KCS Resistance Calculation
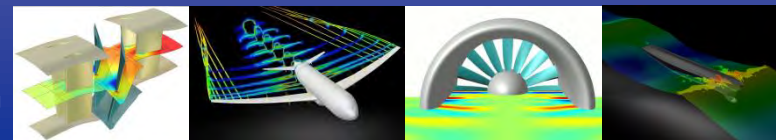
Author: Ludwig Kerner

Last update: 19-09-2014

Reviewed by : Jonathan Brunel

Date of Review : 19-09-2014

www.numeca.com

Copyright **© NUMECA International**
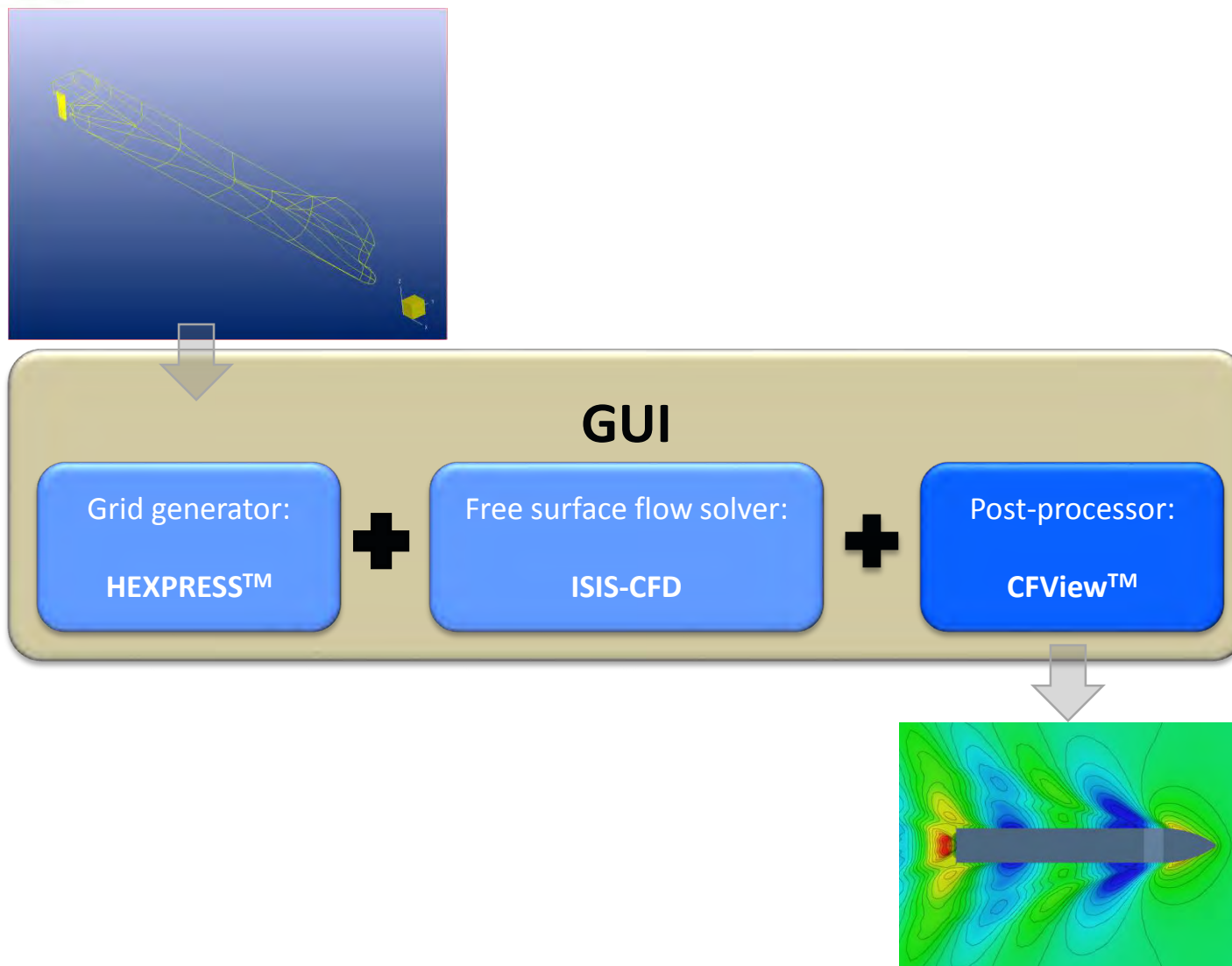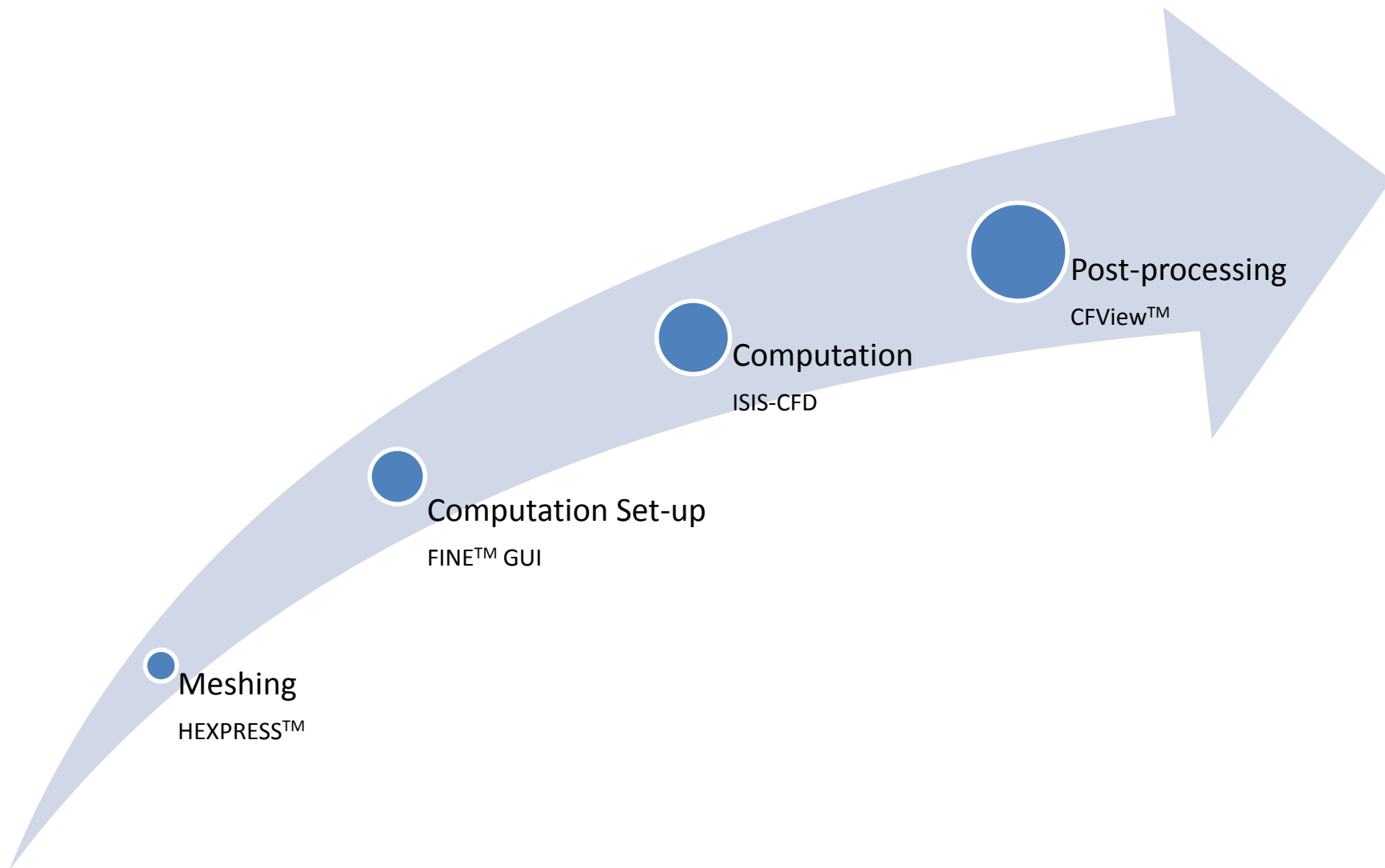
This document presents the CFD calculation of the resistance of the KCS. A model scale hull was simulated in upright conditions for different Froude numbers. The simulations (CAD import – meshing – computations – visualization) were performed with FINE^{TM}/Marine, NUMECA's Flow INtegrated Environment for marine applications, edited and developed by NUMECA in partnership with ECN (Ecole Centrale de Nantes) and CNRS (Centre National de la Recherche Scientifique).

The hull was downloaded in Parasolid format before importing it in HEXPRESS. The mesh was then generated with HEXPRESS^{TM} , NUMECA's full hexahedral unstructured grid generator integrated in FINE^{TM}/Marine.

Results were processed and analyzed with CFView^{TM}, NUMECA's Flow Visualization System integrated in FINE^{TM}/Marine.

GUI

Grid generator: **HEXPRESS™**	**+**	Free surface flow solver: **ISIS-CFD**	**+**	Post-processor: **CFView™**

Post-processing

CFView™

Computation

ISIS-CFD

Computation Set-up

FINE™ GUI

Meshing

HEXPRESS™

1

Test Cases Description

The geometry studied is a model scale of the KCS. The table below shows the main characteristics of the vessel at full scale.

Main particulars		Full scale
Length between perpendiculars	L_{PP} (m)	230.0
Length of waterline	L_{WL} (m)	232.5
Maximum beam of waterline	B_{WL} (m)	32.2
Depth	D (m)	19.0
Draft	T (m)	10.8
Displacement	Δ (m^3)	52030
Wetted area w/o rudder	S_W (m^2)	9424
Wetted surface area of rudder	S_R (m^2)	115.0
Block coefficient (CB)	$\Delta /(L_{PP}B_{WL} T)$	0.6505
Midship section coefficient (CM)		0.9849
LCB (%L_{PP}), fwd+		-1.48
Vertical Center of Gravity (from keel)	KG (m)	7.28
Metacentric height	GM (m)	0.60
Moment of Inertia	K_{xx}/B	0.40
Moment of Inertia	K_{yy}/L_{PP}, K_{zz}/L_{PP}	0.25
Propeller center, long. location (from FP)	x/L_{PP}	0.9825
Propeller center, vert. location (below WL)	$-z/L_{PP}$	0.02913

3D view of model

The computation referred to the **towing tank tests performed by MOERI on the KCS**. The length between perpendiculars is 7.2786 m (scale factor = 31.6) and the body draft is 0.3418 m. The water density considered is 999.1 kg/ m³. The position of the center of gravity along X axis has been estimated using the tool "domhydro" resulting in a location (X=-11.7, Y=0, Z=-0.115) in the global reference frame. The model contains the hull and the rudder of the KCS.

The first case referred to the case 2_1 of the 2010 Gothenburg workshop. The vessel is moving with a speed of 2.196 m/s (corresponding to a Froude number of 0.26) . In these cases, the trim and the sinkage are blocked. The geometry contains only the hull. The output of the first calculation will be :
- Wave elevation along longitudinal section
- Wave elevation along the hull
- Wave elevation contours
- Axial velocity and cross flow near the engine shaft

The second calculation referred to the cases 2_2a of the Gothenburg workshop. it concerns the resistance calculation of the KCS while it is moving with a speed of 2.196 m/s. The trim and the sinkage are still blocked. The only difference with the previous case is the rudder which is present for the second case and not for the first. The output of this calculation will be :
- Drag comparison with experimental data
- Visualization of the free surface
- Wetted area
- Hydrodynamic pressure on the hull

The third case referred to the case 2_2b of the Gothenburg workshop. It concerns the resistance calculation of the KCS at different speeds from 0.92 m/s to 2.38 m/s (corresponding to a variation of the Froude number from 0.1083 to 0.2816) . In this case, the trim and the sinkage are solved. In terms of output, the following will be presented:

- Drag, Trim and Sinkage comparison with experimental data
- Visualization of the free surface
- Wetted area
- Hydrodynamic pressure on the hull

As the conditions of all cases are symmetric only half of the geometry is simulated.

2 Mesh Generation

The Parasolid file is loaded into HEXPRESS^{TM}. A computational domain is constructed by defining a box around the model.

Commonly, we take the following dimensions away from the model in terms of L_{pp} :

Front	1 x L_{pp}
Back	3 x L_{pp}
Top	0.5 x L_{pp}
Bottom	1.5 x L_{pp}
Each side	1.5 x L_{pp}
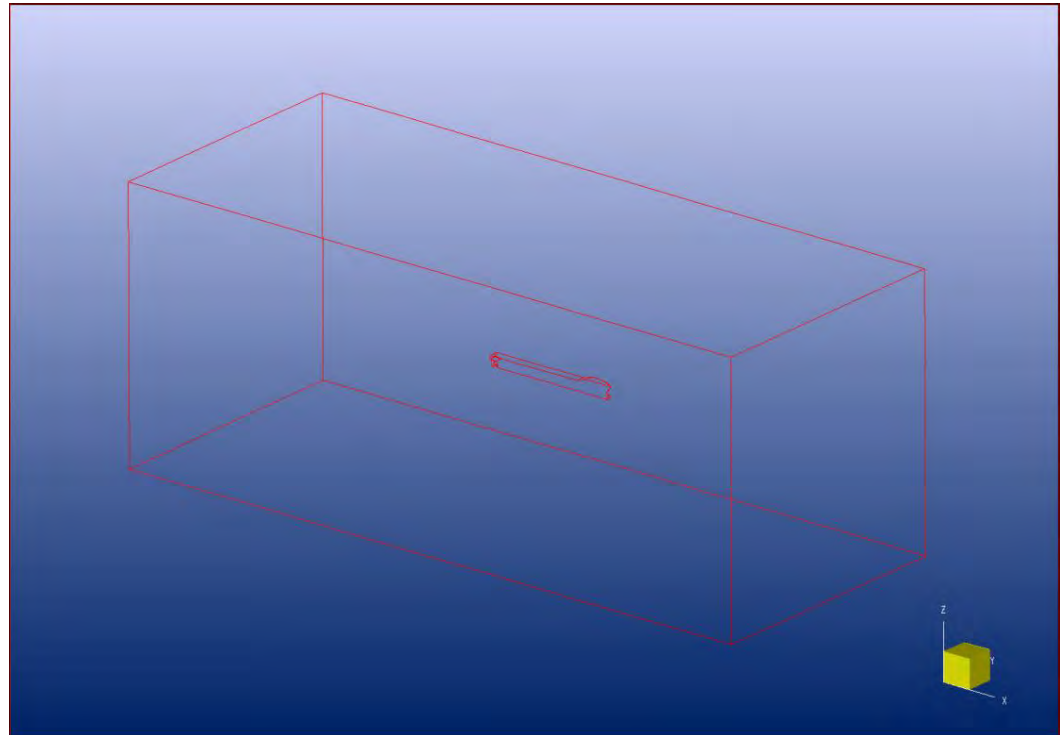
Mesh generation in HEXPRESS^{TM} is done using a five-steps wizard:

Step 1:Initial mesh
an isotropic, Cartesian mesh is generated.

Step 2: Adapt to geometry
the mesh is refined in regions of interest by splitting the initial volumes.

Step 3: Snap to geometry
The volumic, refined mesh is snapped onto the model.

Step 5:
To capture viscous effects, this step inserts viscous layers in the Eulerian mesh.

Step 4:
After snapping, the mesh can contain some negative, concave, or twisted cells. This step will fix those cells and increase the quality of the mesh.

Two meshes are needed. A first one without the rudder for the case 1 and a second mesh with the rudder for the case 2 and 3. The meshes are quite fine and have the following properties:

- The initial mesh size is 2500 cells

- An internal surface is used to spatially discretize the region around the initial free surface. This surface is located at z = 0,3417m and covers the whole domain. Mesh refinement normal to the free surface is set at 0.07m (L_{pp} / 1000). A local diffusion of 4 is used to ensure the free surface capturing during the computation.

- For the viscous layers, the initial spacing is set at 1.476e-3m (ensuring a y^+ of about 30). The stretching ratio is set at 1.2. An inflation technique is used to find the optimum number of layers, ensuring a smooth transition from viscous inner to outer Eulerian mesh. The viscous layers are inserted using the maximum velocity as reference for both cases, as the same meshes will be used to run the complete set of conditions in the future.

In the table below, the mesh characteristics for the vessel with rudder are listed. The repartition of cells for different criteria are also shown.

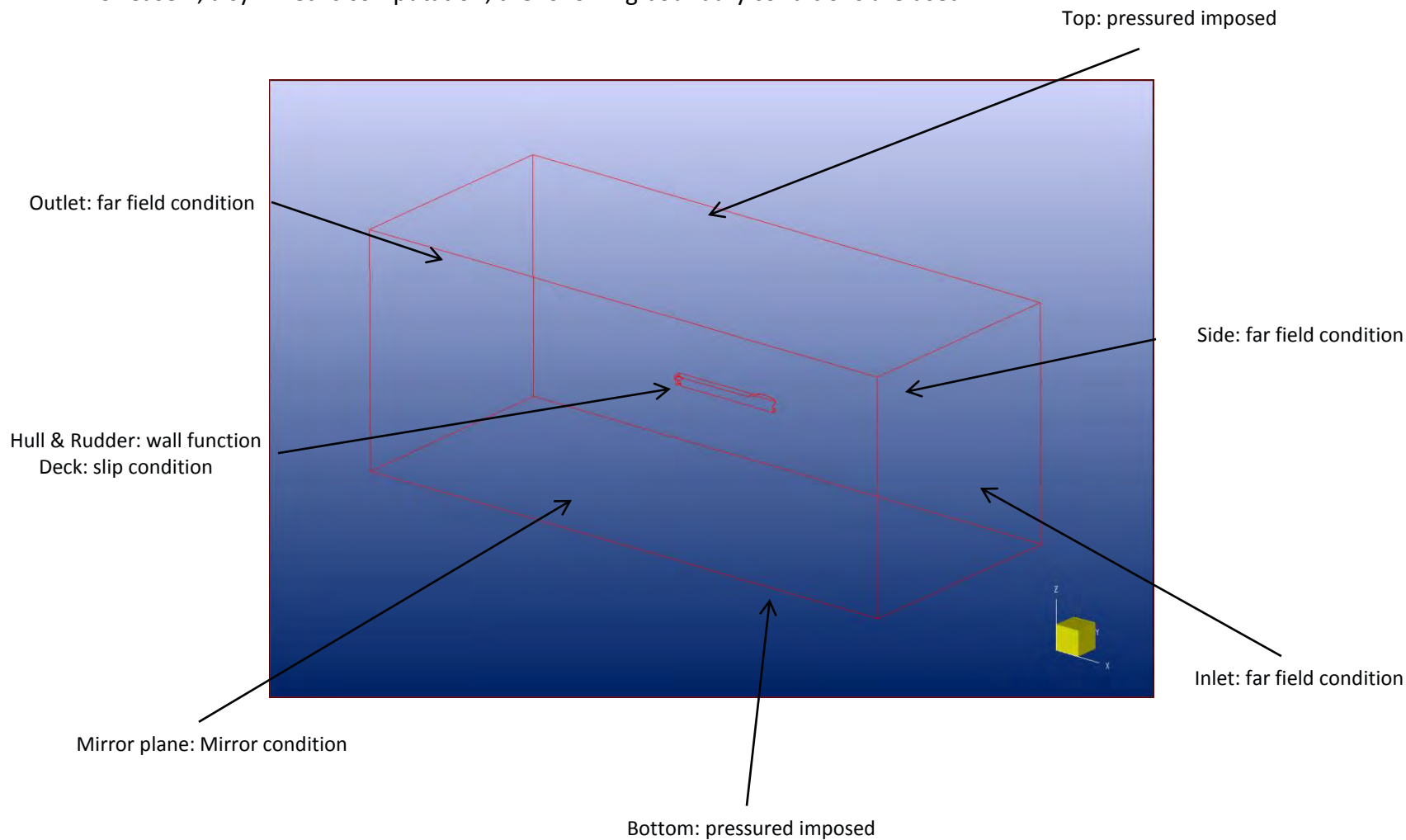| Mesh characteristic	
Number of cell in Eulerian mesh [-]	1,724,459
Number of cells with viscous layers [-]	2,173,232
Minimal orthogonality [deg]	13.71
Maximal aspect ratio [-]	136.85
Maximal expansion ratio [-]	7.62
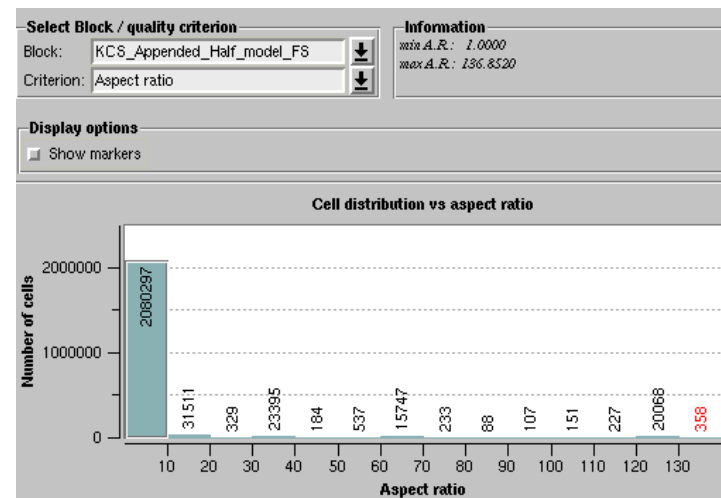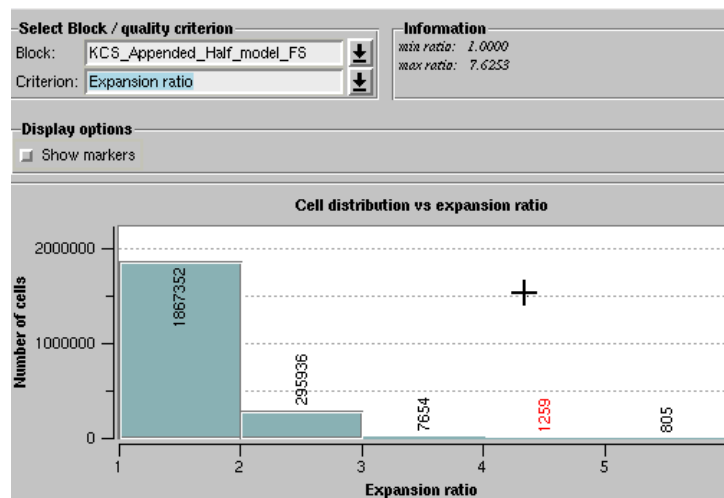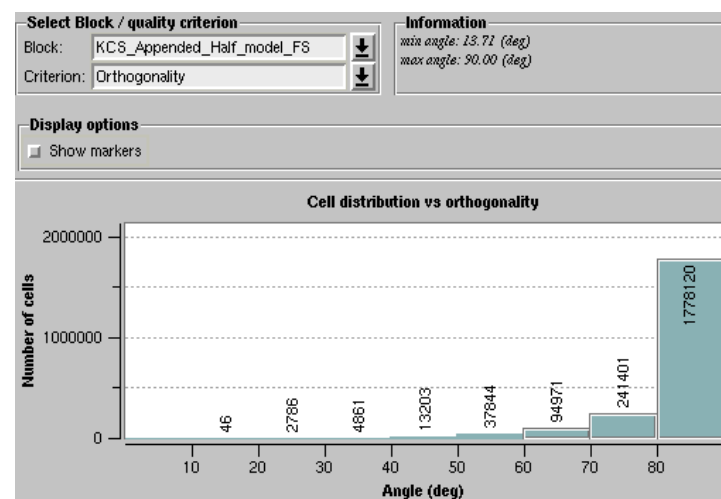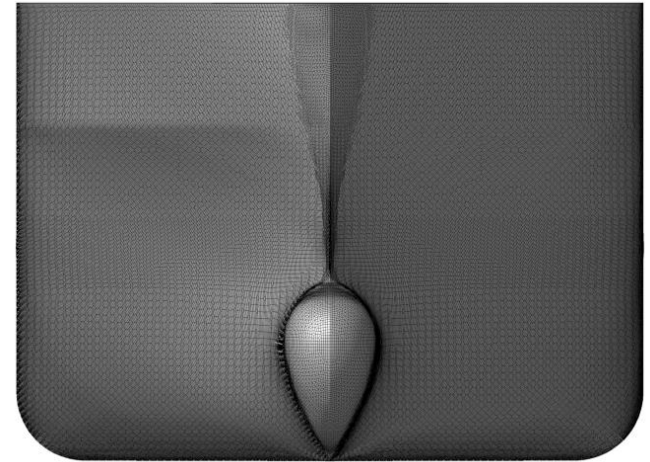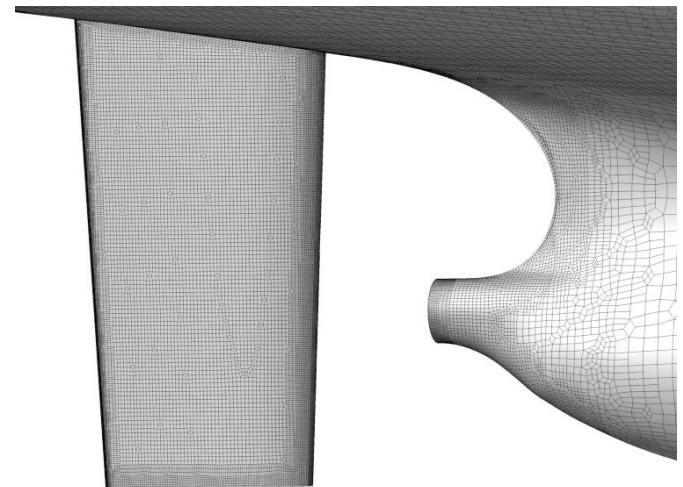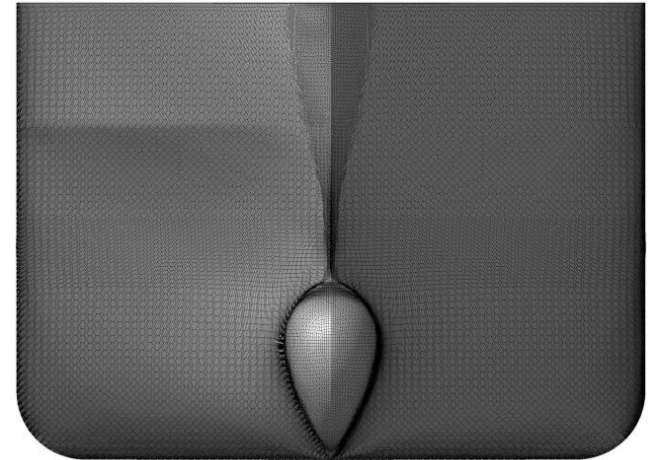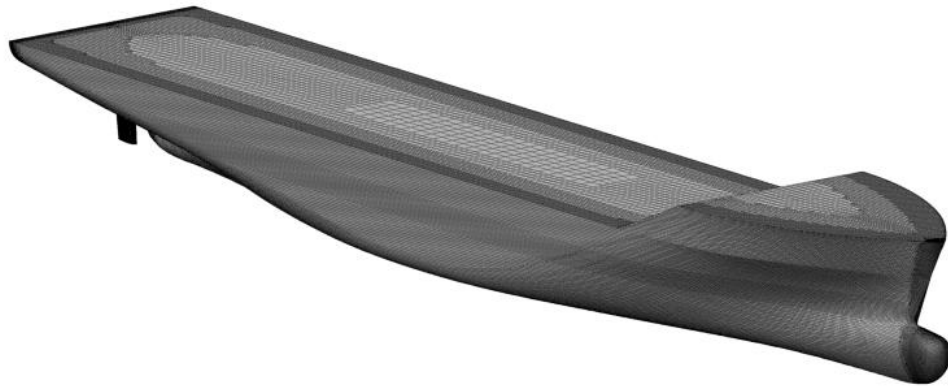
3 Computations

The FINE/Marine solver main features are:

- 3D, face –based approach, pressure equation formulation (SIMPLE)
- Free surface capturing strategy with high-resolution interface schemes
- Spatial discretization: upwind, hybrid, centered, blended, and Gamma (GDS)
- Time discretization: steady, 1st and 2nd order backward schemes
- Adaptive grid refinement
- External forces (towing and wind effects), mooring or tugging lines
- 6 degrees of freedom with strong coupling, mesh deformation algorithm
- Modeling of propeller using actuator disk theory
- Sliding grids for multi-domain simulations
- Euler and Navier-Stokes flows, Laminar and turbulent, steady and unsteady flow problems
- Turbulence modeling:
    - Spalart-Allmaras one-equation model
    - Launder-Sharma k-ε model
    - SST k-ω model
    - BSL k-ω model
    - Wilcox k-ω model
    - EASM (Explicit Algebraic Stress Model)
- All models can be used with low Reynolds formulation, wall functions, or rotation correction, except the Spalart-Allmaras model (only available in low-Reynolds formulation)
- On top of FINE^{TM}/Marine native formats, the solver also reads ICEM CFD and Gridgen meshes. Export to Tecplot, EnSight, FieldView and CGNS formats

For Case 1, a symmetric computation, the following boundary conditions are used:

Top: pressured imposed

Outlet: far field condition

Side: far field condition

Hull & Rudder: wall function
Deck: slip condition

Inlet: far field condition

Mirror plane: Mirror condition

Bottom: pressured imposed

Copyright **© NUMECA International**

The computations settings common to the two cases are:

	Ship characteristics
Lref (m)	7.2786
Y+	30
Mass (Kg)	1652
Coordinates of the CoG (m)	(3.53, 0.0,0.23)
Initial free surface location (m)	0.3418

Common settings:

Time scheme: backward order 1

- K-omega SST turbulence model with wall functions
- Multi-fluid computation:

WATER: - Dynamic viscosity: 1.0122 *(N s/m²) x 10⁻³*

$\qquad$ - Density: 999.1 *kg/m³*

AIR: $\quad$ - Dynamic viscosity: 1.85 *(N s/m²) x 10⁻⁵*

$\qquad$ - Density: 1.2 *kg/m³*

For the case 1 and 2, trim and sinkage are blocked. Only one speed is studied and the settings is summarized below :

Case 1 and case 2	
Vref [m/s]	2.196
Froude	0.2599
Time step [s]	0.018
Acceleration time [s]	3.51

For the case 3, trim and sinkage are solved. 6 speeds are studied to plot the resistance curve of the KCS. The table below summarizes the settings :

Case 3						
Vref [m/s]	0.915	1.281	1.647	1.921	2.196	2.379
Froude	0.1083	0.1516	0.1949	0.2274	0.2599	0.2816
Time step [s]	0.042	0.030	0.023	0.020	0.018	0.016
Acceleration time [s]	8.43	6.02	4.69	4.02	3.51	3.24

4.1

Results case 1

The wave elevation on the hull has been measured along a section situated at y/Lpp = 0.0741. The experimental values (EFD) and computational results (CFD) results are superimposed to allow comparison.

Y/Lpp = 0.0741


Wave elevation at y/Lpp = 0.0741
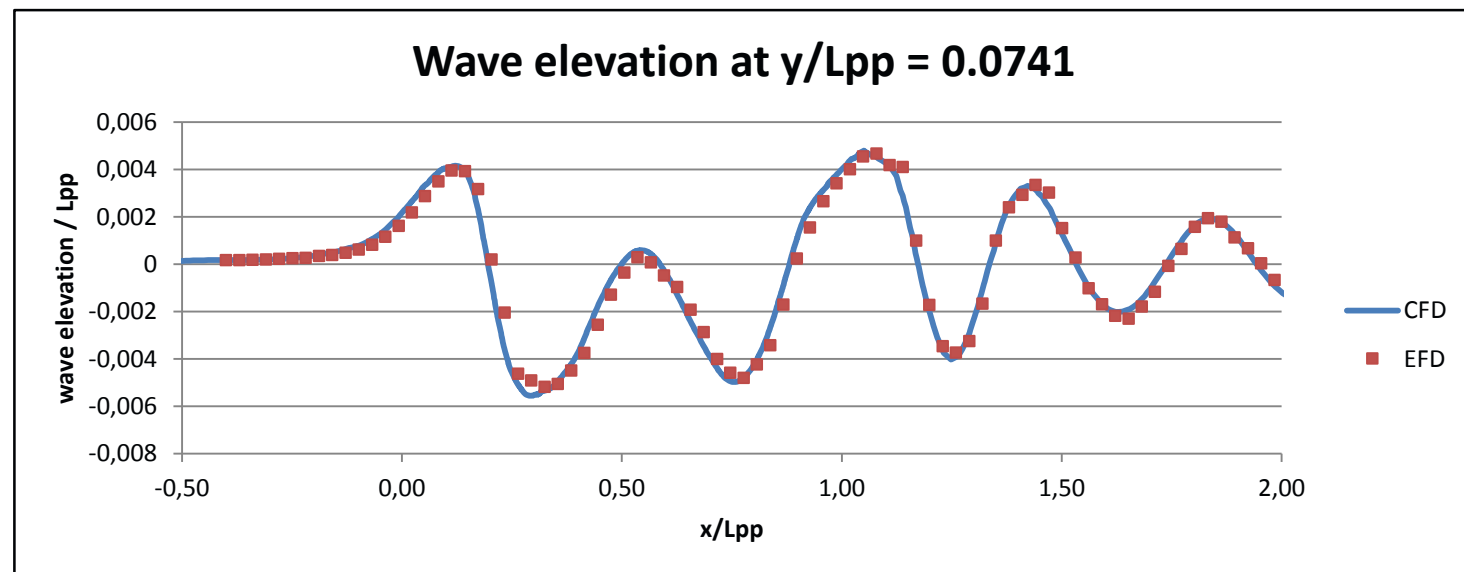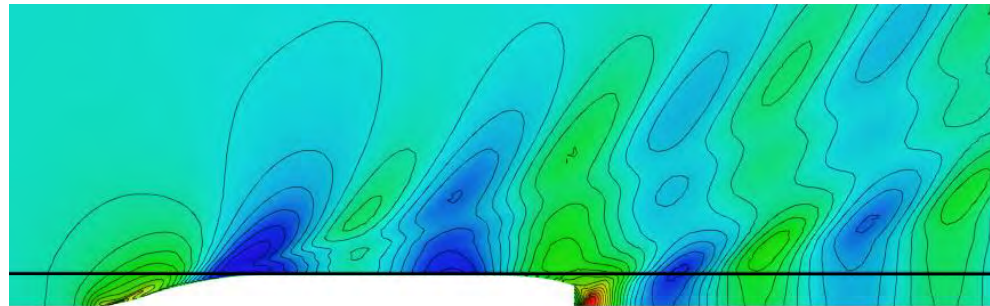
Wave elevation

The wave elevation on the hull has been measured along a section situated at y/Lpp = 0.1509. The experimental values and CFD results are superimposed to allow comparison.

Y/Lpp = 0.1509

wave elevation at y/Lpp = 0.1509

Copyright **© NUMECA International**
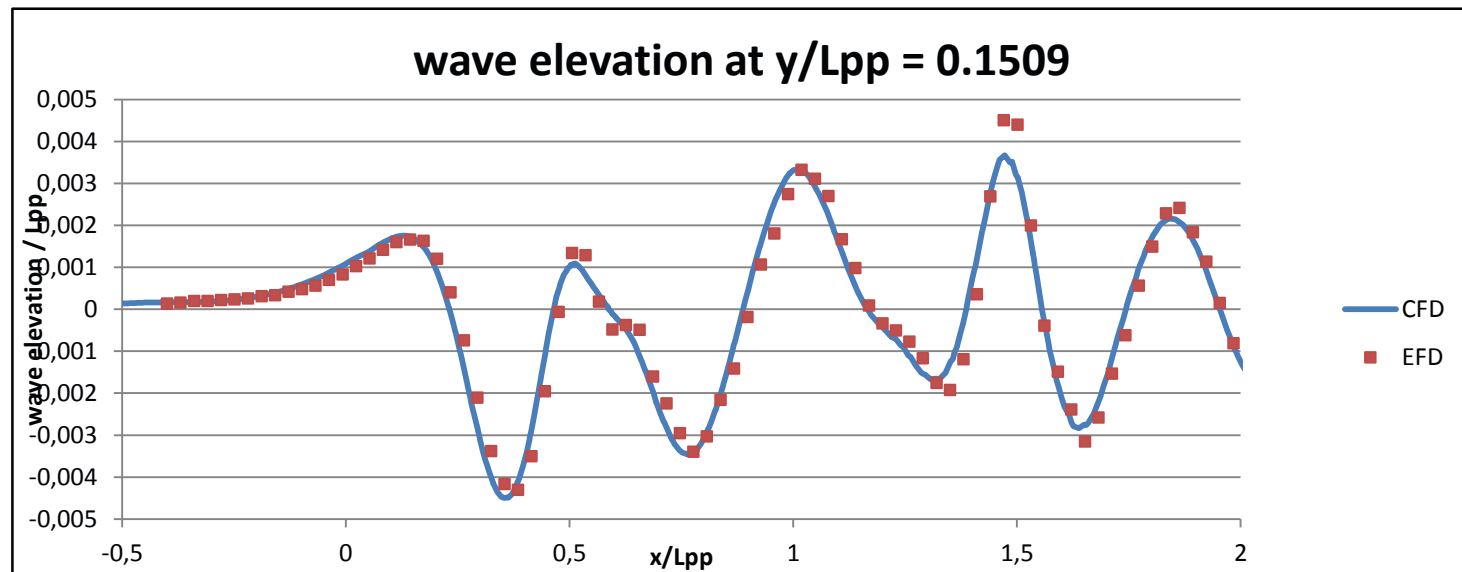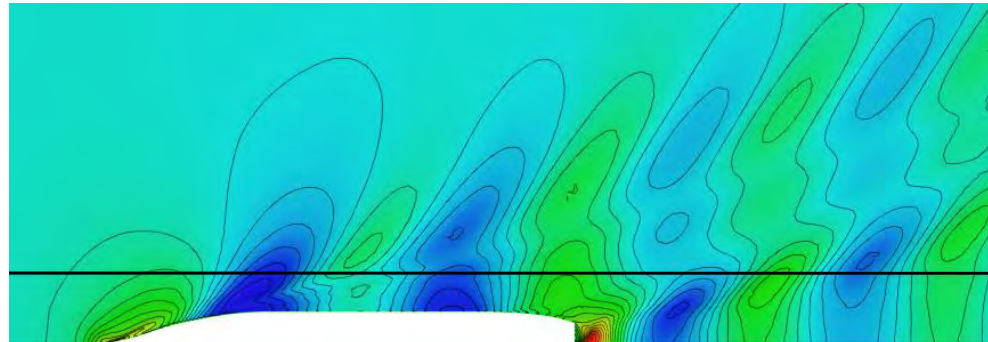
Wave elevation

The wave elevation on the hull has been measured along a section situated at y/Lpp = 0.4224. The experimental values and CFD results are superimposed to allow comparison.

Y/Lpp = 0.4224

Wave elevation at y/Lpp = 0.4224
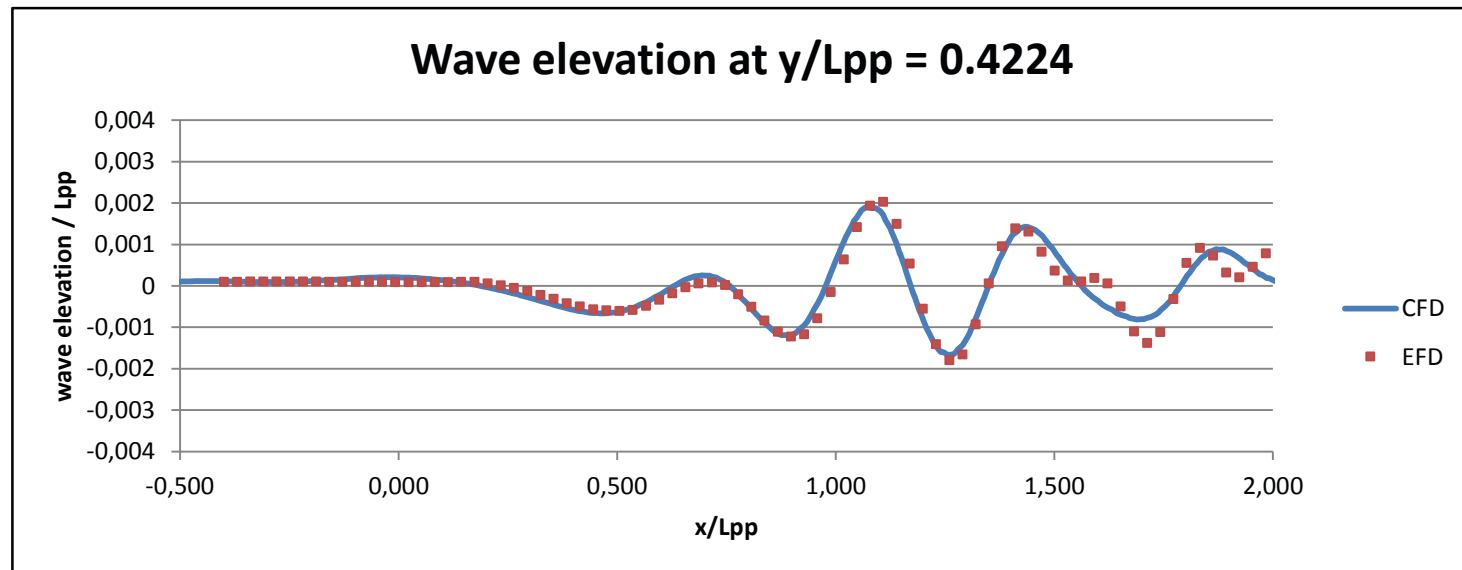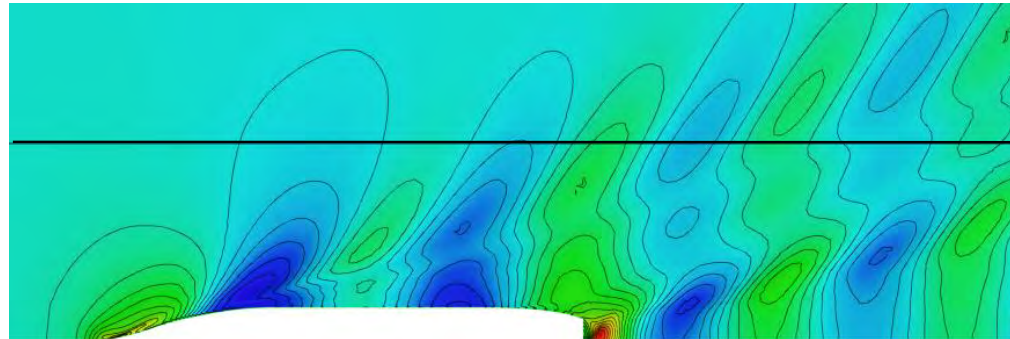
The wave elevation on the hull has been measured and plotted. The experimental values are superimposed to this result to allow comparison.
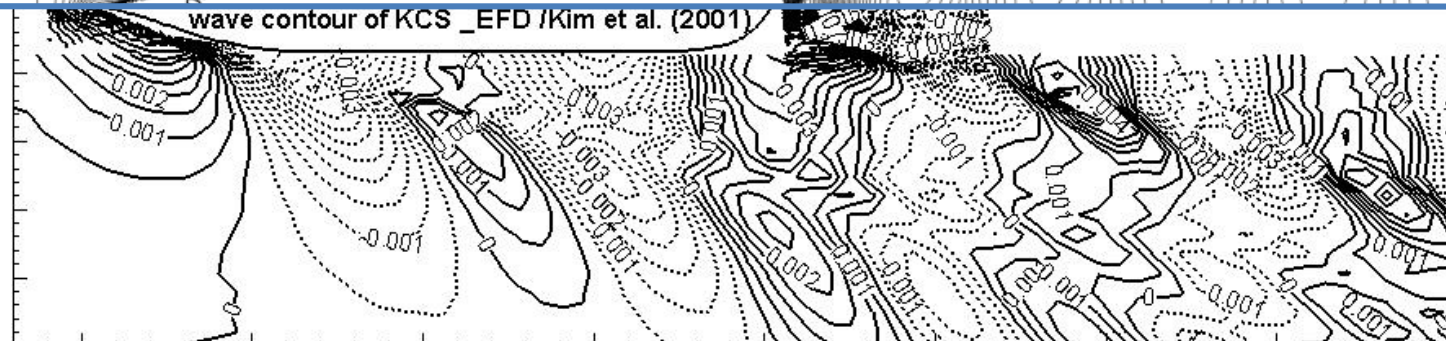

Wave elevation on the hull
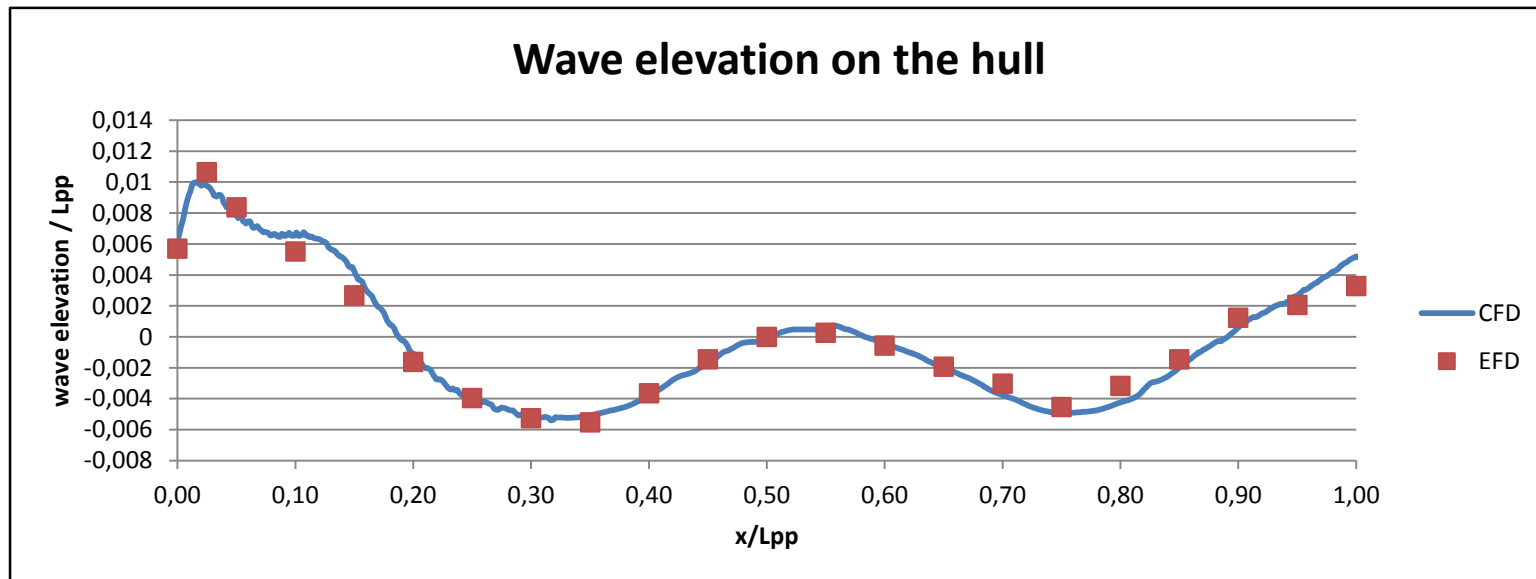
Wave elevation

The views below shows the wave elevation divided by Lpp. Above is the FINE™/Marine computation and below is the experimental result provided to the participants of the Gothenburg workshop in 2010.

FINE™/Marine
result

Experimental
result

Speed field
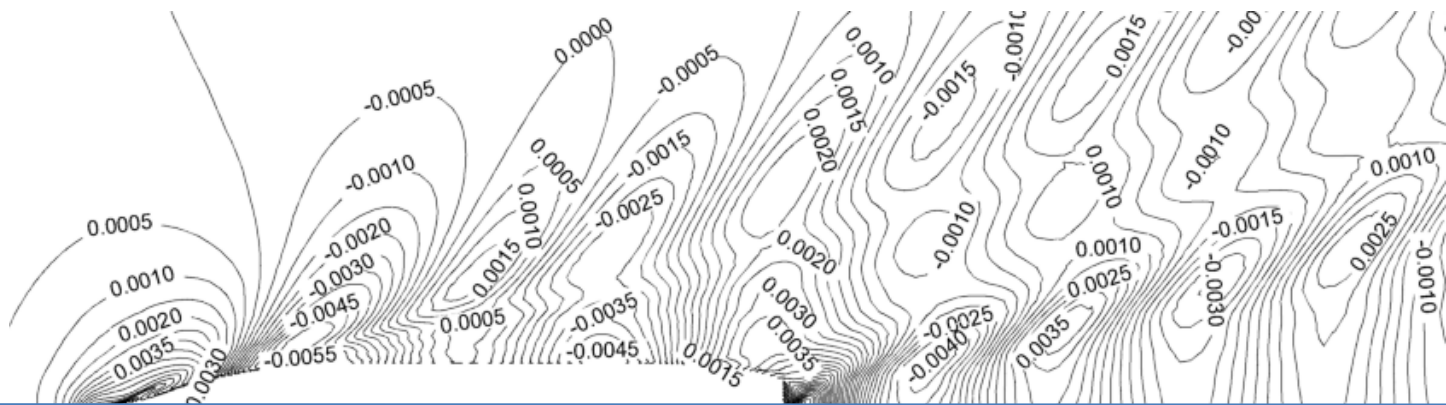
The views below show the relative axial velocity divided by Lpp on the right panels and cross flow vectors and streamlines on the left panels. The cutting plane is situated at x/Lpp = 0.9825 from the AP. The view above has been realized in CFView and the view below is the experimental result.

FINE™/Marine
result

Experimental
result

4 .2

Effect of the turbulence model

Comparison of wave elevation
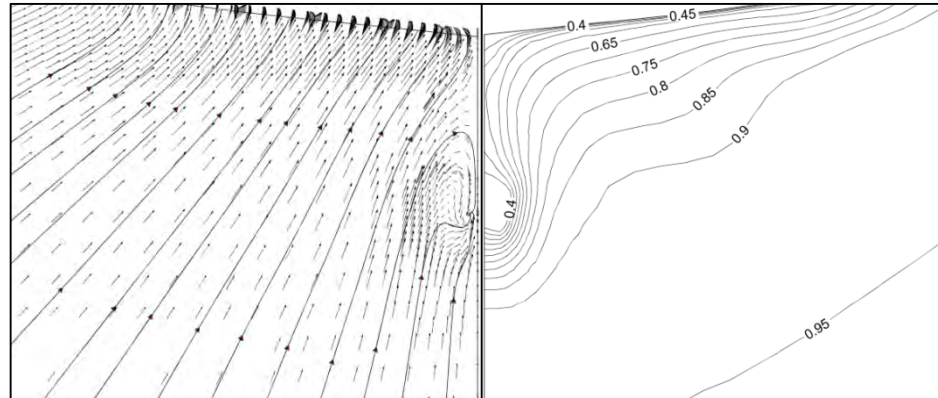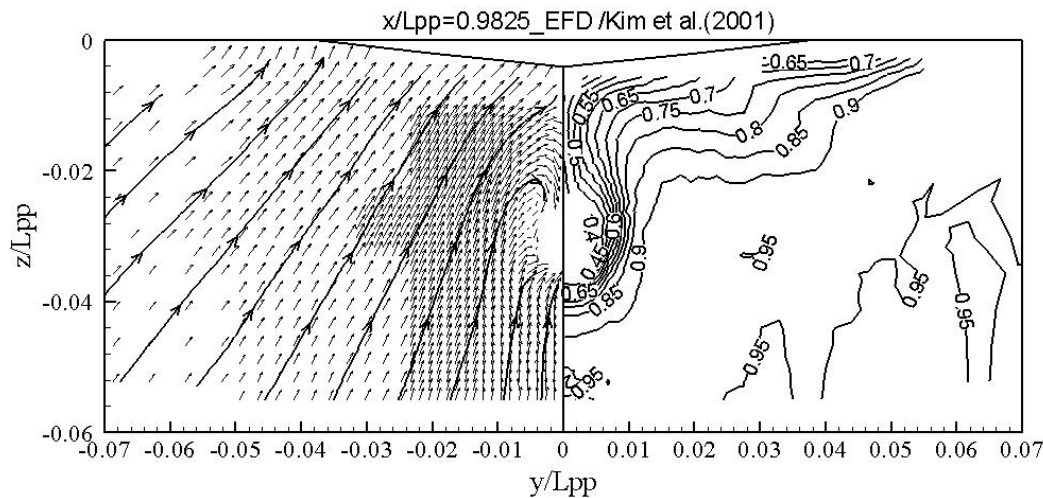
In this section, we test the effect of the turbulence model on the result of the case 1. To this aim, we relaunch the calculation with the EASM model instead of the kω-SST. The same post processing as for the previous calculation is done and results are compared. The charts below compare wave elevation along sections obtained with different turbulence model and the experimental data. We can see that both models give exactly the same results.

Comparison of wave elevation

The charts below compare wave elevation along sections obtained with different turbulence model and the experimental data. We can see that both models give exactly the same results.

Wave elevation at y/Lpp = 0,4224

Wave elevation on the hull
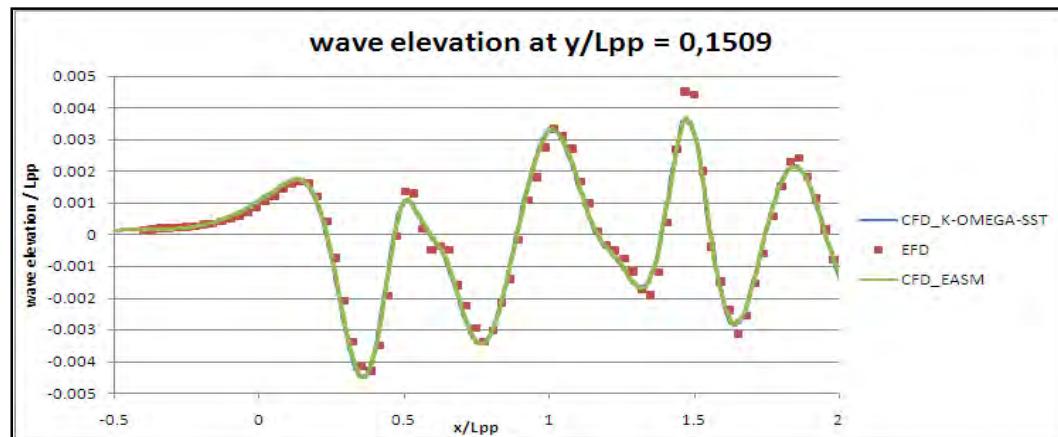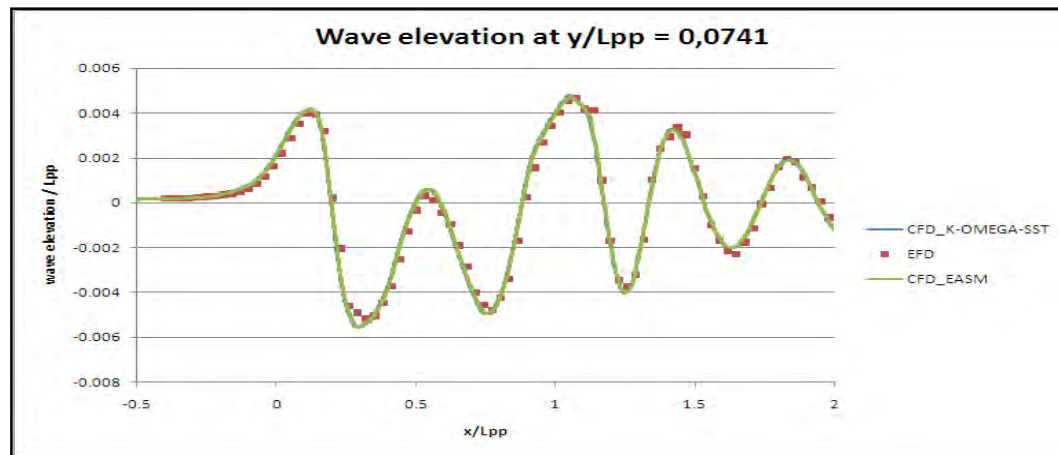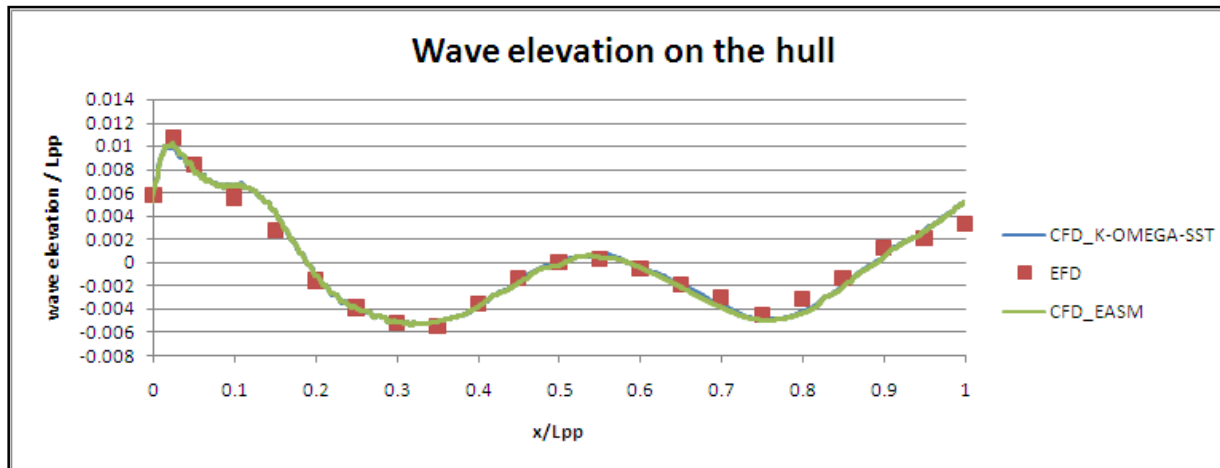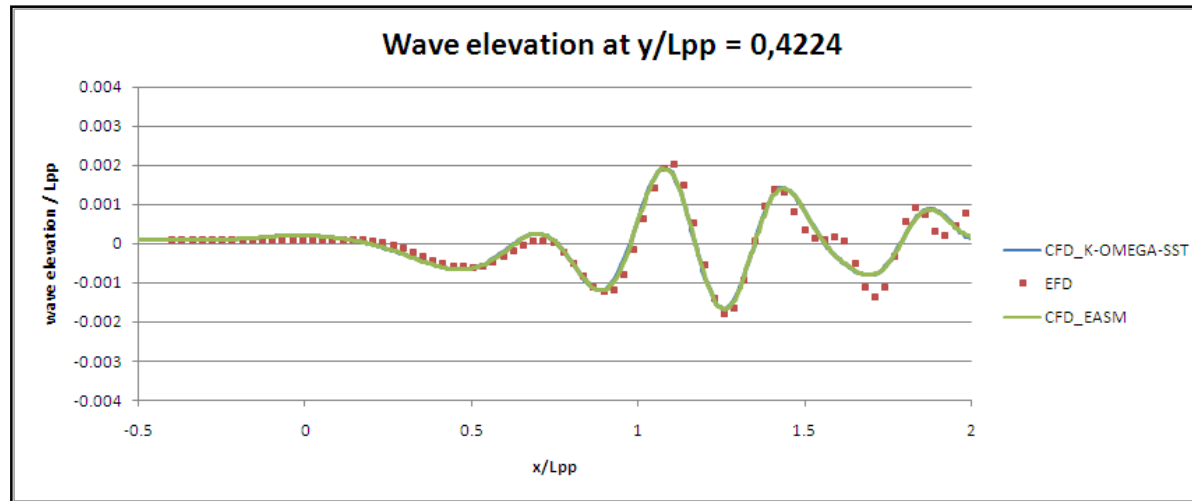
The views below shows the wave elevation divided by Lpp. Above is the result obtained with the turbulence model K-ω-SST and below with the EASM model. We can see that the isolines match perfectly. The free surfaces are quite the same.
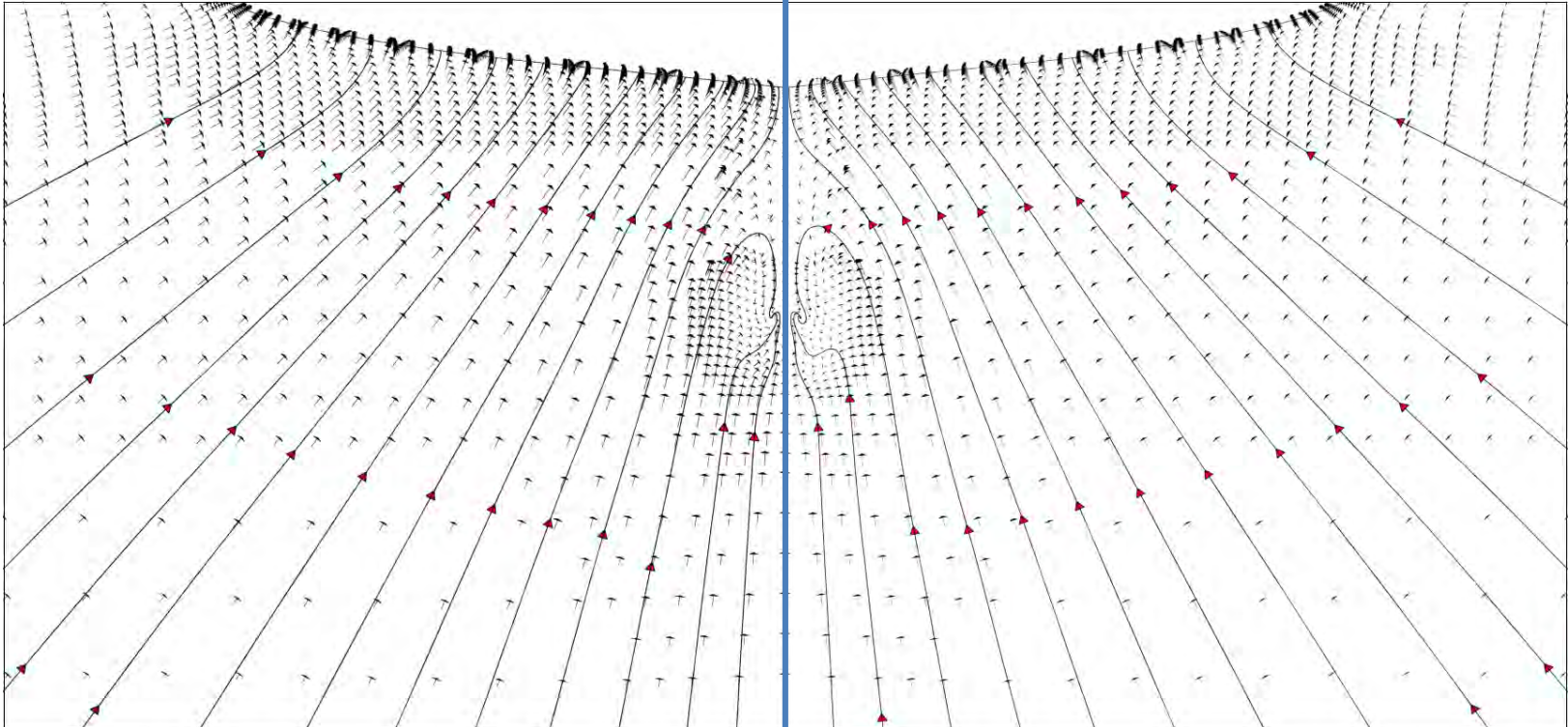
FINE™/Marine
K-ω-SST

FINE™/Marine
EASM
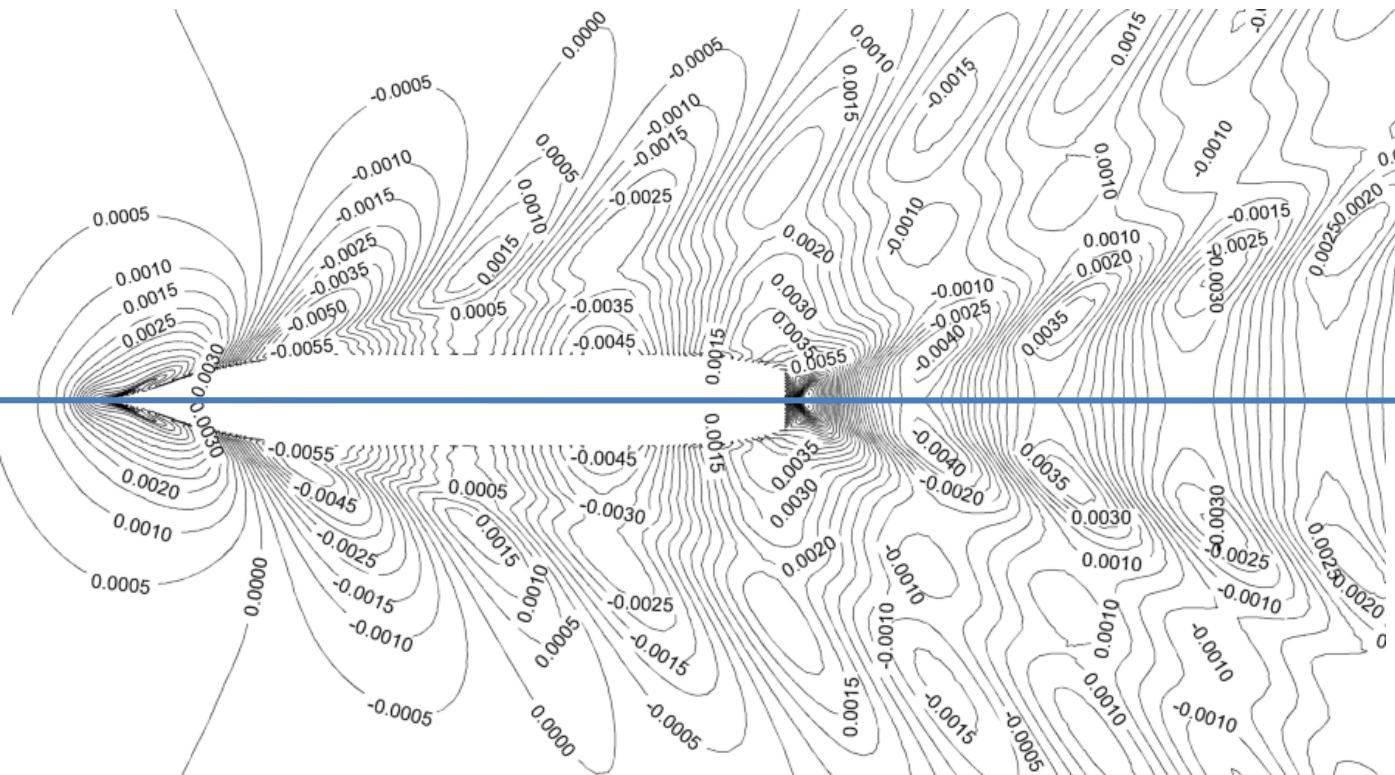
The views below show the cross flow vectors and streamlines. The cutting plane is situated at x/Lpp = 0.9825 from the AP. The speed field is quite similar. Negligible differences can be observed in the center of the picture (in the wake of the shaft).
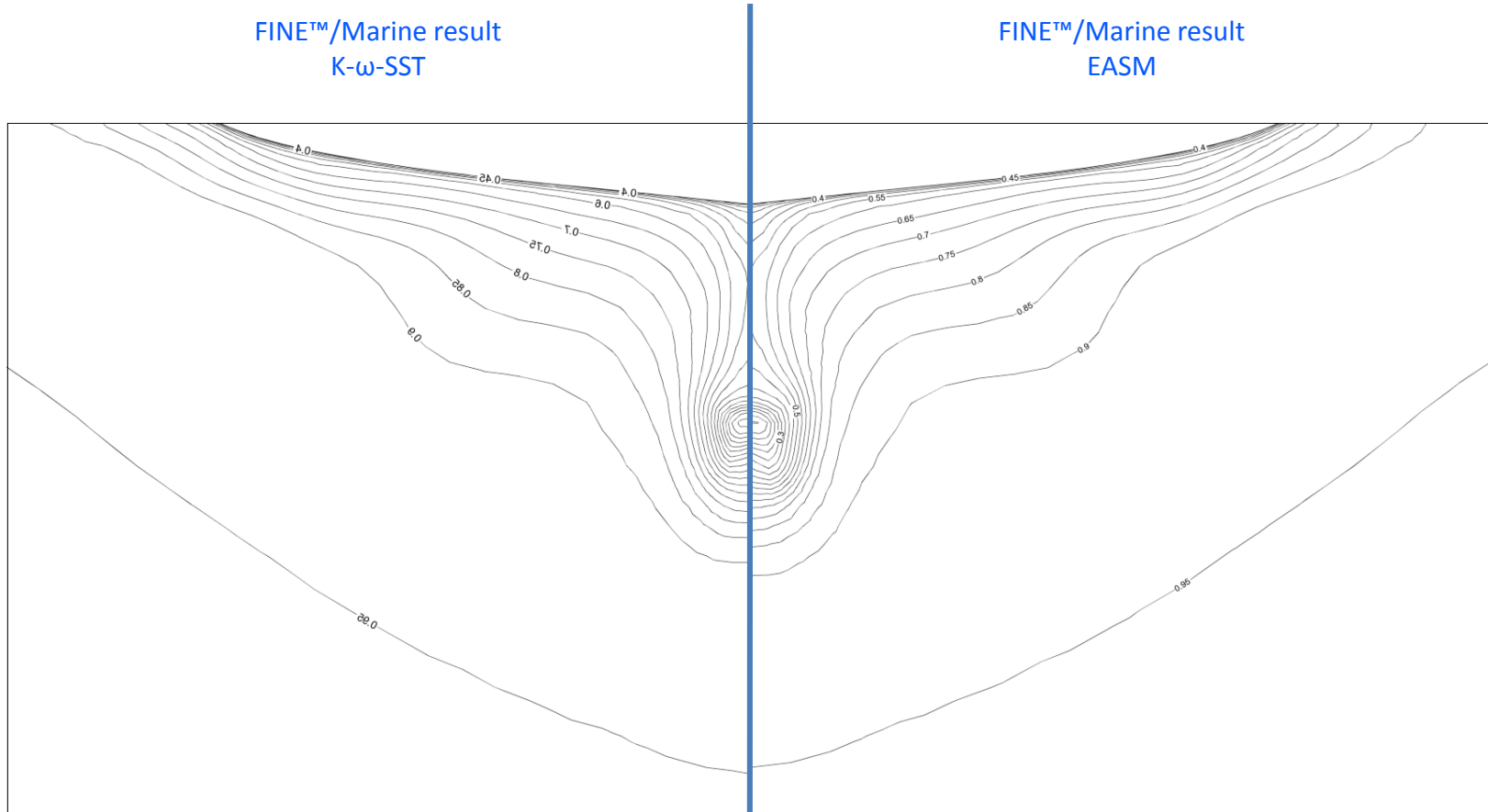
FINE™/Marine result
K-ω-SST

FINE™/Marine result
EASM

The views below show the relative axial velocity divided by Lpp. The cutting plane is still situated at x/Lpp = 0.9825 from the AP. The shapes of the isolines are similar but we can observe little differences.

FINE™/Marine result
K-ω-SST

FINE™/Marine result
EASM

4.3 Results case 2

Convergence

Overview of the convergence history

Case 1 : Convergence history

Converged simulation (less than 2% oscillation)

Converged simulation (less than 1% oscillation)
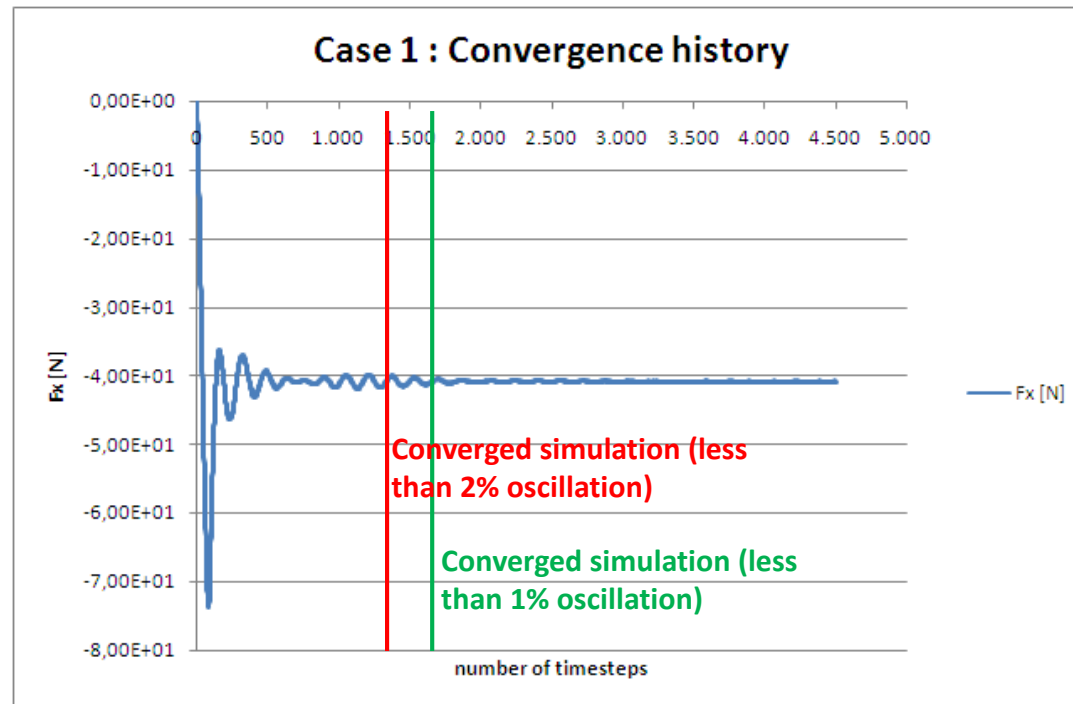
Computation time to convergence :
-Convergence 1 %: 9 h on 16 partitions
-Convergence 2 %: 8 h on 16 partitions

The final value of the drag is taken as twice (because the calculation is for half of the ship) the average of Fx over 10 last iterations. This leads to -81.8 N.

We can defined a drag coefficient with this relation :

$$C_t = \frac{2F_x}{0.5\rho SV^2}$$

Where S is the wetted area at static position and S/Lpp = 0.1803 with the rudder

Fx [N]	Total drag [N]	Ct_CFD X10³	Ct_EFD X10³	ERROR %
-40.9	-81.8	3.551	3.557	0.17

Wave Elevation

Copyright **© NUMECA International**
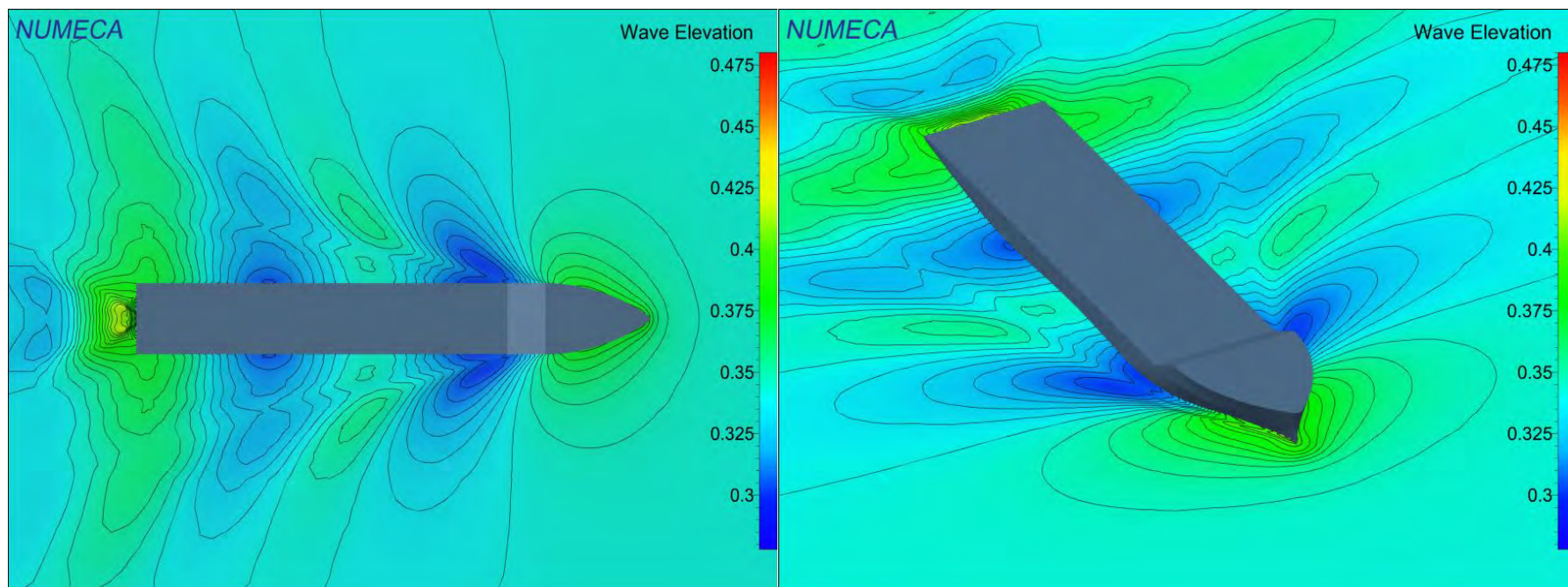
Wetted Surface
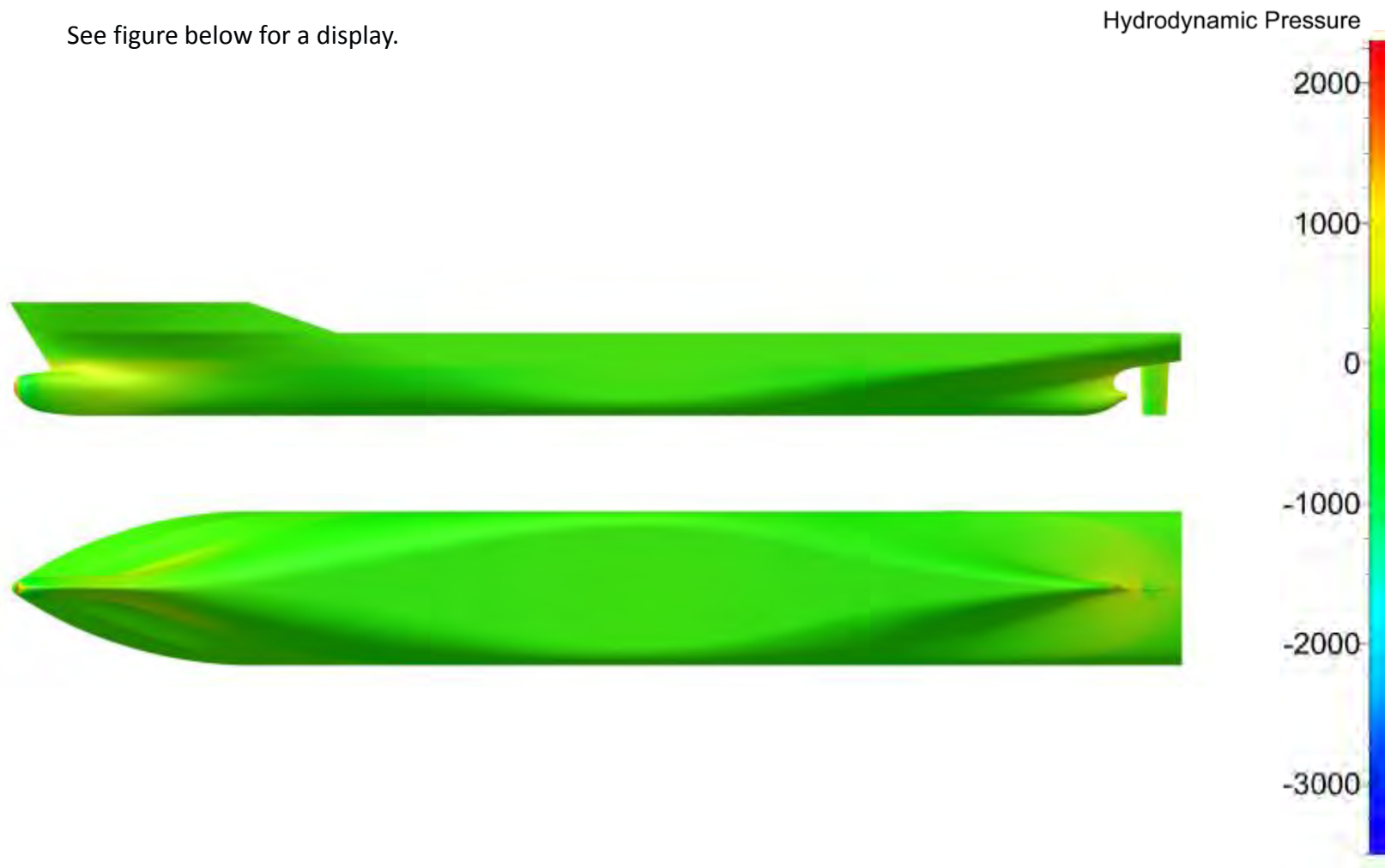
See figure below for a display (blue is air, red water).
In CFView, the wetted surface has been computed and
is equal to 4.91 m².

Copyright **© NUMECA International**

Hydrodynamic Pressure
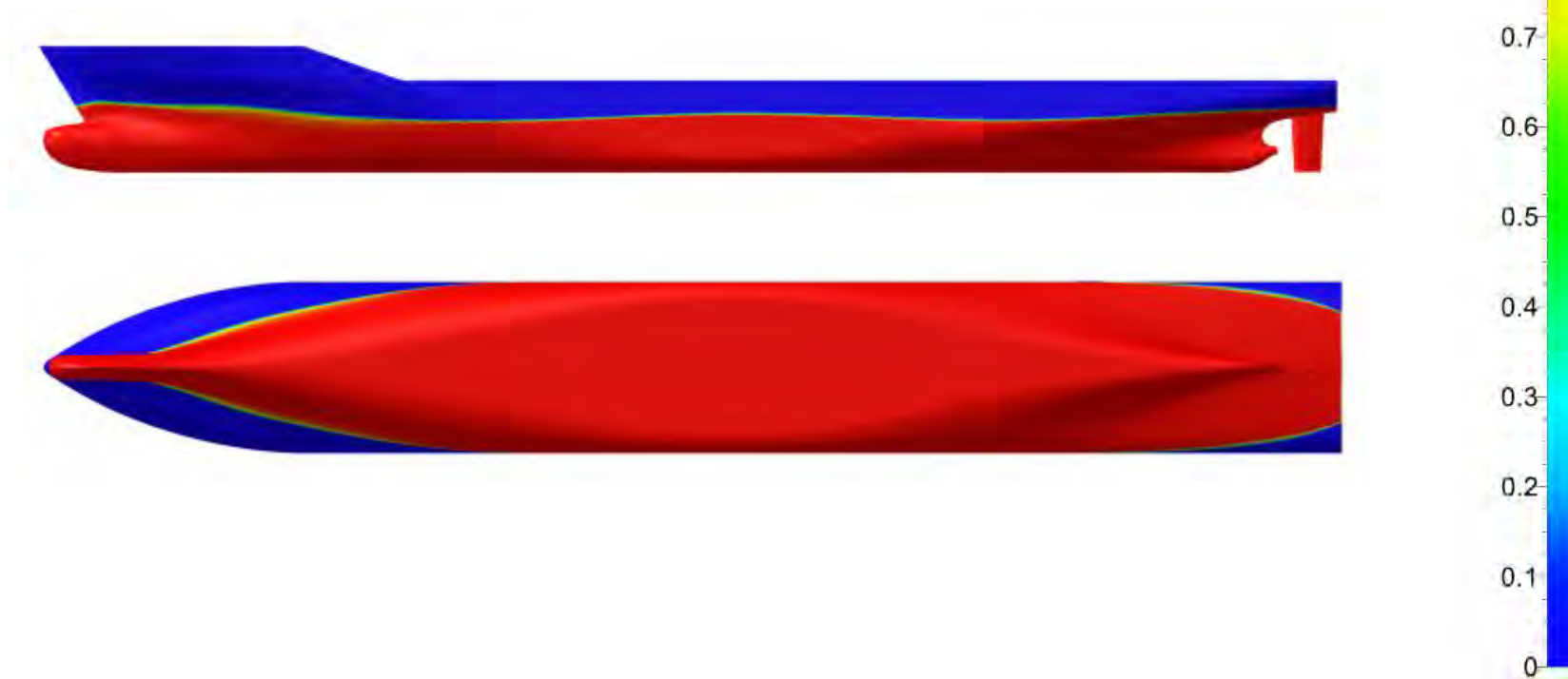
See figure below for a display.

4.4 Results case 3

Basic Data

See below for the basic data extracted from the computations .

Comparison				
Fr	C_{T_efd} X10³	C_{T_cfd} X10³	error %	error
0.1083	3.80	3.73	-1.85	-0.07
0.1516	3.64	3.60	-1.24	-0.05
0.1949	3.48	3.46	-0.55	-0.02
0.2274	3.47	3.49	0.61	0.02
0.2599	3.71	3.70	-0.40	-0.01
0.2816	4.50	4.52	0.39	0.02

Wave Elevation
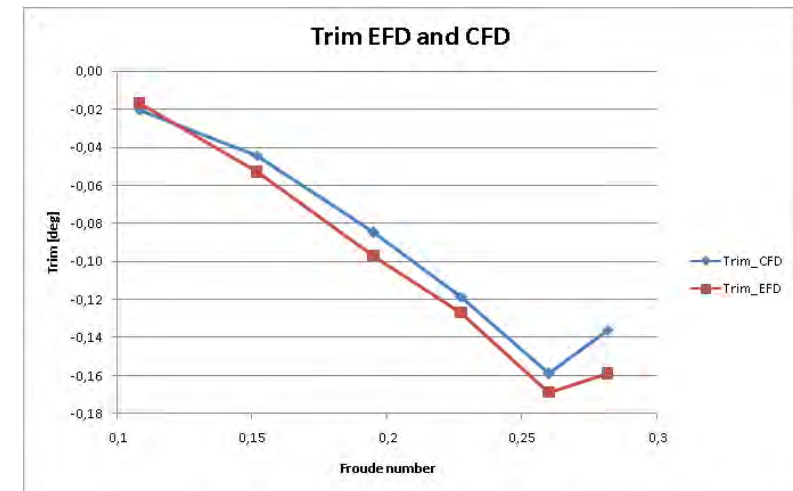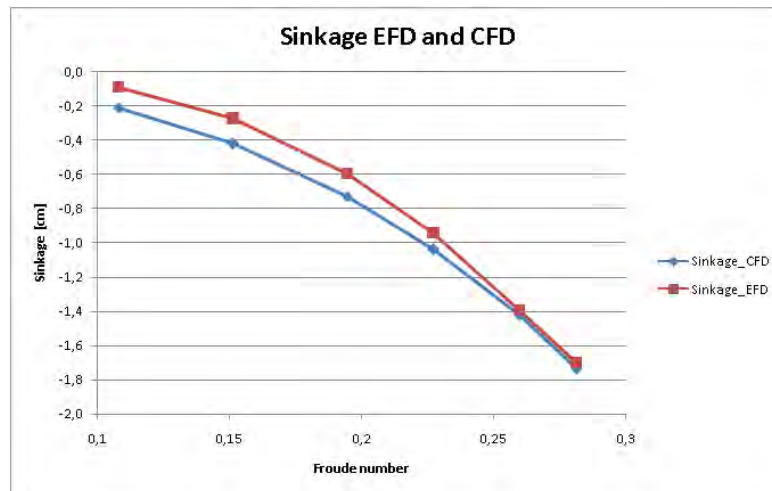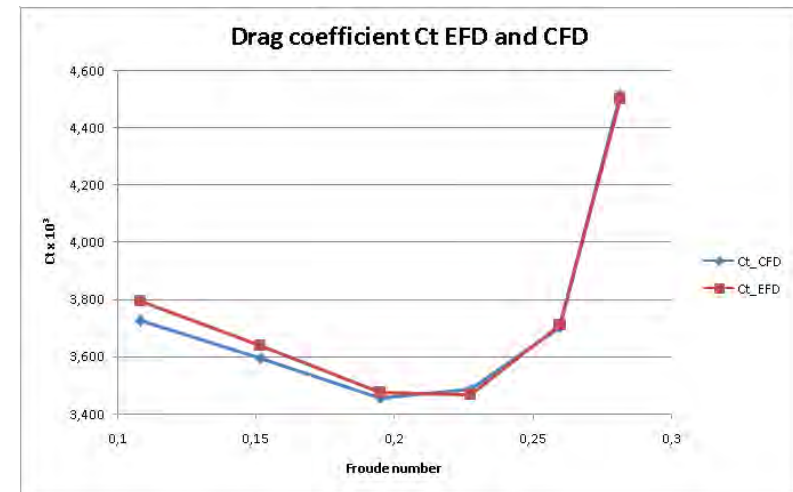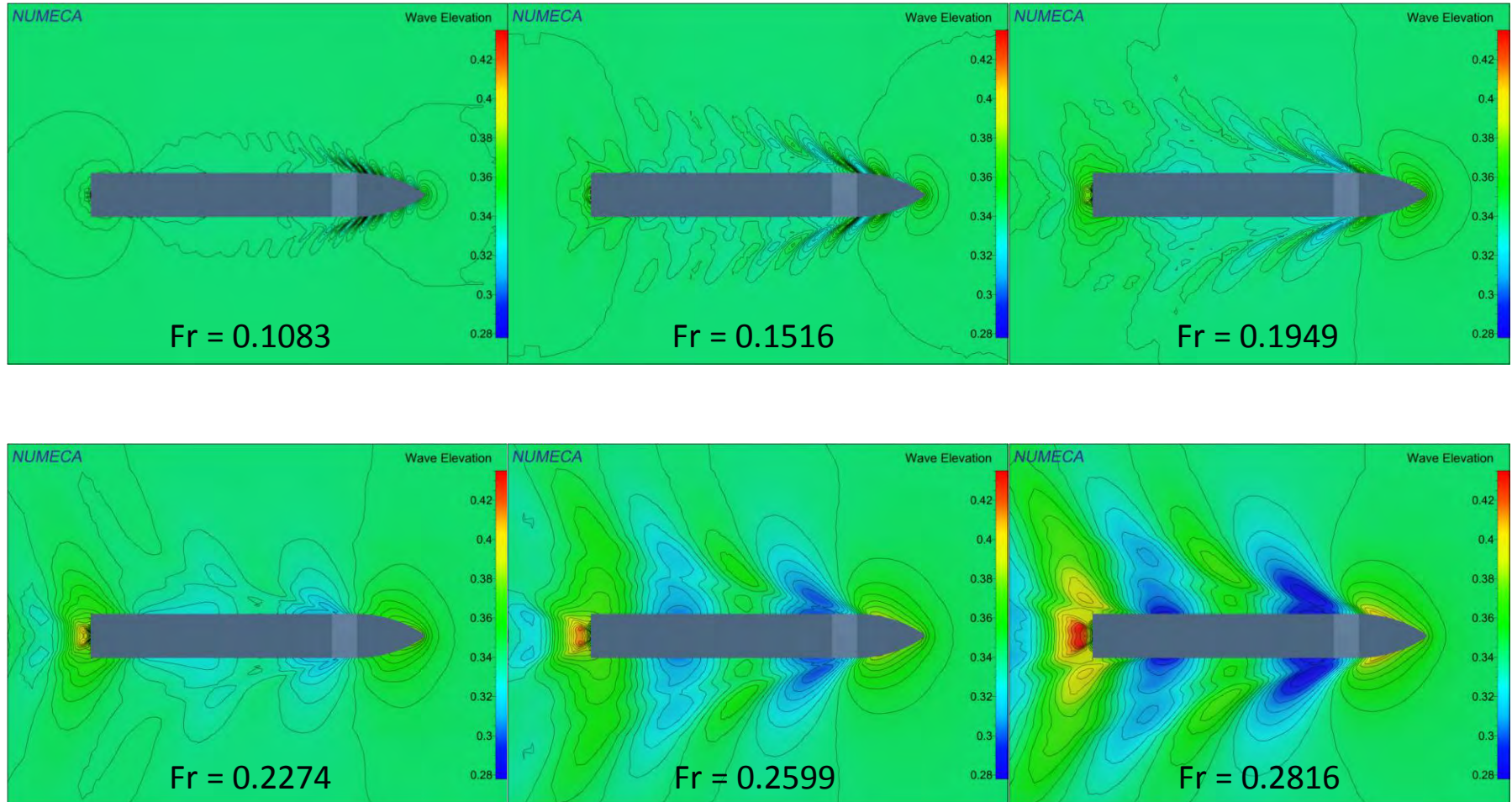
The table below summarize the wetted surface of the hull obtained for different speeds :

Wetted Surface						
Froude	0.1083	0.1516	0.1949	0.2274	0.2599	0.2816
Wetted Surface (m²)	9.78	9.84	9.92	9.98	10.06	10.12

Wetted Surface

Fr = 0.1083

Fr = 0.1516

Fr = 0.1949

Fr = 0.2274

Fr = 0.2599

Fr = 0.2816

Wetted Surface
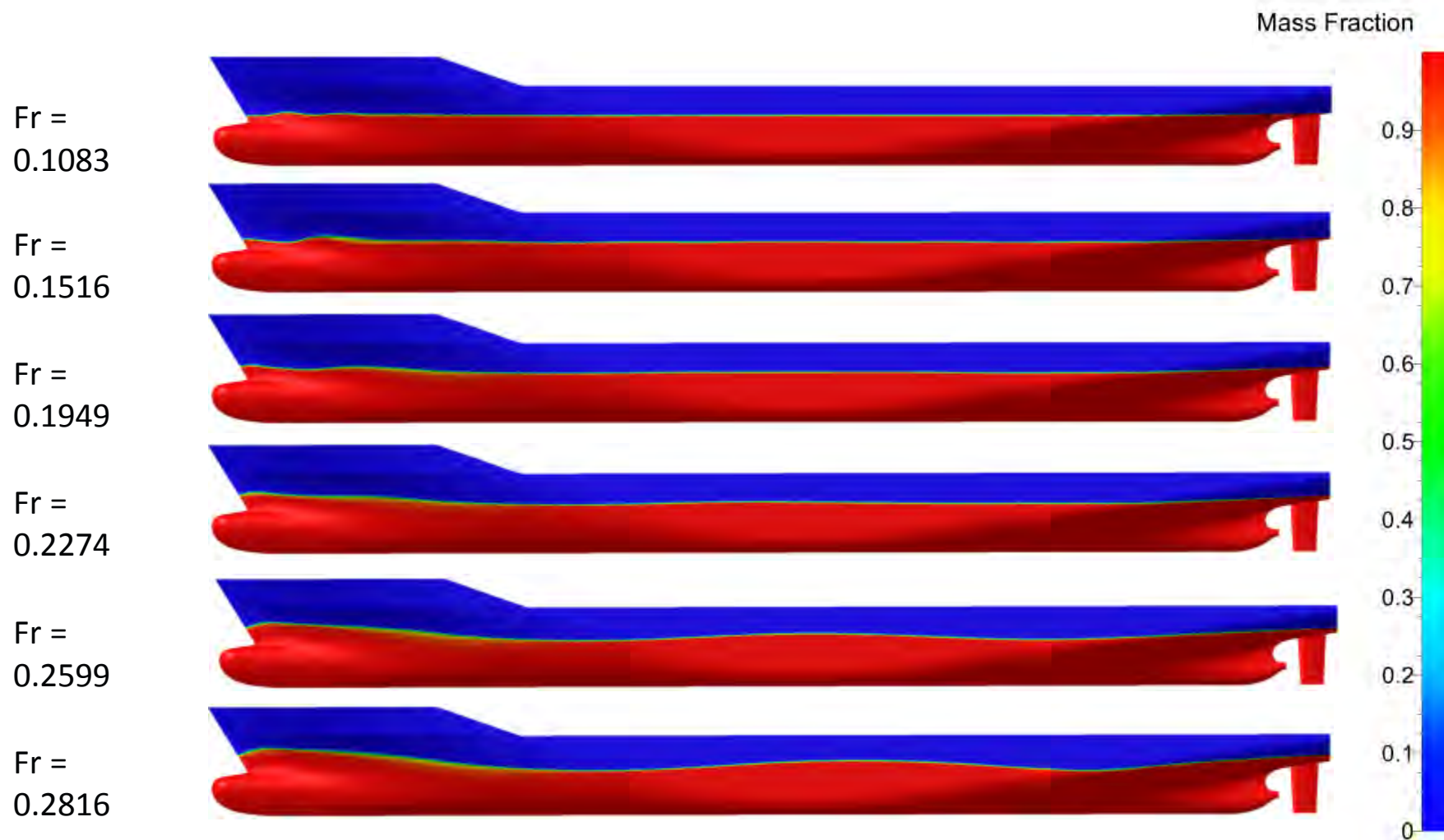
Fr = 0.1083

Fr = 0.1516

Fr = 0.1949

Fr = 0.2274

Fr = 0.2599

Fr = 0.2816

Mass Fraction

Hydrodynamic Pressure

Fr = 0.1083

Fr = 0.1516

Fr = 0.1949

Fr = 0.2274

Fr = 0.2599

Fr = 0.2816

Hydrodynamic Pressure

2000

1000

0

-1000

-2000

-3000

Hydrodynamic Pressure
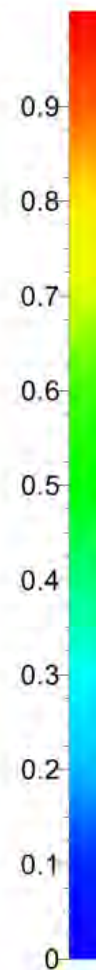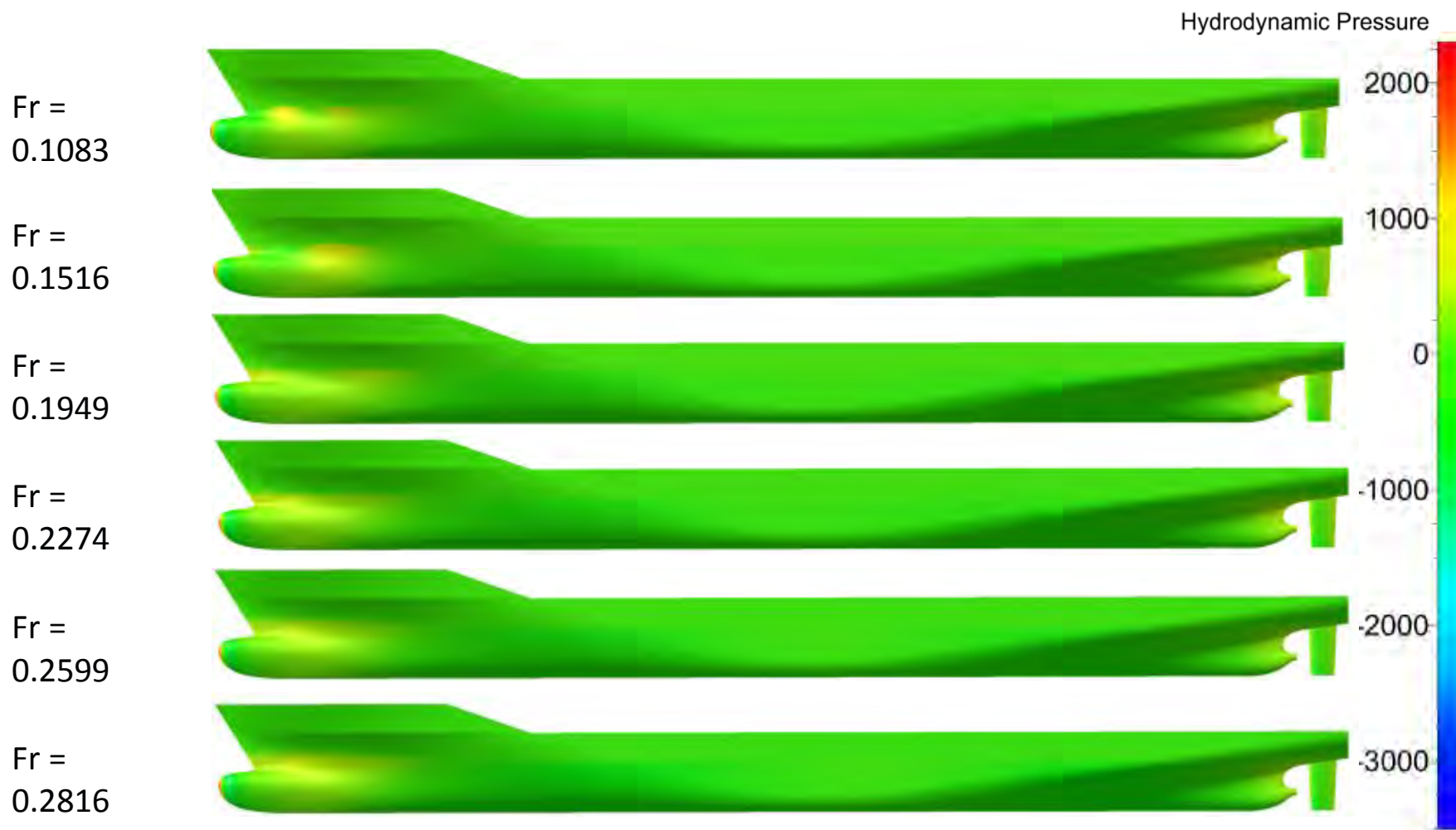

Fr = 0.1083

Fr = 0.1516

Fr = 0.1949

Fr = 0.2274

Fr = 0.2599

Fr = 0.2816

Hydrodynamic Pressure

2000

1000

0

-1000

-2000

-3000

4.5 **Reduction of the computational time**

In this part of the presentation, we want to know how fast FINE™/Marine can be to finish the resistance calculation. To this aim, we have selected the speed of 1.65 m/s and calculations have been launched with different settings. The numerical parameter that we want to study is the number of non-linear iterations. We also want to know if the sub-cycling acceleration method is interesting. The combination of these two parameters leads to 8 calculations summarized below :

- number of non-linear iterations of 2 without sub-cycling
- number of non-linear iterations of 2 with sub-cycling
- number of non-linear iterations of 3 without sub-cycling
- number of non-linear iterations of 3 with sub-cycling
- number of non-linear iterations of 4 without sub-cycling
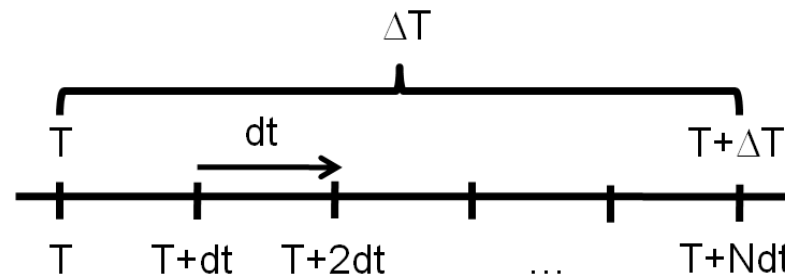- number of non-linear iterations of 4 with sub-cycling
- number of non-linear iterations of 5 without sub-cycling
- number of non-linear iterations of 5 with sub-cycling

Note on the sub-cycling acceleration method:

The discretization of the fraction volume transport equation needs specific compressive schemes to accurately preserve the sharpness of the interface. This leads to small time steps since the CFL constraint comes only from the resolution of the volume fraction.

The idea of the splitting method of the volume fraction equation is to reduce that CFL condition by using a specific time step for the fraction volume which is a multiple of the time step associated with the global simulation. In other words, the global time step is split into a sequence of smaller ones, naturally leading to lower Courant numbers. Consequently, the volume fraction equation is solved multiple times during a single global time step. As the CPU time related to the volume fraction equation is not high compared to other parts of the solver, the global CPU time of the simulation is strongly reduced.

A second way of benefiting from the splitting method is to increase the accuracy of the result at a small additional CPU cost. By adopting the same time step as the fully unsteady approach, the reduction of the Courant number due to the splitting will lead to a more accurate representation of the free surface.

Computational cost

In order to quantify the computational cost, we define a quantity that we want to reduce :

Computational cost = time to reach the convergence * number of processor on which the calculation has run

Nb of non-linear iterations	subcycling	Time per time step [s]	nb of timstep for convergence 1%	nb of timstep for convergence 2%	nb of proc	cpu cost [cpu.h] Convergence 1%	cpu cost [cpu.h] Convergence 2%
2	no	8.52	4661	3605	16	176.5	136.5
2	yes	13.10	860	775	16	50.1	45.1
3	no	12.92	3079	1935	16	176.9	111.1
3	yes	19.24	539	475	16	46.1	40.6
4	no	17.14	2197	1537	16	167.4	117.1
4	yes	26.00	420	307	16	48.5	35.5
5	no	20.56	1935	1405	16	176.8	128.4
5	yes	21.80	352	292	16	34.1	28.3

Computational cost

Finally, the fastest calculation is the one combining 5 non-linear iteration and the sub-cycling. It has a computational cost of about 30 cpu.h, which means approximately that the calculation can converge in 1h on 30 processors.

The computational cost is plotted on the chart below. We can see that the sub-cycling option allows to reduce the computational cost by 4, **passing from 176.8 h.cpu to 34.1 h.cpu**
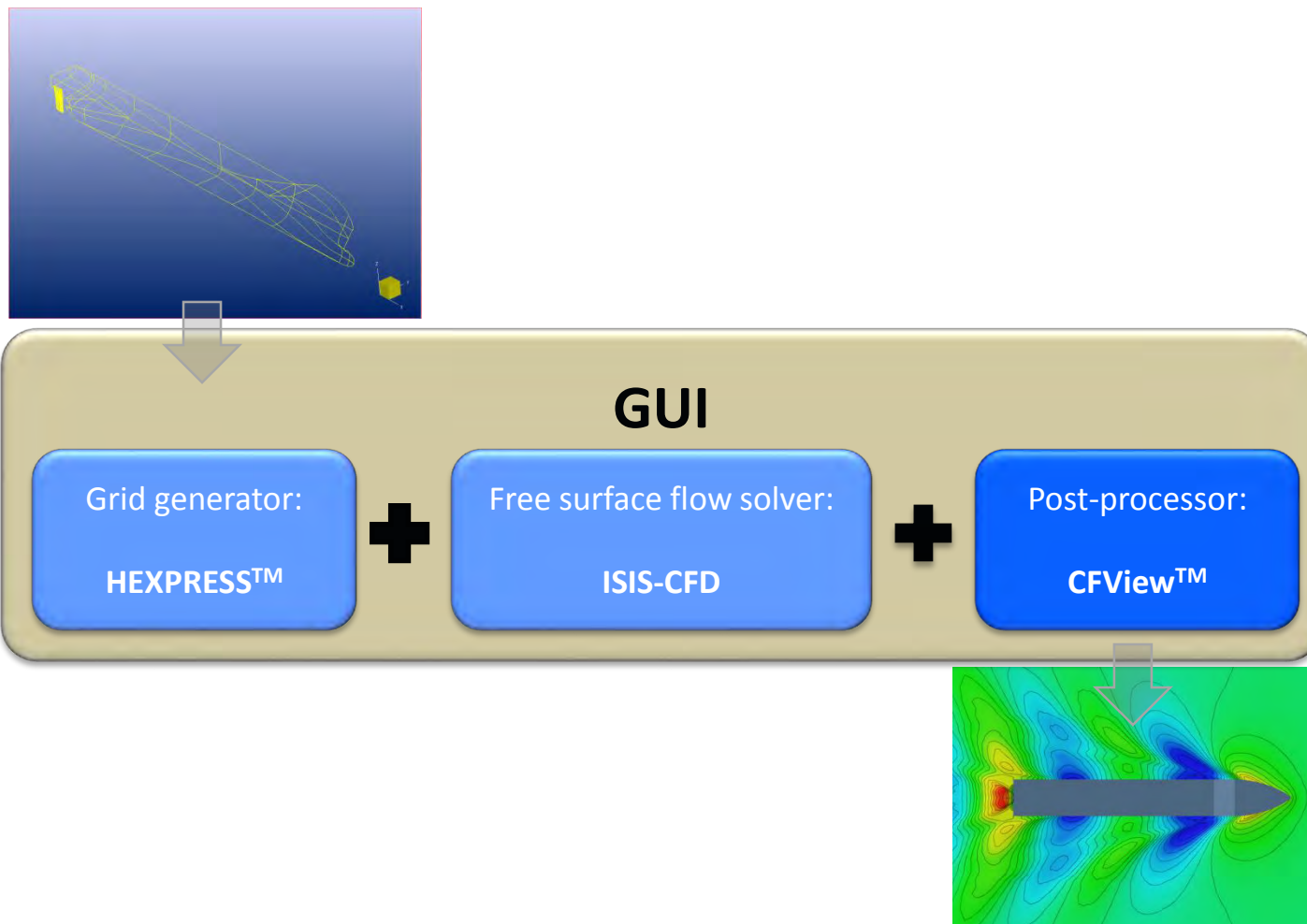
The number of non-linear iteration has only a little impact on the computational cost when it varies between 2 and 5.
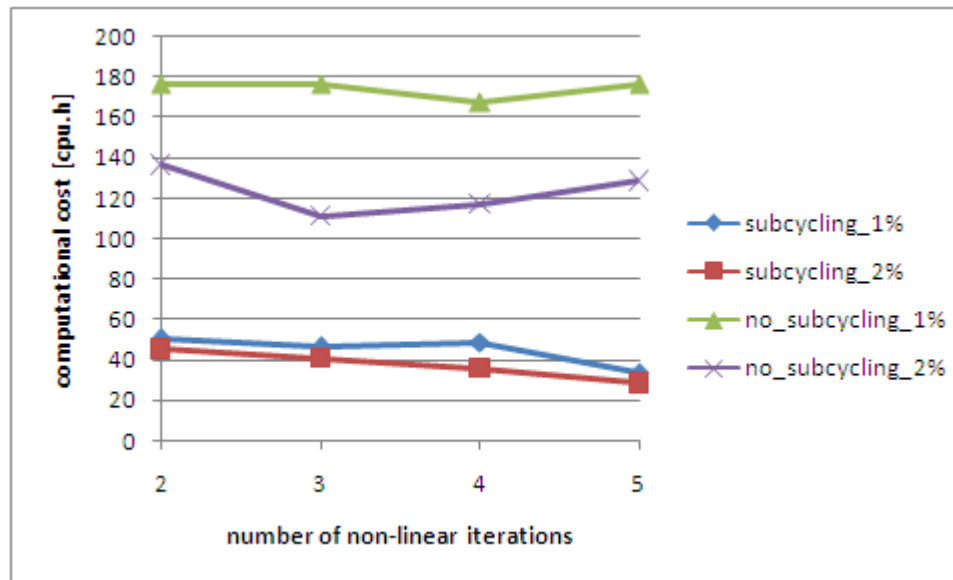
5 Conclusion

This case covers a wide part of the FINE^{TM}/Marine features and demonstrates the ability of the product to handle this kind of project in a short time frame.

GUI

Grid generator:	Free surface flow solver:	Post-processor:
HEXPRESS^{TM}	**ISIS-CFD**	**CFView^{TM}**

Copyright **© NUMECA International**

It is shown in this work that FINE/Marine is:

- Accurate

- Dedicated to Marine applications

- Very fast in meshing: 2.3 million s of cells in 5 minutes

- Very fast computing time: 1 hour of CPU time using 32 cores PC on 2.3 million of cells

- Dedicated post-processing

- Seamless automated design process can be integrated into existing customers' working process

KCS Resistance Calculation

Open Discussion