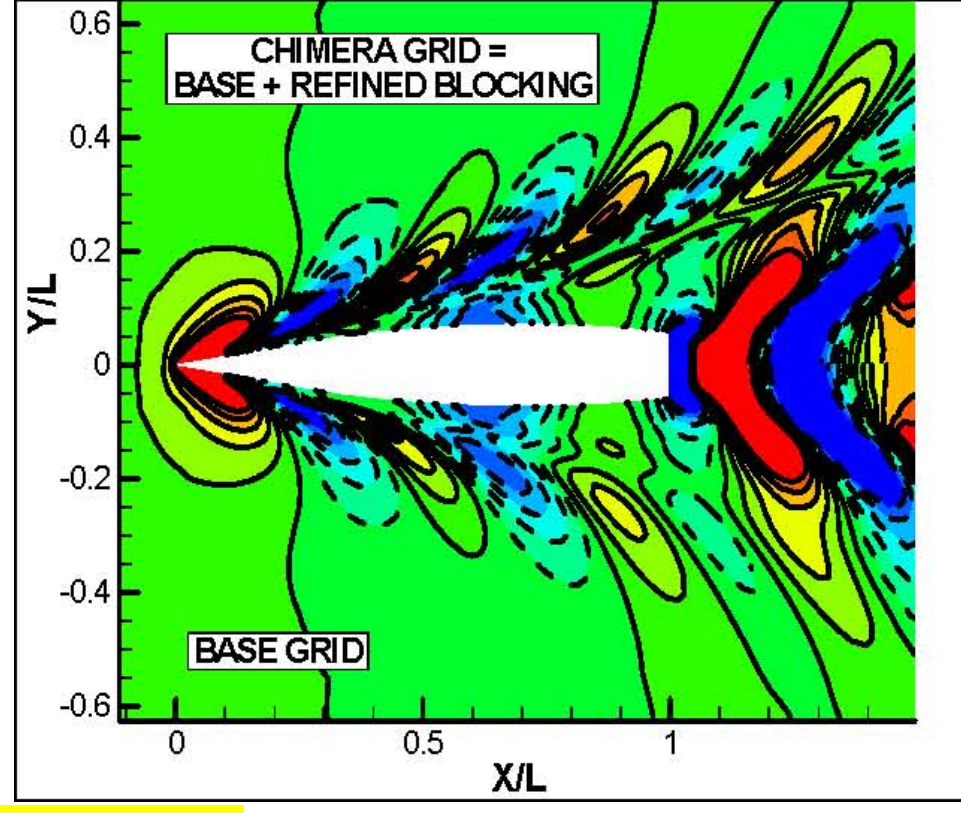
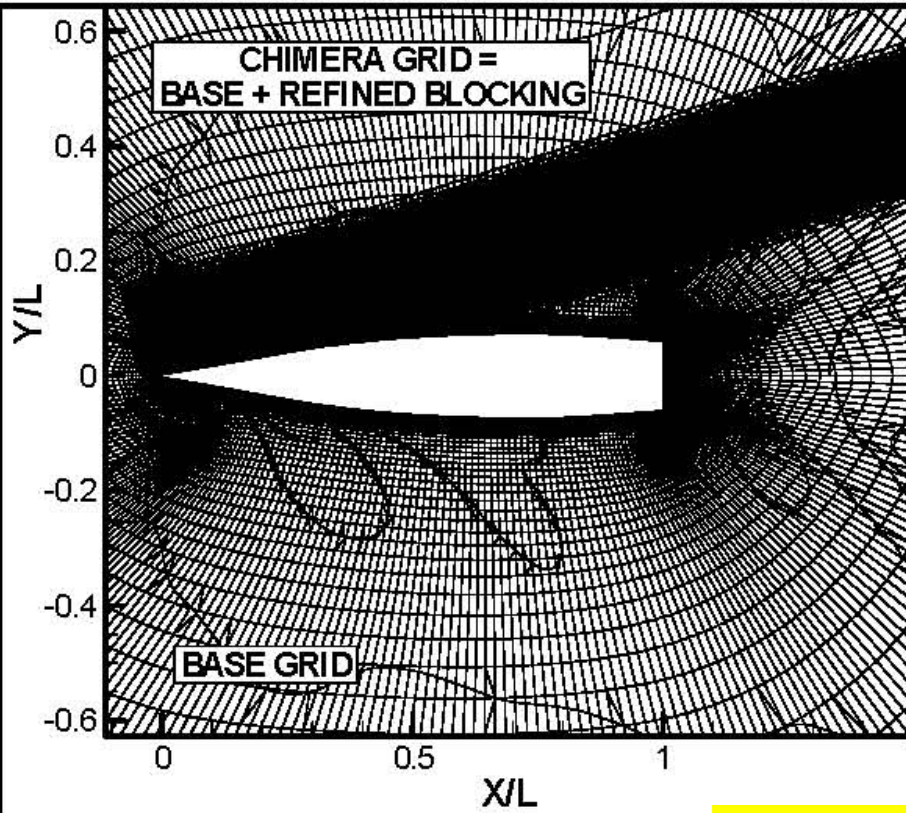
RANS solution / User



RANS solution / Developer



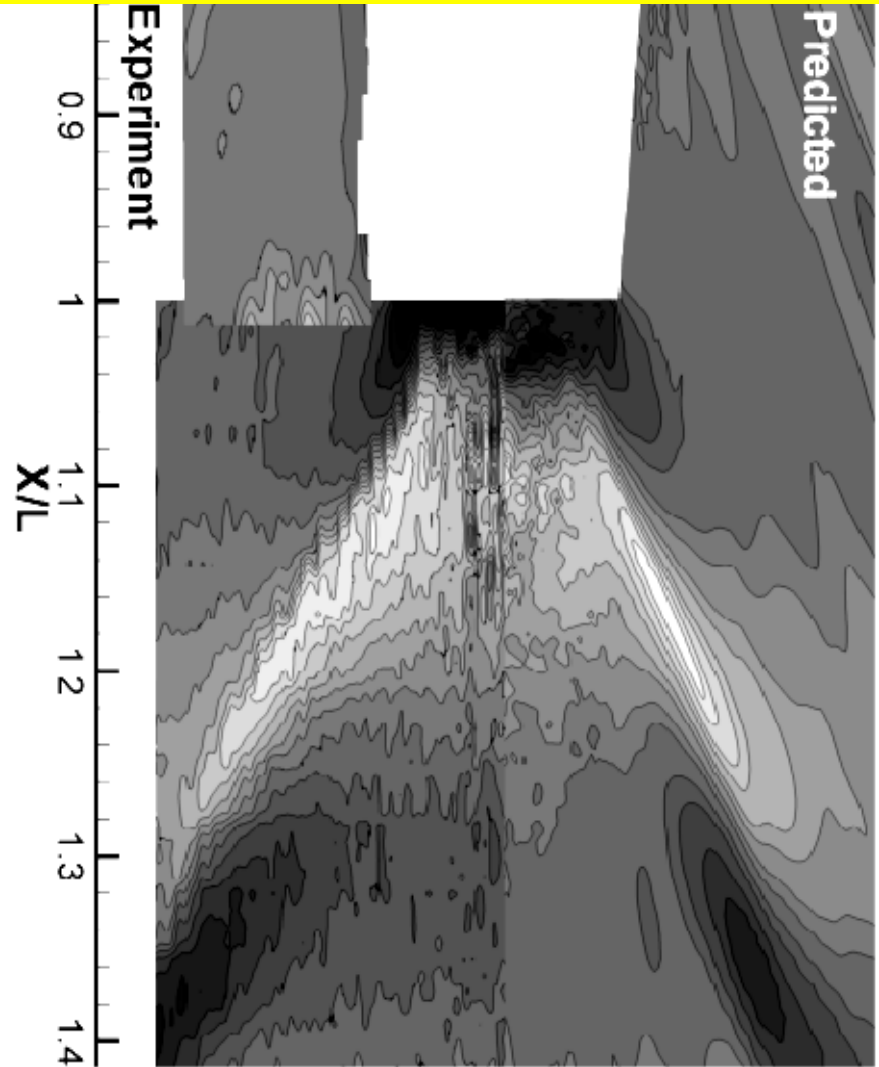
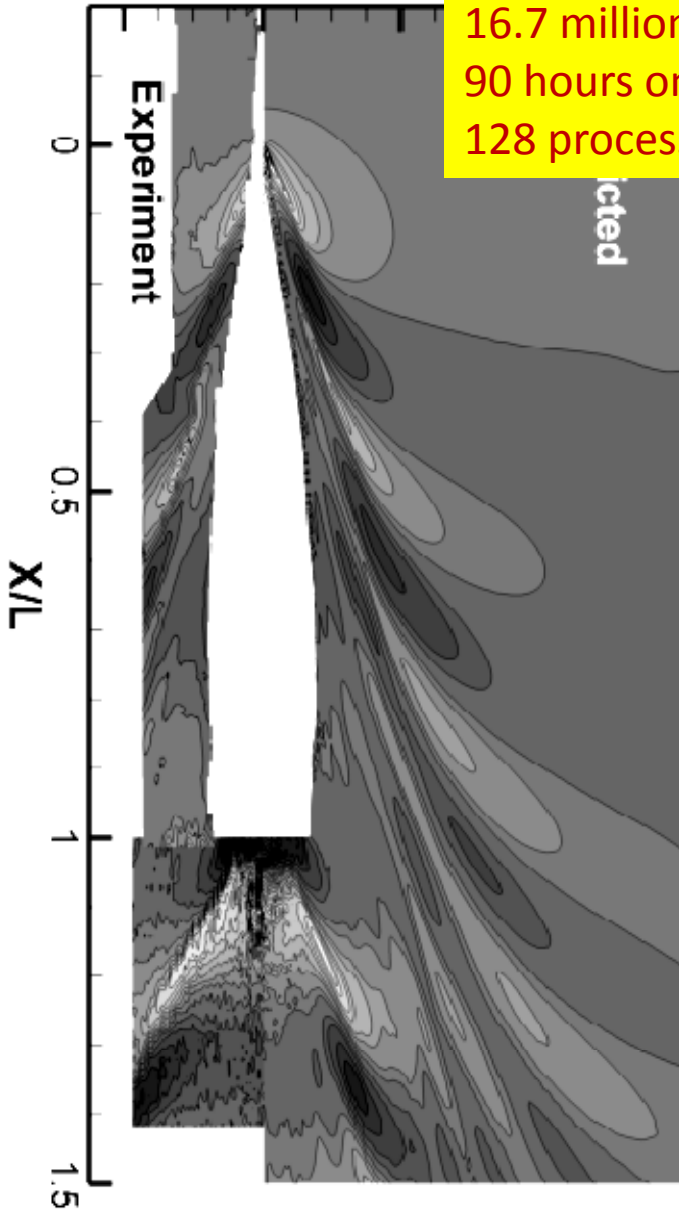
Free Surface grid refinement



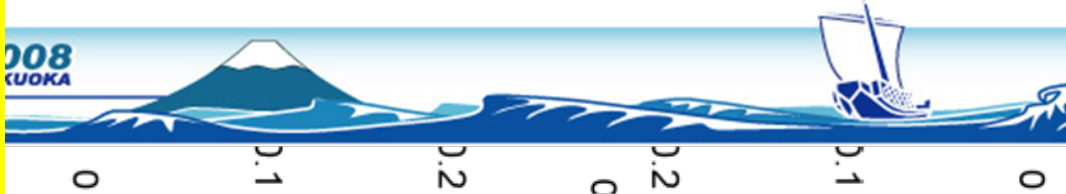
RANS solution / User

16.7 million cells
90 hours on
128 processors T3E

Excellent prediction of the stern region
Small-scale details in the stern wave topology.

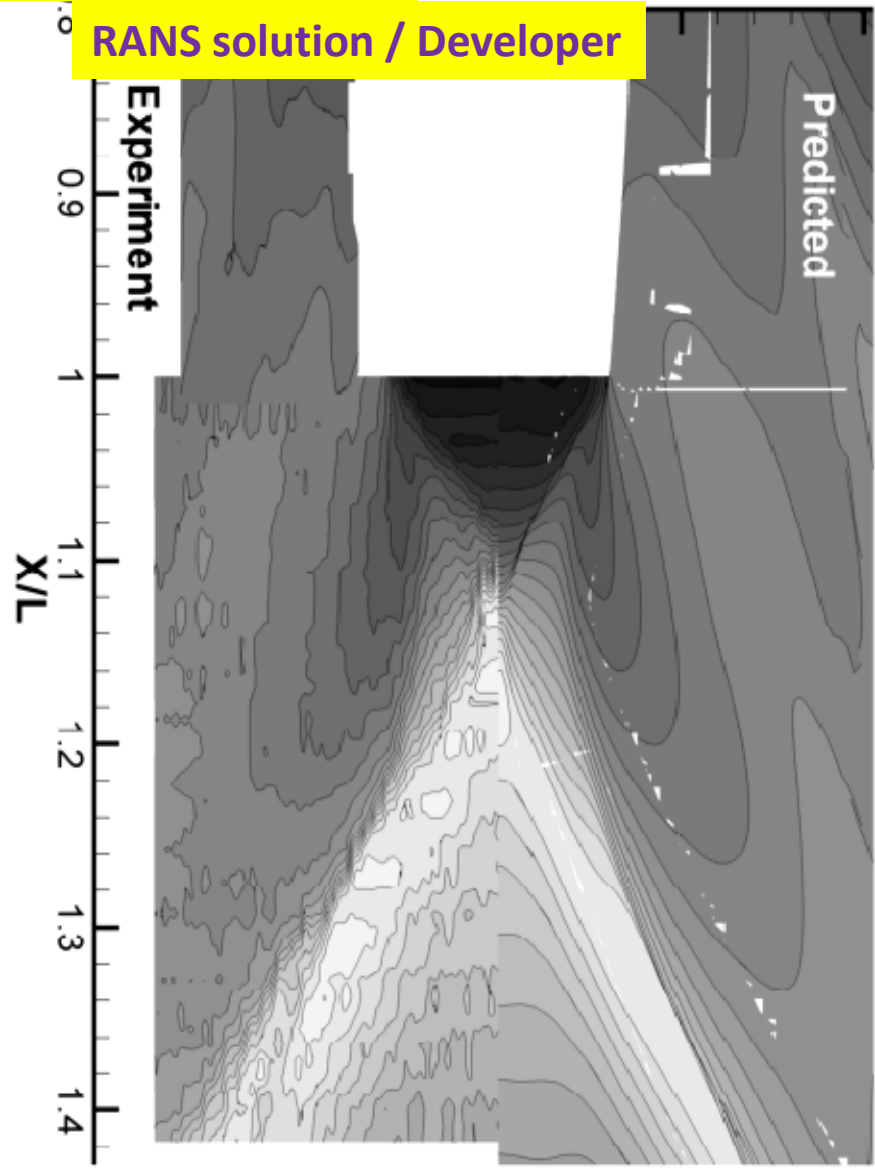


6 million cells
64-88 processors on
SGI Origin 3800

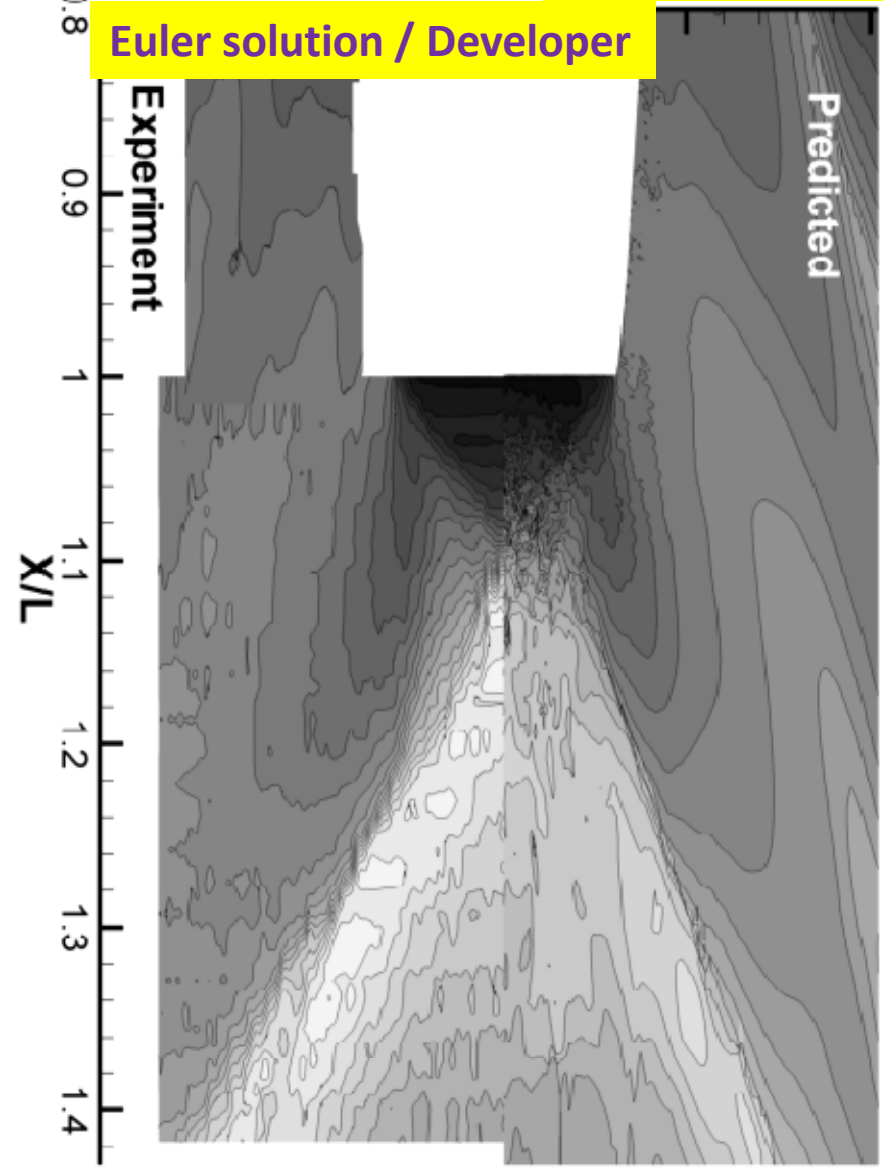


89.1 million cells
55+75 hours on
256 processors T3E

RANS solution / Developer



Euler solution / Developer



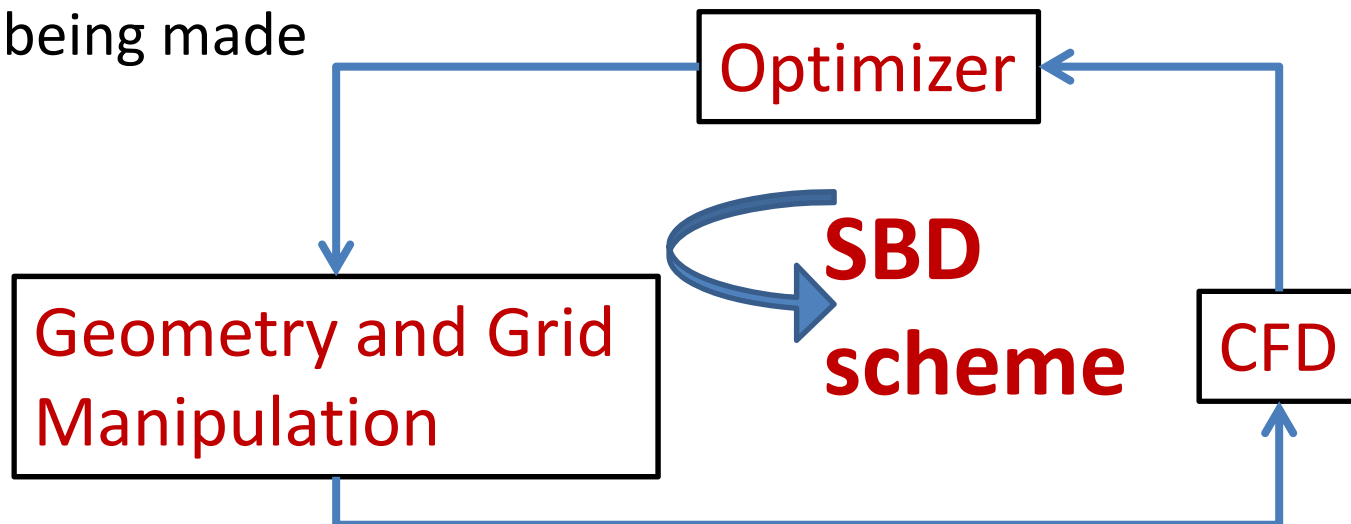


Fx (lbs)	Code	Fx (lbs)	Code
11.43	Das Boot (SAIC)	31.03	Das Boot (SAIC)
10.43	CFDSHIP-IOWA (U. of Iowa)	28.24	CFDSHIP-IOWA (U. of Iowa)
11.38	CFDSHIP-IOWA (NSWCCD)	31.76	CFDSHIP-IOWA (NSWCCD)
11.33	CFDSHIP-IOWA w/Chimera	31.81	CFDSHIP-IOWA w/Chimera
9.56	Fluent (NSWCCD)	28.46	Fluent (NSWCCD)
10.78	Comet (NSWCCD)	33.05	Comet (NSWCCD)
11.00	1979 (Interpolated)	32.30	1979 (Interpolated)

Each of the different solution methods has different advantages and disadvantages.

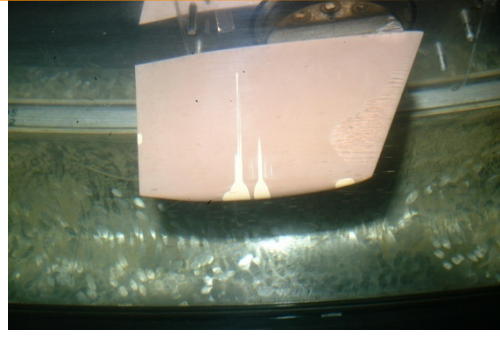
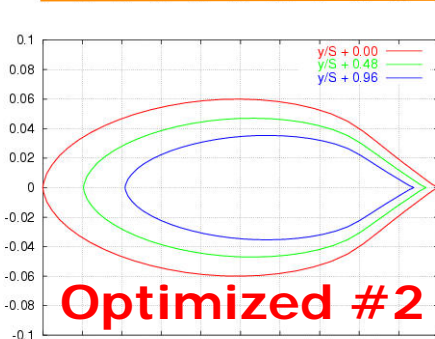
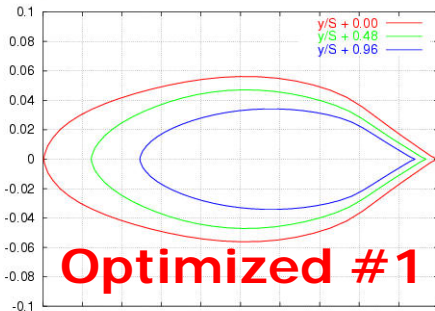
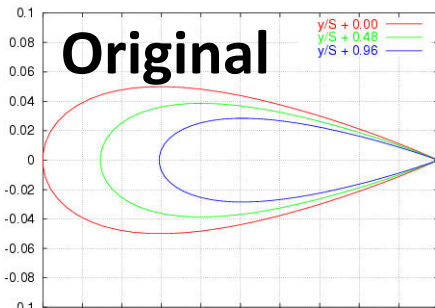
Each has certain specific requirements for obtaining accurate solutions of a surface ship wave field.

- Many good codes with many groups able to use the codes
- RANS having a larger role for viscous flow study
- Realistic geometries at model and full scale
- Expected to have larger role in the future with increasing experience and computer power
- Inroads to the **design process** (e.g. CFD on trial solutions) and to **Simulation Based Design (SBD)** being made



Minimize

(i) **Drag/Lift** and (ii) **cavitation volume** for two angles of attack



Global Optimization of an Anti Torpedo-Torpedo tail rudder

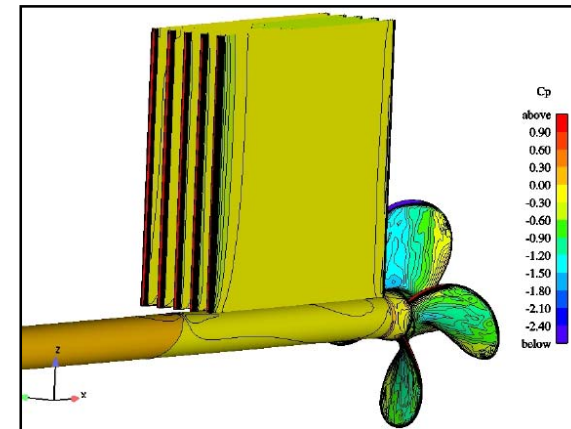
Current status in CFD - Propulsion

- Propulsion by CFD: challenges
- Propulsor flow: cavitation
- Cavitation: radiated pressures modelling
- From O.W. to propeller in behind conditions (hybrid RANS/BEM, local & global flow analysis)
- Validation data
- Analysis and design of propulsors

Propulsion by CFD: challenges

- Modelling by CFD marine propulsors is made complex by:
 - Geometry and kinematics of thrust-generation devices
 - Operating conditions in highly turbulent, vortical, unsteady flows
 - Cavitating flow features and related detrimental effects
 - Necessity to consider vessel and propeller as a unit
 - Demand for high-accuracy predictions to meet design requirements
 - Unconventional propulsors and layouts

Propeller behind wake generator
Italian Navy Cavitation Tunnel (CEIMM)



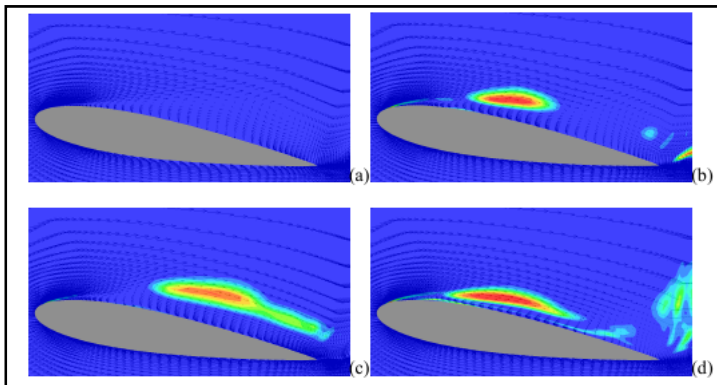
Simulation by RANS code FINFLO,
Sipila et al., VTT, Finland

Propulsion by CFD: a bit of history

- Current targets:
 - Compute propeller KT , KQ within 2-5% accuracy
 - Predict cavitation inception and analyse cavitating-flow dynamics
 - Describe off-design conditions
 - Simulate propelled vessel operations (propulsion test, manouvers, ...)
- Review of methodologies:
 - From early 1990' first applications of RANS to model non-cavitating propellers in uniform flow
 - Milestone: 22nd ITTC Workshop, 1998
 - By end of 1990' extensions to hull-propeller flows and to cavitation
- State-of-the-art:
 - RANS models being widely used for analysis (... and design?)
 - Commercial as well as in-house developed codes (most of the latter derived from existing hull-viscous flow codes)
 - Promising results by LES models

Propulsor flow: cavitation

- Interplay between
 1. Multiphase flow
 2. Turbulence & Vorticity
 3. Mesh adaptation



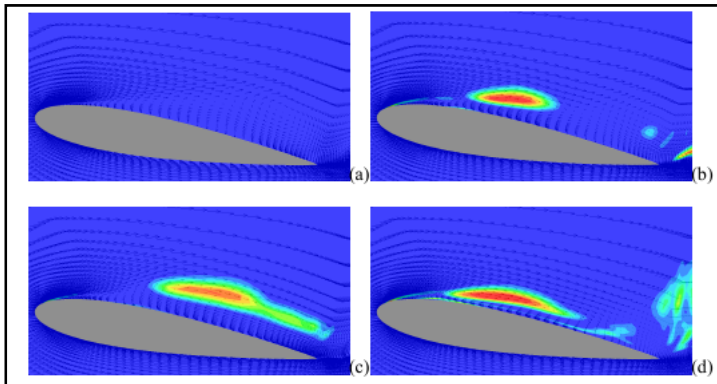
LES simulation by
OpenFOAM, Bensow et al.,
Chalmers Univ.

Current modeling efforts
toward the prediction of:

- Induced noise
- Pressure pulses
- Vibrations
- Erosion
- Efficiency reduction

Propulsor flow: cavitation

- Interplay between
 1. **Multiphase flow**
 2. Turbulence & Vorticity
 3. Mesh adaptation



LES simulation by
OpenFOAM, Bensow et al.,
Chalmers Univ.

Barotropic models:

- Arbitrary state eq. : $p=f(\rho)$
- Same continuity+momentum eqs. as non-cavitating flow
- Limit: no variable-density induced vorticity production

Multi-phase homogeneous mixture models:

Phases: water, vapor (in some models also non-condensable gas)

Interface capturing scheme: VoF

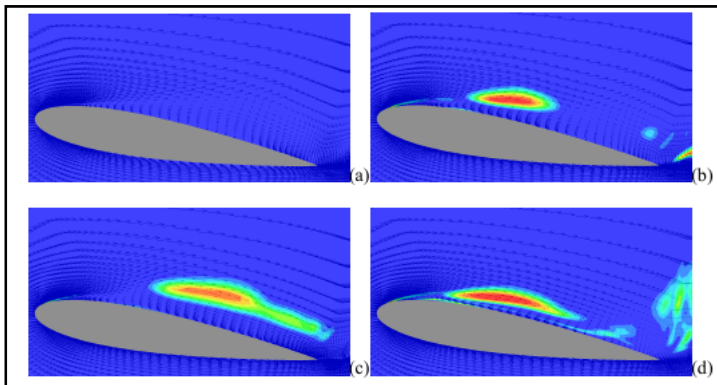
Transport equation for phases concentration (e.g., vapor volume fraction)

Key issue: vapor source and destruction terms (i.e., from R-P eq.)

Pressure-density coupling: pressure correction or artificial compressibility

Propulsor flow: cavitation

- Interplay between
 1. Multiphase flow
 - 2. Turbulence & Vorticity**
 3. Mesh adaptation



LES simulation by
OpenFOAM, Bensow et al.,
Chalmers Univ.

Turbulence models same as
for hull flow studies

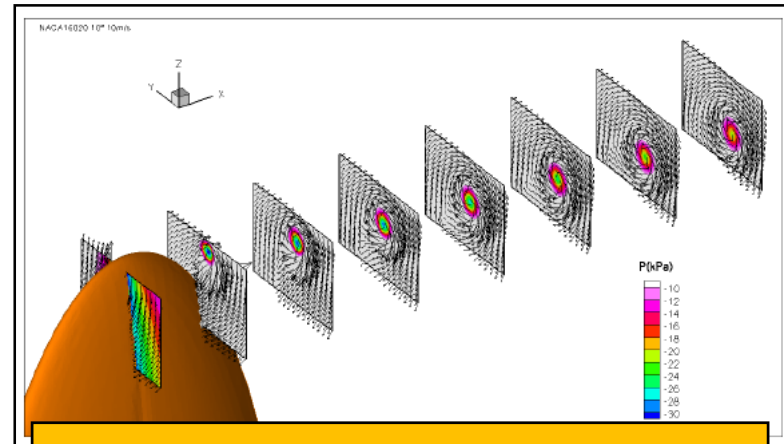
Peculiar for multi-phase flow
Correct description of small
time / space scales is crucial

Recent studies **suggest** the
opportunity to go for the LES

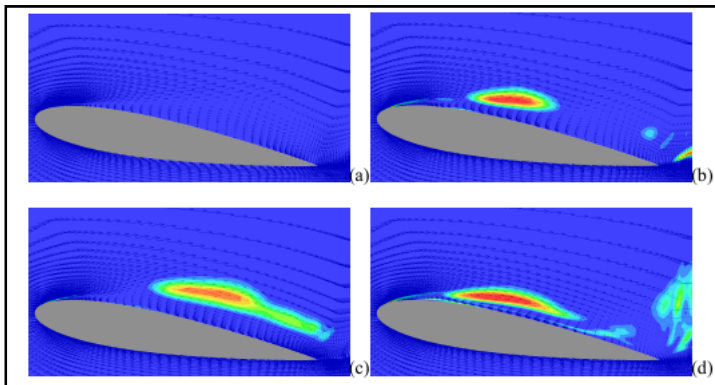
Computational costs **force** to
go for hybrid RANS / LES
model (DES, ...)

Propulsor flow: cavitation

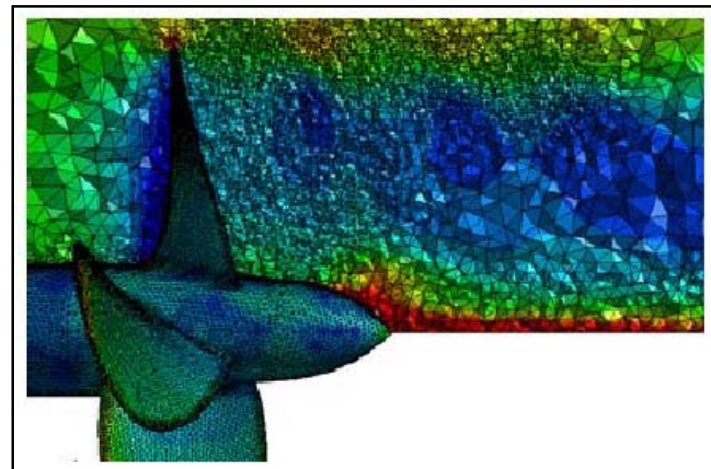
- Interplay between
 1. Multiphase flow
 2. Turbulence & Vorticity
 - 3. Mesh adaptation**



RANS code ISIS, Visonneau et.al., CNRS



LES simulation by OpenFOAM, Bensow et al., Chalmers Univ.

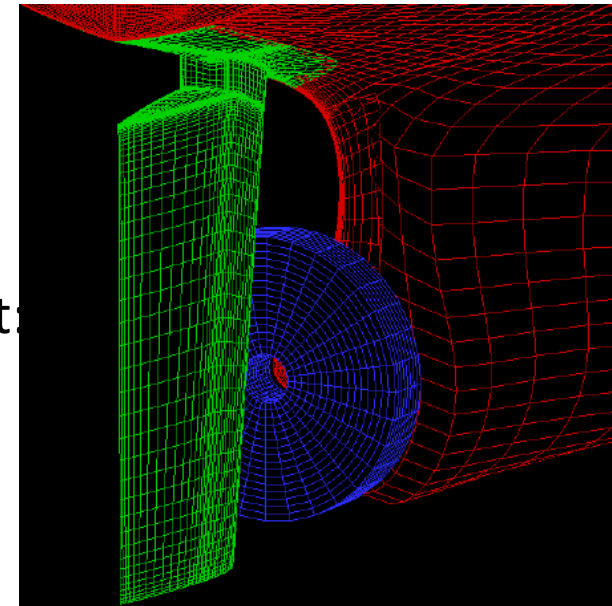


Cavitation: radiated pressures modelling

- Reference problem:
 - *Compute pressure fluctuations induced by propeller on plate hull*
 - *Propeller excitations at multiples of blade-passing frequency*
- Viscous-flow methods: **direct** computation of pressure field
 - **Scale-resolving is critical**: LES better than RANS
 - **Compressibility effects** should be taken into account
- Hydroacoustic models:
 - Excitation generation and propagation problems decoupled (see **ITTC Cavitation Committee** report for references)
 - Pressure pulses from wave-propagation equations (compressible flow)
 - Effect of solid boundaries through suitable scattering models

From O.W. to propeller in behind conditions

- Severe impact on grid generation and numerical scheme
 - Flow variables exchanged between rotating and fixed blocks
 - Sliding-mesh techniques
 - Correct transfer of fluxes across fixed/rotating interfaces
 - Parallel coding
- Flow unsteadiness: URANS solutions
- Simplified models to limit the computational effort
 - Quasi-steady RANS
 - Steady RANS with actuator-disk models
 - Hybrid RANS/BEM



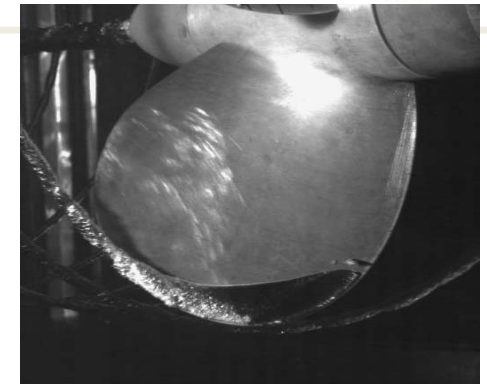
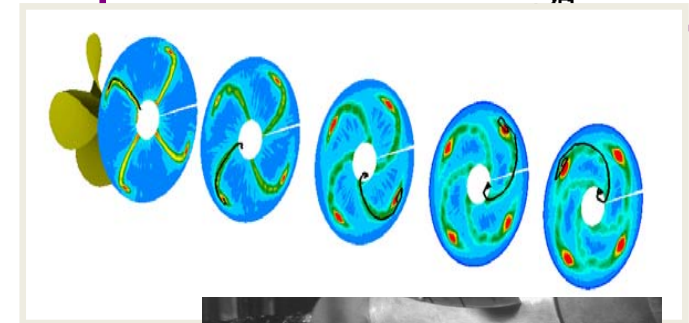
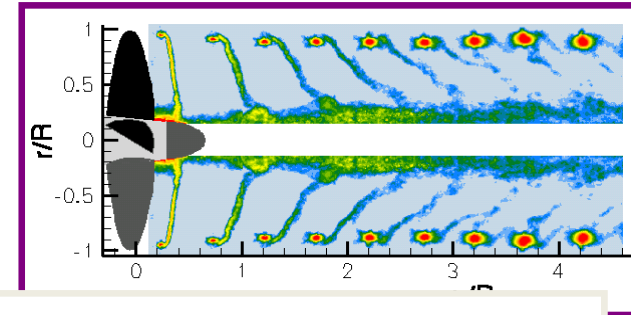
RANS code ChiNavis, Di Mascio et.al.,
INSEAN

Zoom in: hybrid RANS/BEM

- The concept of actuator disk revisited
- ‘Smart’ coupling of viscous and inviscid solvers:
 - *RANS to describe viscous flow around hull w/o propeller*
 - *Inviscid flow BEM to describe propeller flow*
- RANS-BEM coupling via generalized body-force approach
 - *Propeller action recast as source terms in the RHS of N-S equations*
 - *Intensity of source terms from propeller loading by BEM*
- Hull-propeller-rudder interactions by steady-RANS

Validation data: the INSEAN E779A dataset

- A comprehensive set of experimental data on propeller flow
 - Propeller O.W. characteristics
 - Wake field by LDV and PIV (velocity, vorticity, turbulence, ...)
 - Pressure/velocity correlations
 - Cavity pattern (uniform & non-uniform inflow)
 - Pressure pulses in cavitating flow
- Data presentation suitable for validation of CFD codes
- Several computational studies in the literature for comparisons
- Experimental activity in progress to expand dataset contents



Analysis and design of propulsors

- Impressive enhancements have been achieved in analyzing propulsors flow by CFD
- In contrast, the impact of CFD on design is still limited
- Standard approach still rely on designer's expertise and on inviscid-flow models: *lifting-line* , *vortex lattice methods*
- CFD limited to late-stage verifications (similar to model tests)
- True CFD-based design still missing
- Existing applications demonstrate that modern optimization techniques (multi-objective, multi-disciplinary, variable-fidelity models) can provide a sensible improvement of design techniques



Conclusions - propulsion

- RANS models widely used for isolated propeller flow studies
 - Open water characteristics reasonably accurate
- LES models being promising
 - Attempts to limit LES computational effort: hybrid LES/RANS
- Hull-propeller flow by fully RANS still very demanding
 - Hybrid RANS/Inviscid models appealing for hull-propeller studies
- Cavitation modelling under development
 - Reliable predictions of blade sheet cavitation
 - Current efforts to improve prediction of pressure pulses, erosion risk
- Examples of validity of CFD for extrapolation to full scale
- Impact of CFD into design to be increased