WP 3 / Activity 3.3

A Computational Fluid Dynamics Analysis of a Ro-Pax Ferry Hull Form

Prepared by:

University of Rijeka - Faculty of Engineering

Tehnomont Shipyard Pula Ltd

Summary

1. INTRODUCTION

The practical application of the Computational Fluid Dynamics (CFD), for predicting the flow pattern around ship hull and for calculating ship resistance has made much progress over the last decades and nowadays the CFD tools play an important role in the ship hull form design.

This document presents the CFD analysis of the hull form of the Ro-Pax ferry. The numerical 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), [1].

Three series of simulations were made: one for a scale model of the ferry and the other two for a full size ferry for two different draughts (design and summer). The simulations were done for the hulls in upright conditions, for a series of different speeds.

The hull models were exported in Parasolid format before importing it in HEXPRESS^{TM}, [2]. The mesh required for the numerical calculation 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}, [3], NUMECA's Flow Visualization System integrated in FINE^{TM}/Marine, [1]. The workflow of the procedure is presented on Figure 1.

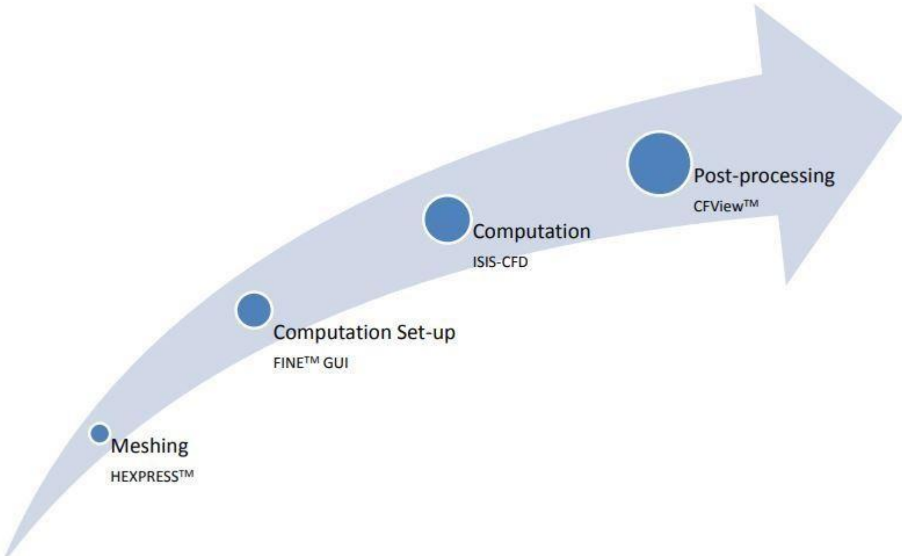

Figure 1. NUMECA's Workflow

The hull models were exported in Parasolid format before importing it in HEXPRESS, [2]. The mesh required for the numerical calculation was then generated with HEXPRESS™, NUMECA's full hexahedral unstructured grid generator integrated in FINE™/Marine.

Results were processed and analyzed with CFView™, [3], NUMECA's Flow Visualization System integrated in FINE™/Marine, [1].

Through this report following outputs for all three cases are presented: the ferry resistance, relative velocity streamlines, wetted surface, hydrodynamic pressure on the hull and wave elevation contours.

At the end of the report, a power-speed diagram for one selected propeller design point is shown.

2. RO-PAX FERRY

The main particulars of the Ro-Pax ferry hull are shown in the Table 1.

Table 1. Ro-Pax ferry hull main particulars

L_{OA}	129.0 m
L_{PP}	123.0 m
B	23.6 m
T_{design}	5.25 m
$T_{scantling}$	5.6 m
$D_{to\ freeboard\ deck}$	8.0 m

Figure 2. presents the Ro-Pax ferry hull lines drawing that served as the basis for the CFD analysis.

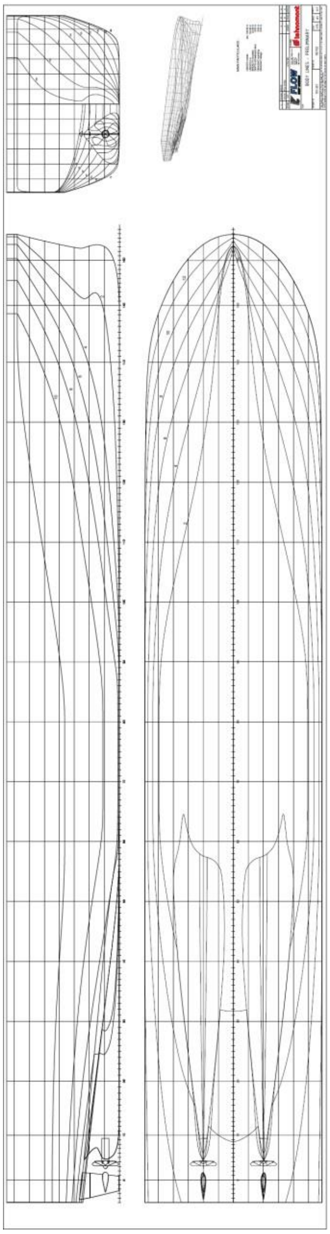

Figure 2. Ro-Pax ferry hull lines

The estimated Ro-Pax ferry speed is 15.5 knots. Table 2. shows the range of speeds and Froude numbers for which CFD analysis was performed.

Table 2. Speeds and Froude numbers used in CFD analysis

V_{KN}, kn	2.0	4.0	6.0	8.0	10.0	12.0	14.0	15.5	16	18
V, m/s	1.029	2.058	3.087	4.116	5.144	6.173	7.202	7.974	8.231	9.260
Fr	0.033	0.066	0.099	0.132	0.164	0.197	0.230	0.224	0.231	0.260

The speeds range from 2.0 knots (1.029 m/s) to 18 knots (9.260 m/s), and these speeds correspond to Froude numbers ranging from 0.033 to 0.260.

3. PREPARATION OF A 3D MODEL FOR CFD SIMULATIONS

The basic model of the Ro-Pax ferry was prepared by the company Flow Ship Design d.o.o., who was a subcontractor of the project partner Tehnomont Shipyard Pula d.o.o. The model was initially prepared in IGES format. Before this 3D model can be used and imported into the software for CFD simulations, it was necessary to make a comprehensive check of the model, which included, among other things, the check for any irregularities in the model. These actions allow all irregularities to be identified and corrected in advance, in order to subsequently get as smooth as possible mesh required for the numerical simulations. Mesh generation is very sensitive to geometric irregularities, so identifying all critical spots is a necessity. Once all these critical spots are identified and appropriate corrections made, a model without irregularities can be obtained.

After that the model needs to be closed since the CFD software can recognize only solid bodies and therefore the model has to be defined in that way. In order for the resulting model to be accepted as a solid, it must not have any discontinuities or gaps, and has to be completely closed on all surfaces. This is a numerical model, but if it were a physical model, it could be said that the model must be watertight. The model was well defined and already closed with the main deck, and therefore after the checking of the geometry, what remained was to define the model as solid.

Figure 3. shows the initial 3D model, and Figure 4. shows the 3D model defined as a solid ready to be imported into the CFD software.

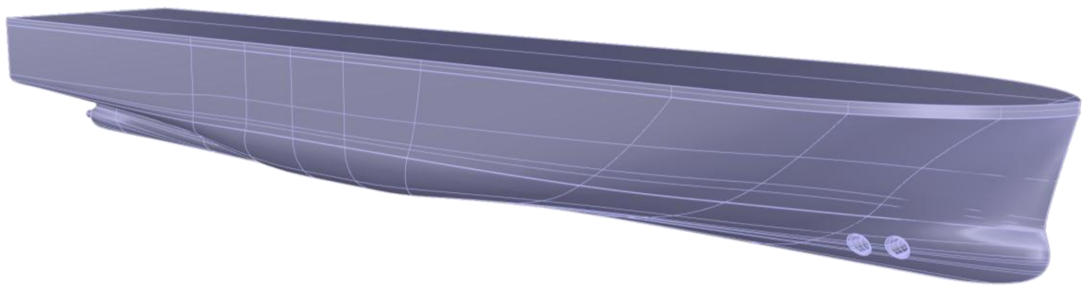

Figure 3. Initial 3D model of the Ro-Pax ferry

After the model was checked in detail and no gaps between the surfaces were found, last step is to join all of the surfaces to get the model as a single solid, Figure 4.

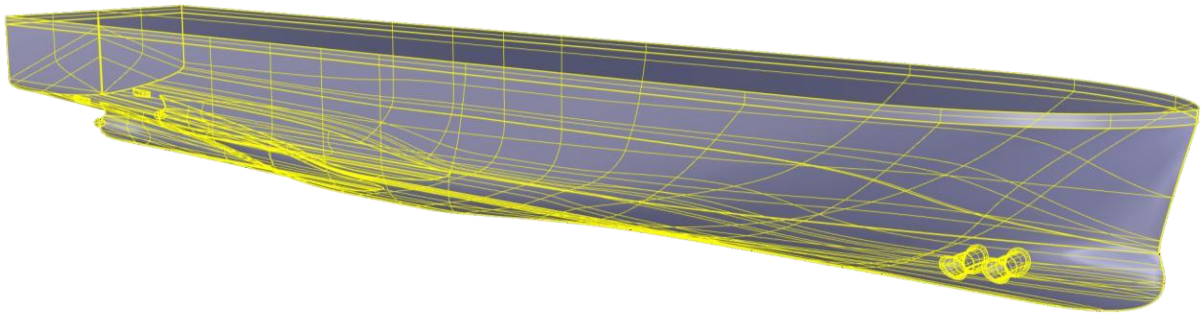

Figure 4. 3D model of the Ro-Pax ferry as a solid, ready to be imported in the CFD software

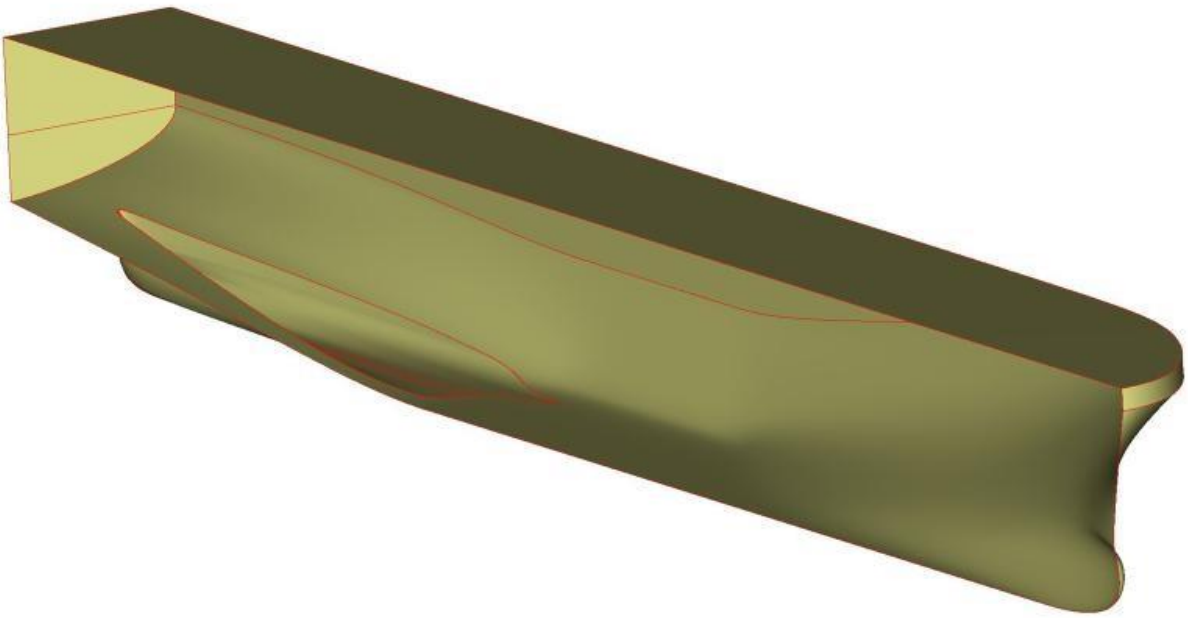

Figure 5. Model (half) imported in the CFD software ready to be meshed

The final model is exported as a “Parasolid” and is ready for importing into the CFD software for the mesh generation, Figure 5. Since the model (i.e. the hull) is symmetric, only one side of the model is meshed.

4. SCALE MODEL OF THE FERRY

In order to determine the hydrodynamic characteristics of the ship hull, physical tests with ship models are usually performed in the institutions with towing tanks. Given such a common practice, a similar procedure is in general taken also in numerical simulations and in that sense the analysis carried out here makes no exception. For the ferry model, the scale λ = 15 was chosen and the obtained main particulars of the model are presented in Table 3.

Table 3. Ro-Pax ferry model hull main particulars

L_{OAm}	8.6 m
L_{PPm}	8.2 m
B_m	1.573 m
$T_{m,\text{design}}$	0.35 m
$T_{\text{m,scantling}}$	0.373 m
$D_{\text{m,to freeboard deck}}$	0.533 m

Table 4. shows the range of model speeds and Froude numbers for which the CFD analysis was carried out.

Table 4. Model speeds and Froude numbers used in CFD analysis

V_{KN}, kn	0.516	1.033	1.549	2.066	2.582	3.098	3.615	4.002	4.131	4.648
Fr_m	0.029	0.058	0.087	0.116	0.145	0.173	0.202	0.224	0.231	0.260

For the model, the speeds range from 0.516 kn to 4.648 kn, and these values correspond to a variation of the Froude number from 0.029 to 0.260. Due to the dynamic similarity, the values of the Froude numbers for the ferry and its model are equal.

The CFD computations were done for the draught of 0.35 m which corresponds to a ferry design draught.

4.1. Mesh generation

In order to form an appropriate mesh, the “Parasolid“ model is loaded into HEXPRESS^{TM}. It is then necessary to define the computational domain around the model, which is constructed by defining a box around the model. The following size of the domain was defined with the following dimensions:

- In front: one model length,
- Behind: three model length,
- Up: one half of model length,
- Down: one and half of model length,
- Laterally: one and half of model length on each side.

The created domain box is presented in Figure 6.

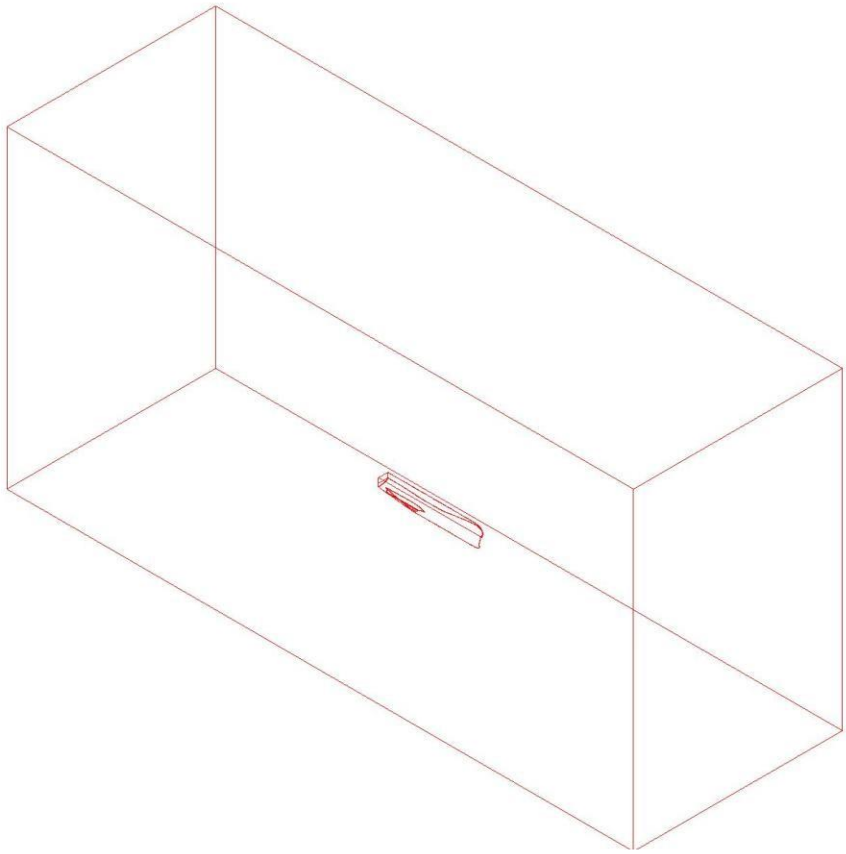

Figure 6. Domain box

The mesh generation in HEXPRESS™ is done in five-steps, Figure 7:

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 4: Optimization – after snapping, the mesh can contain some negative, concave, or twisted cells. This step fixes those cells and increases the quality of the mesh.

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

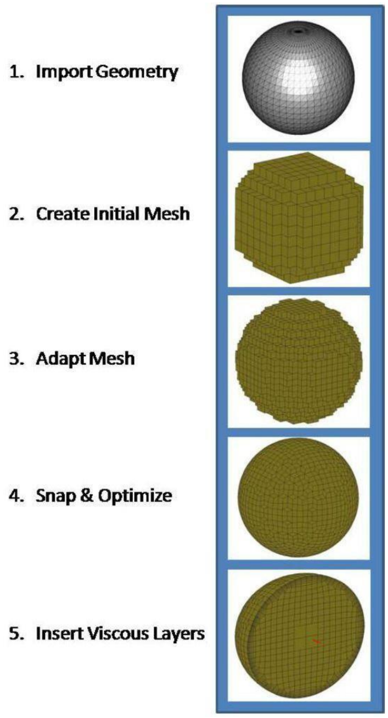

Figure 7. The process of creating a network in HEXPRESS™

Figures from 8 to 12 show some examples of the fine mesh for the scale model of the Ro-Pax ferry. The number of used cells was 1,026,907 with 1,092,048 vertices.

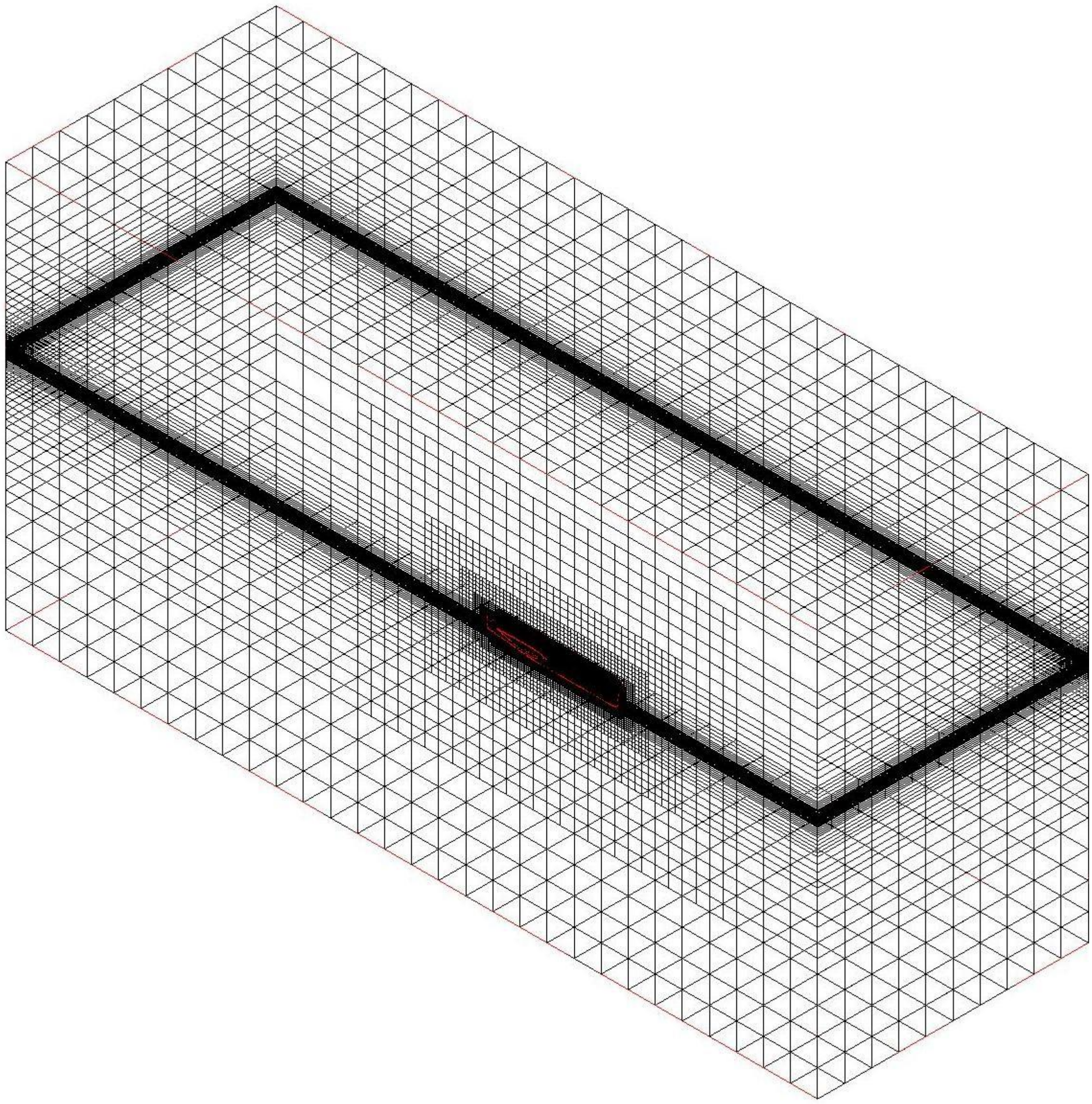

Figure 8. Fine mesh for the scale model

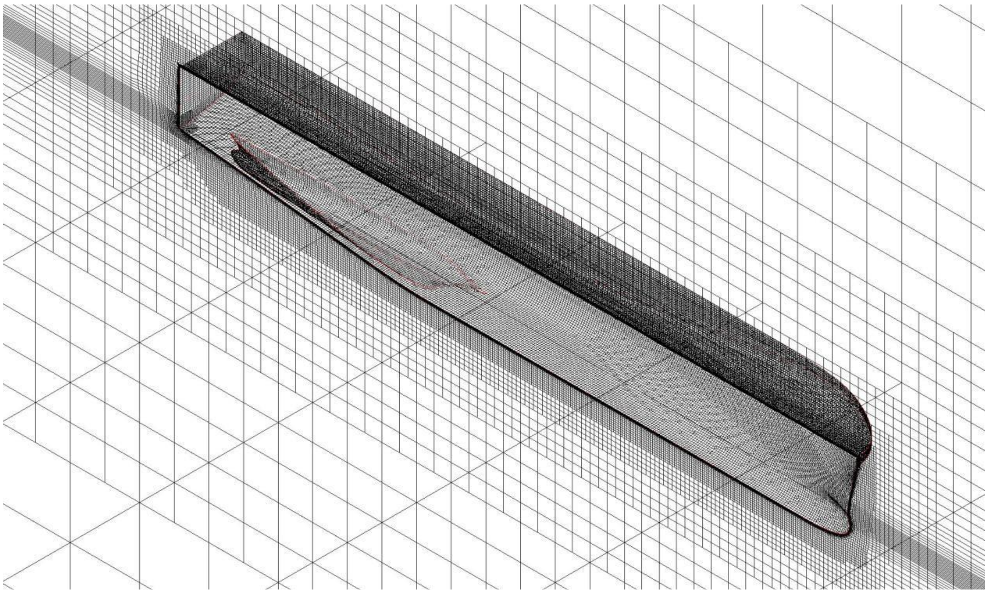

Figure 9. Fine mesh for the scale model, zoomed view

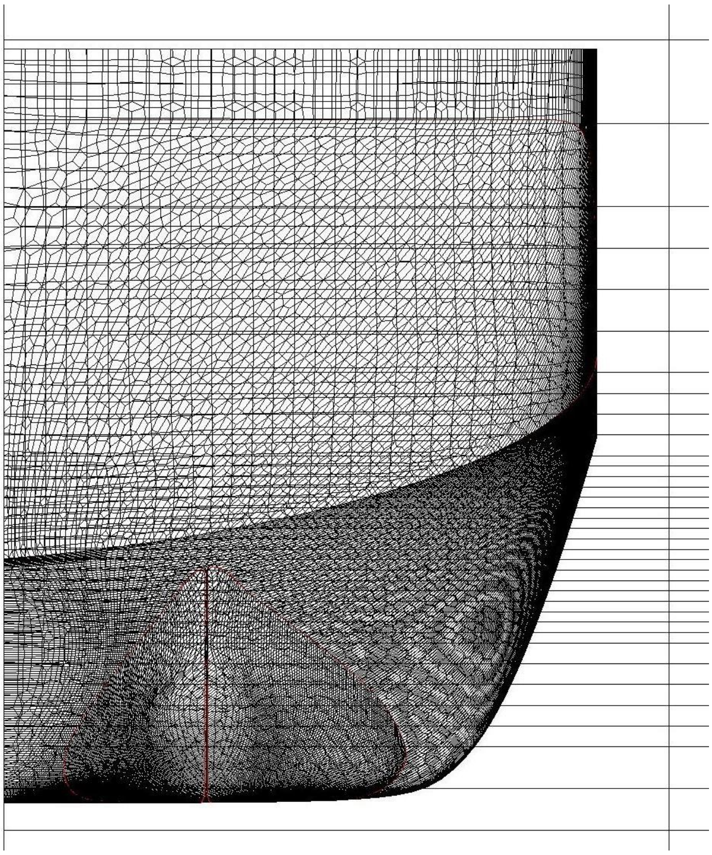

Figure 10. Fine mesh for the scale model, *X*-plane view

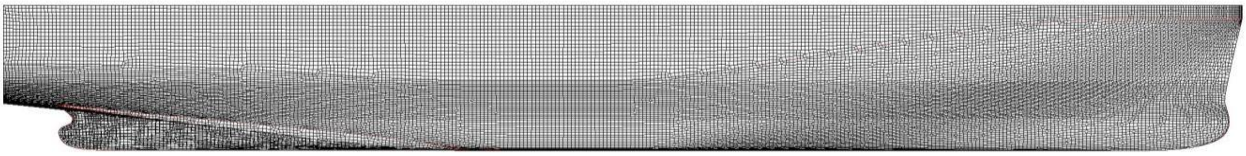

Figure 11. Fine mesh for the scale model, *Y*-plane view

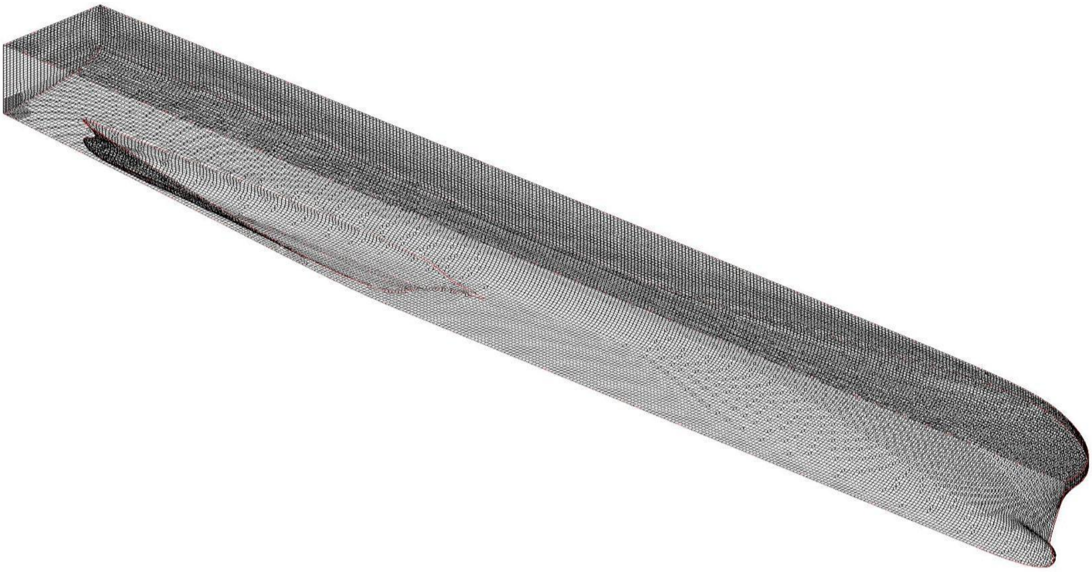

Figure 12. Fine mesh for the scale model, isometric view

4.2. Computations

The boundary conditions shown in Figure 13. are set for the calculations.

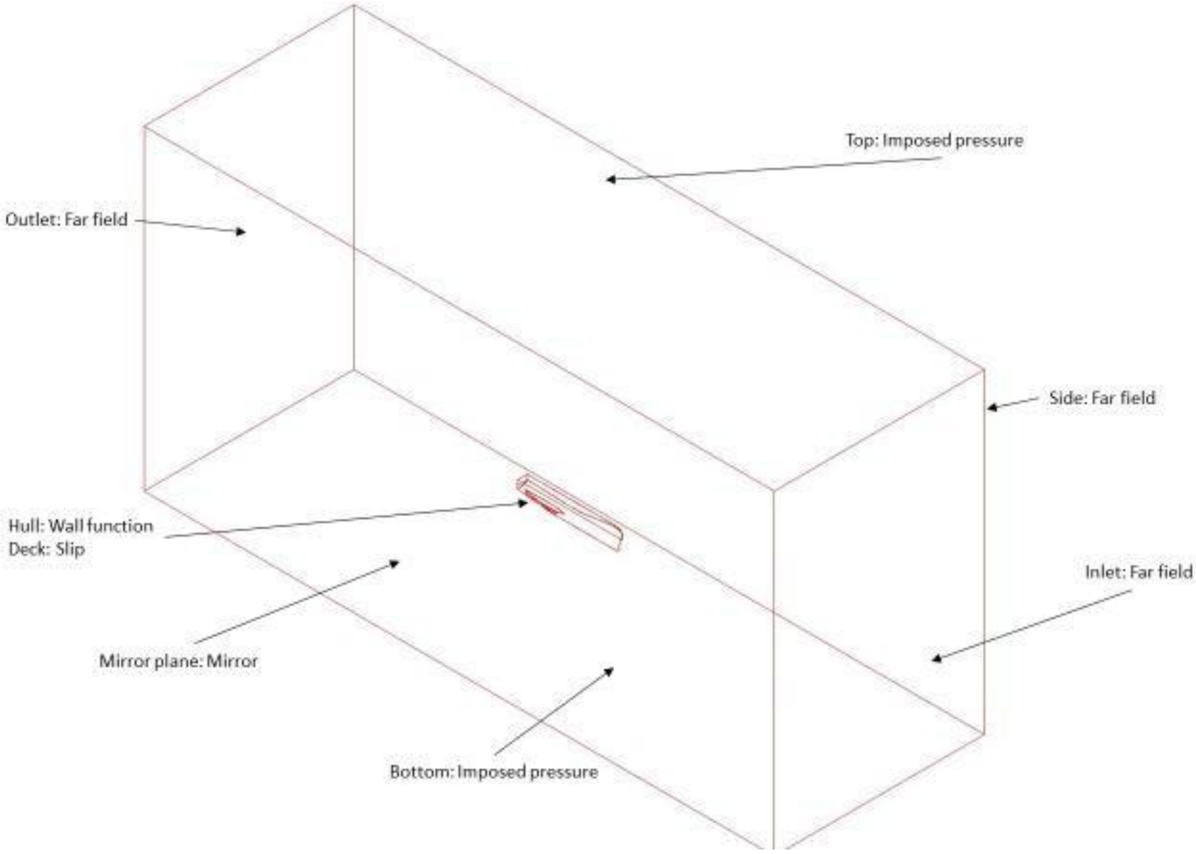

Figure 13. Boundary conditions

The main settings used in the software were:

- $k-\omega$ SST turbulence model with wall functions,
- multi-fluid computations.

The fluid characteristics were set as:

- water: density 999.1026 kg/m^3, dynamic viscosity 0.001138 x 10^{-3} N s/m^2,
- air: density 1.2 kg/m^3, dynamic viscosity 1.85 x 10^{-5} N s/m^2.

4.3. Results

In this section the results of extensive CFD analysis were presented. Table 5. shows the numerically obtained resistance for the corresponding Froude numbers at design draught of 0.35 m. The numerical values are graphically shown in the Figure 14.

Table 5. Resistance vs. Froude number

V_m, m/s	0.266	0.531	0.797	1.063	1.328	1.594	1.860	2.059	2.125	2.391
V_{KNm}, knot	0.516	1.033	1.549	2.066	2.582	3.098	3.615	4.002	4.131	4.648
Fr_m	0.029	0.058	0.087	0.116	0.145	0.173	0.202	0.224	0.231	0.260
Fx_m, N	2.270	7.980	16.190	28.230	45.490	68.880	98.120	124.900	135.220	183.330

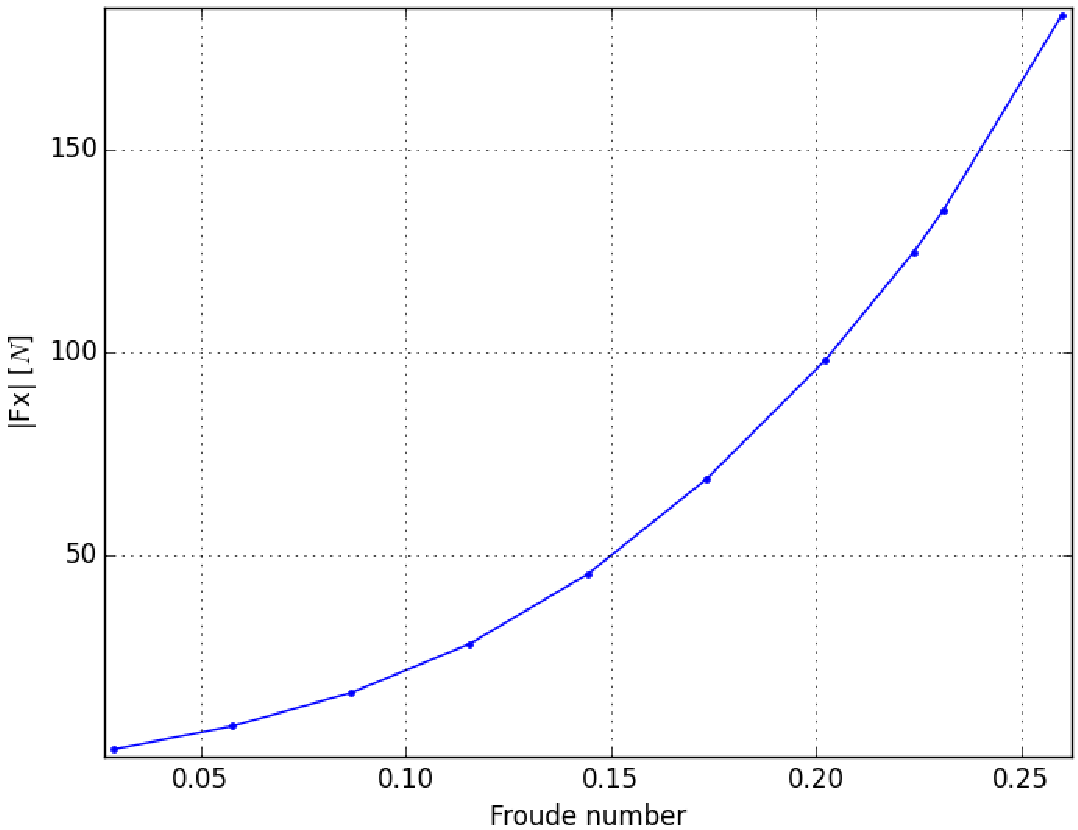

Figure 14. Resistance vs. Froude number

Figures 15. - 24. visually show some results obtained by CFD simulations. First, the results for the speed of 4.002 kn that corresponds to a ferry design speed and then the results for other speeds between 0.516 kn and 4.648 kn were shown.

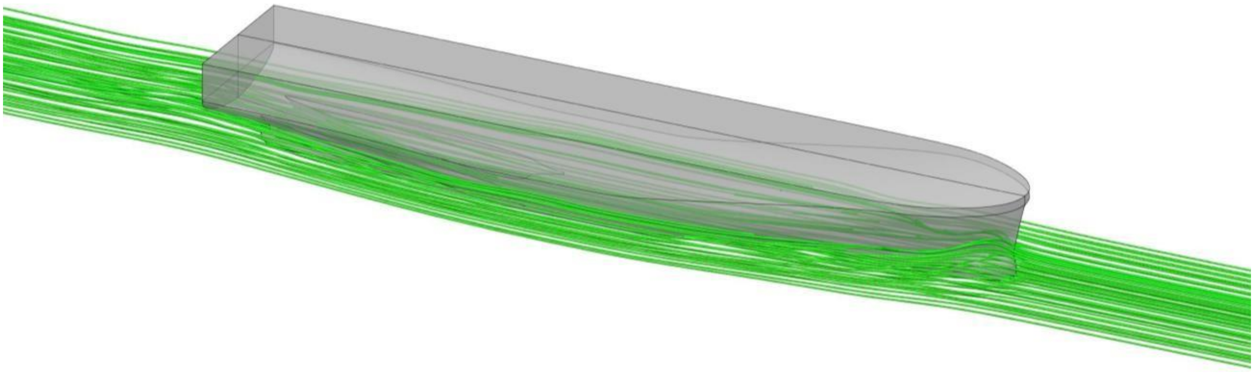

Figure 15. Relative velocity streamlines, V_m= 4.002 kn

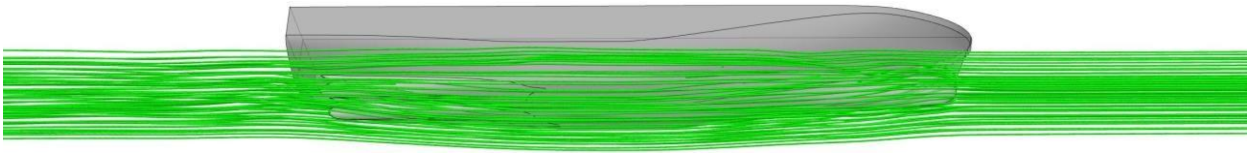

Figure 16. Relative velocity streamlines, V_m= 4.002 kn

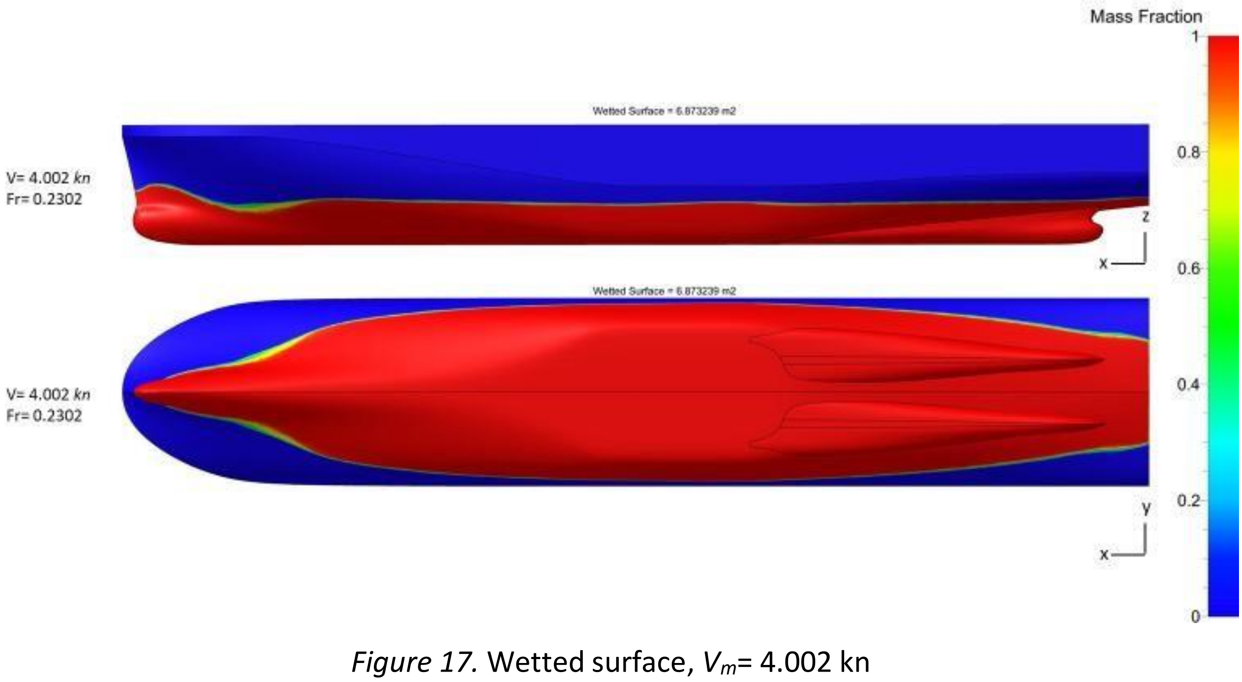

Figure 17. Wetted surface, V_m= 4.002 kn

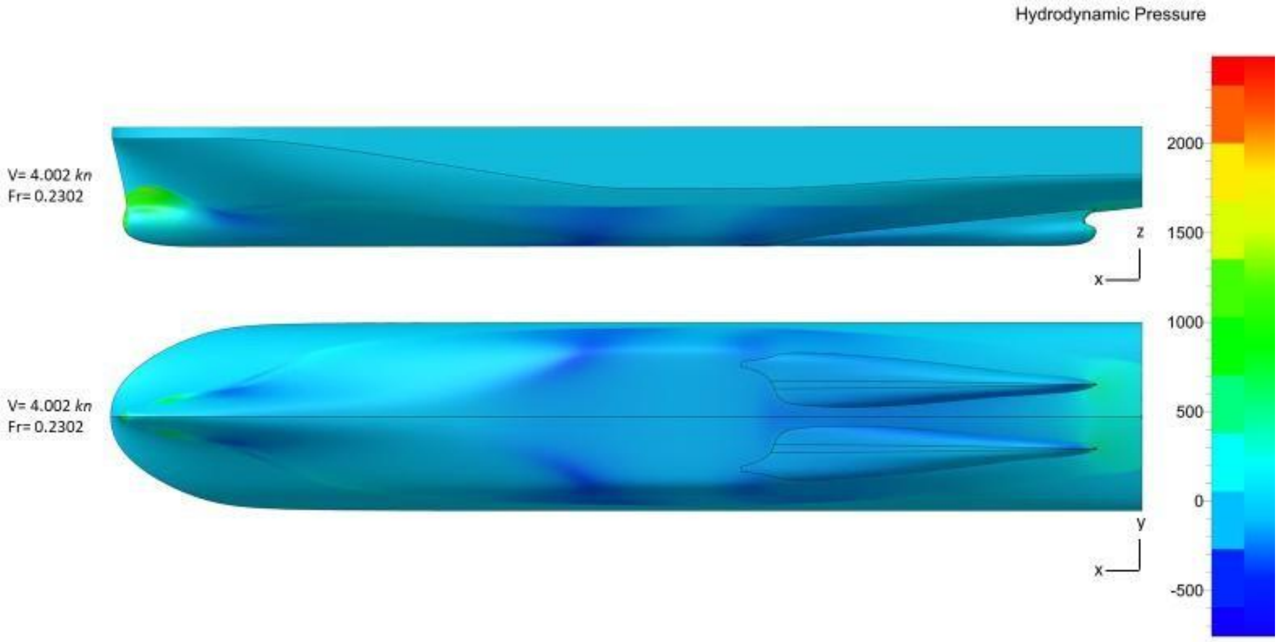

Figure 18. Hydrodynamic pressure, V_m= 4.002 kn

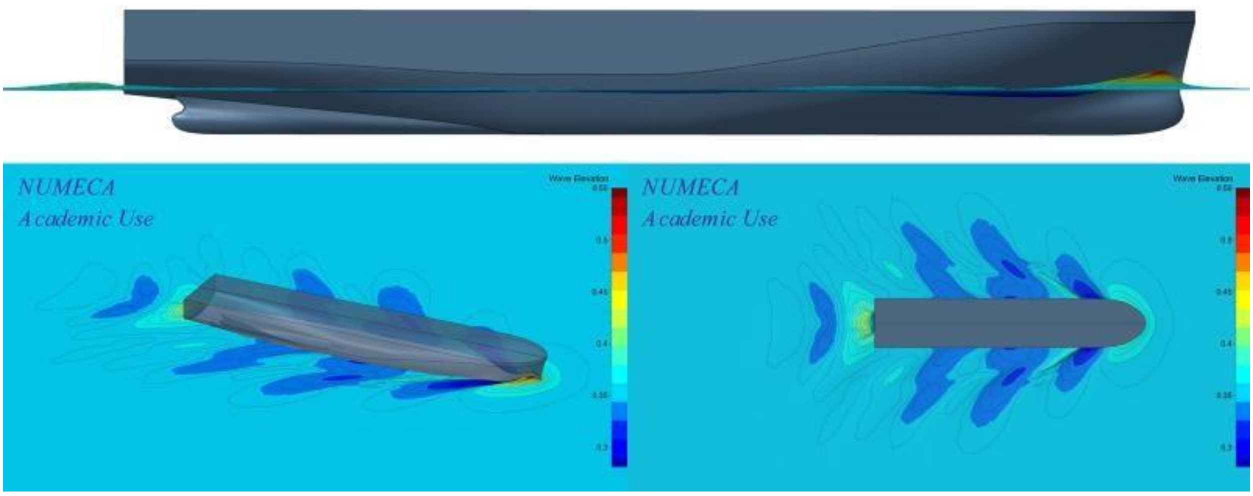

Figure 19. Wave elevations, V_m= 4.002 kn

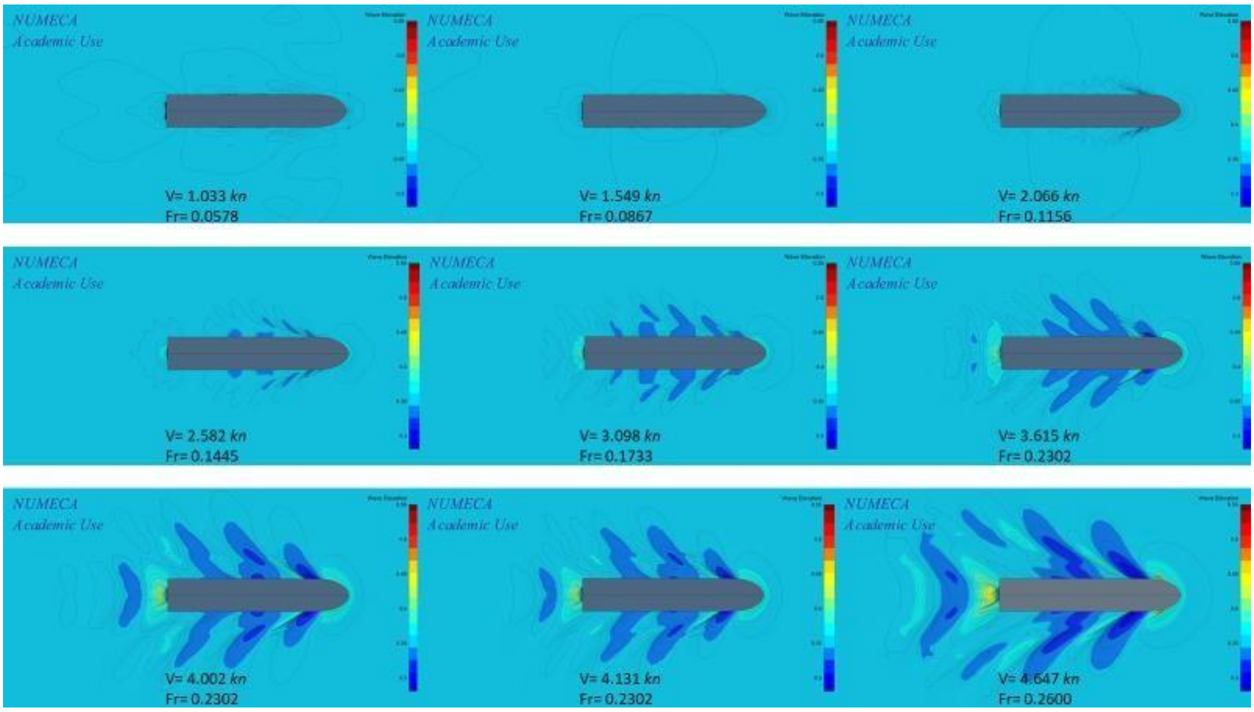

Figure 20. Wave elevations, V_m= 1.033 kn to 4.648 kn, *Y*-plane view

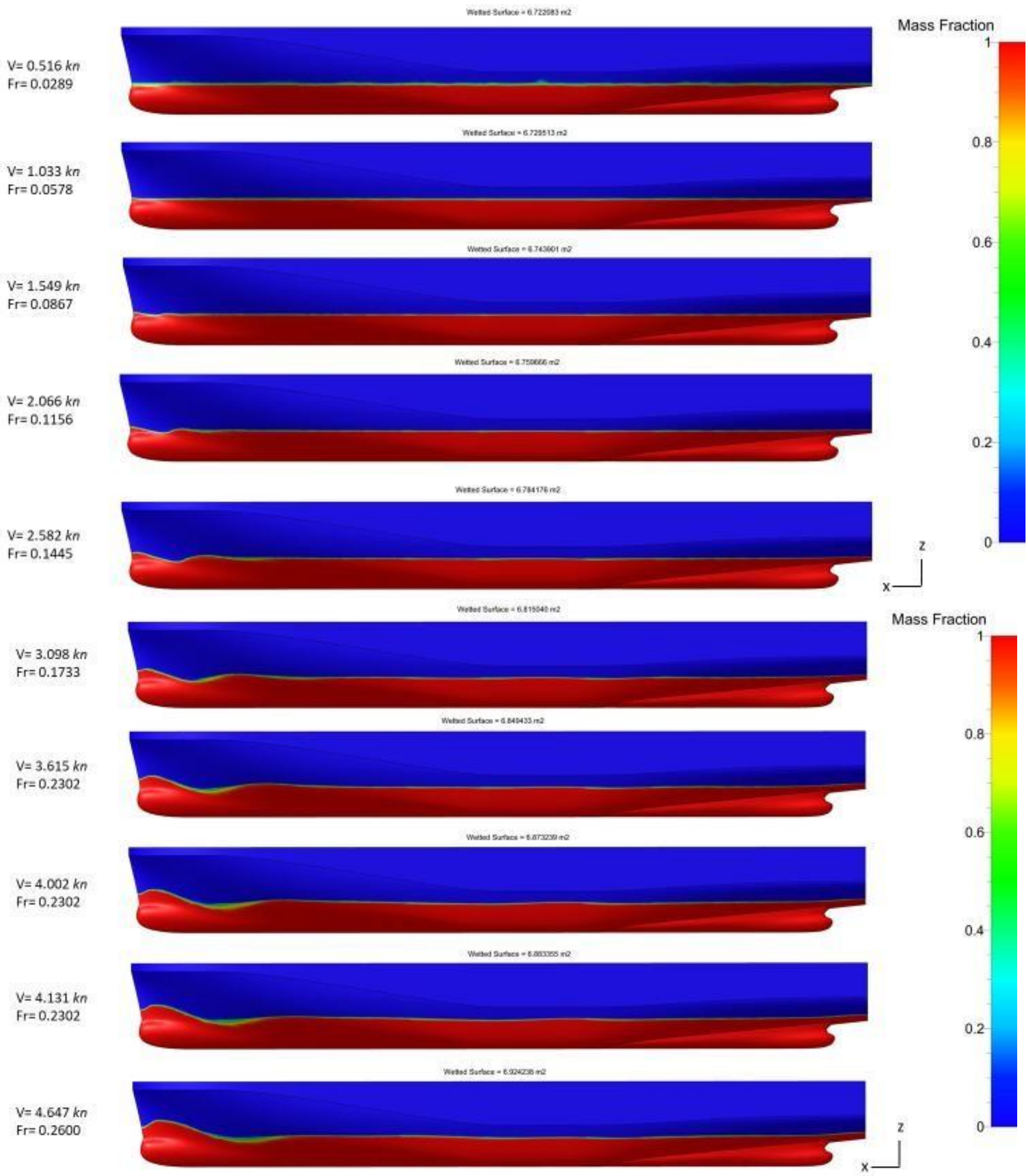

Figure 21. Wetted surface, V_m= 0.516 kn to 4.648 kn, *Y*-plane view

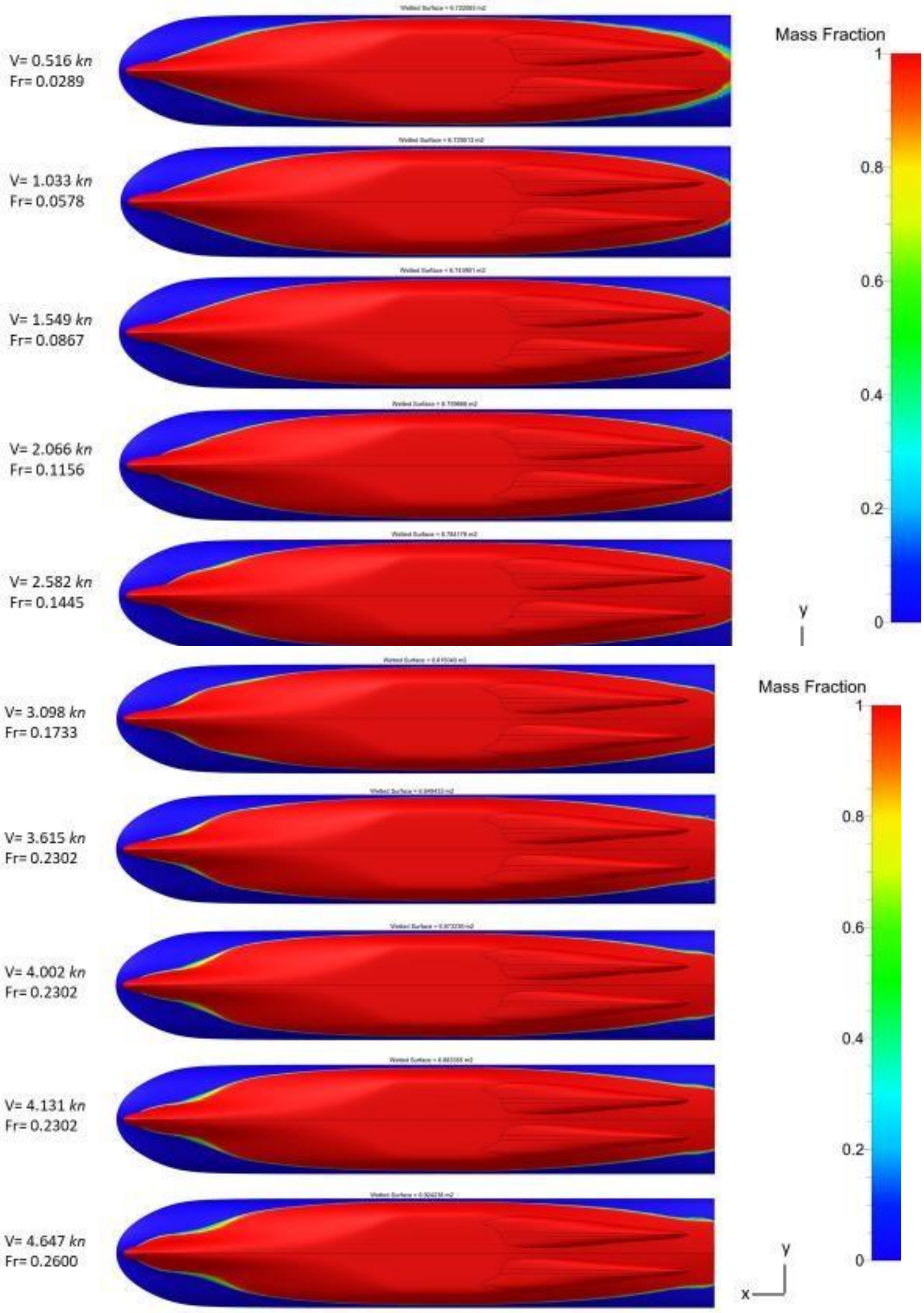

Figure 22. Wetted surface, V_m= 0.516 kn to 4.648 kn, *Z*-plane bottom view

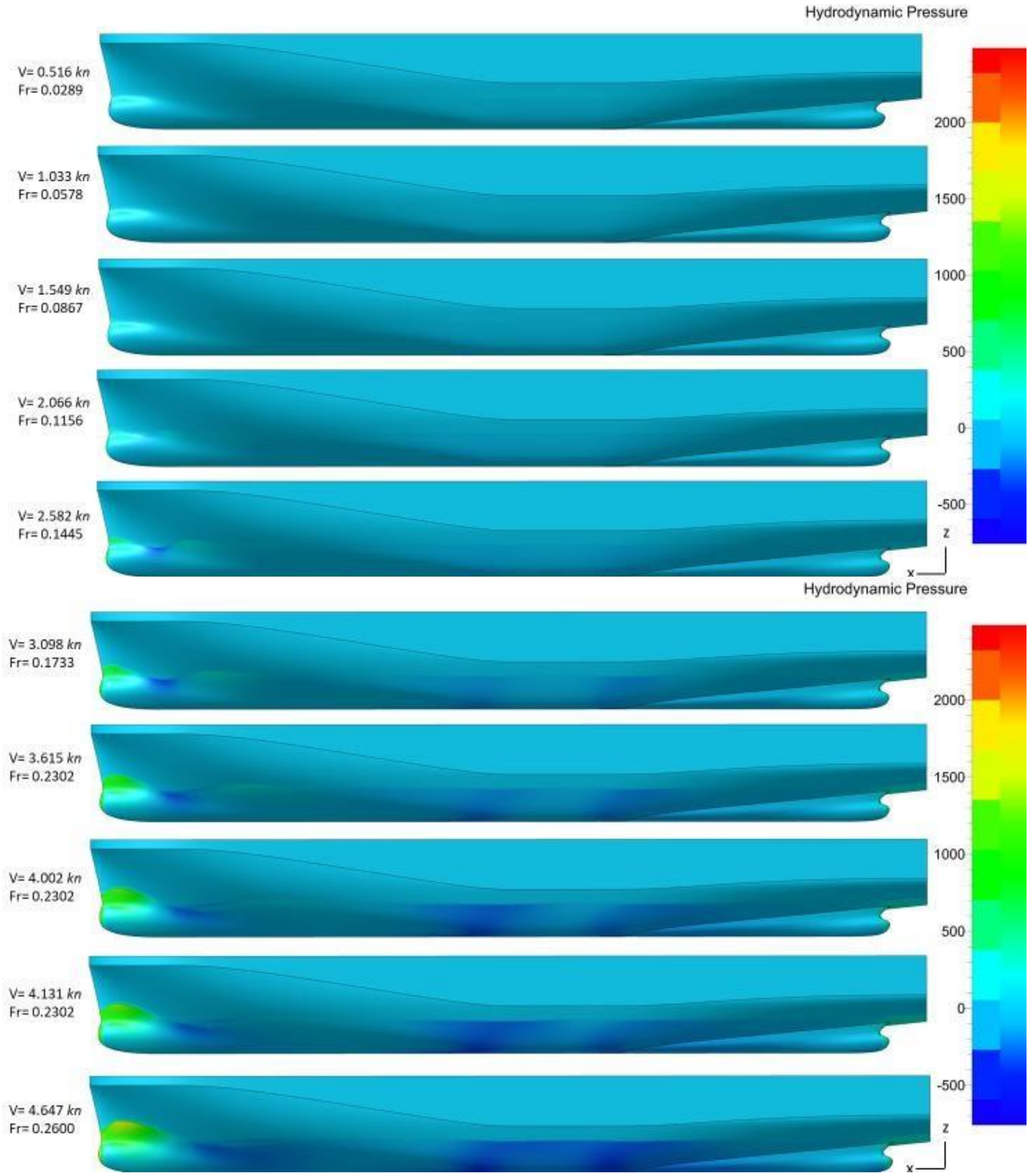

Figure 23. Hydrodynamic pressure, V_m= 0.516 kn to 4.648 kn, *Y*-plane view

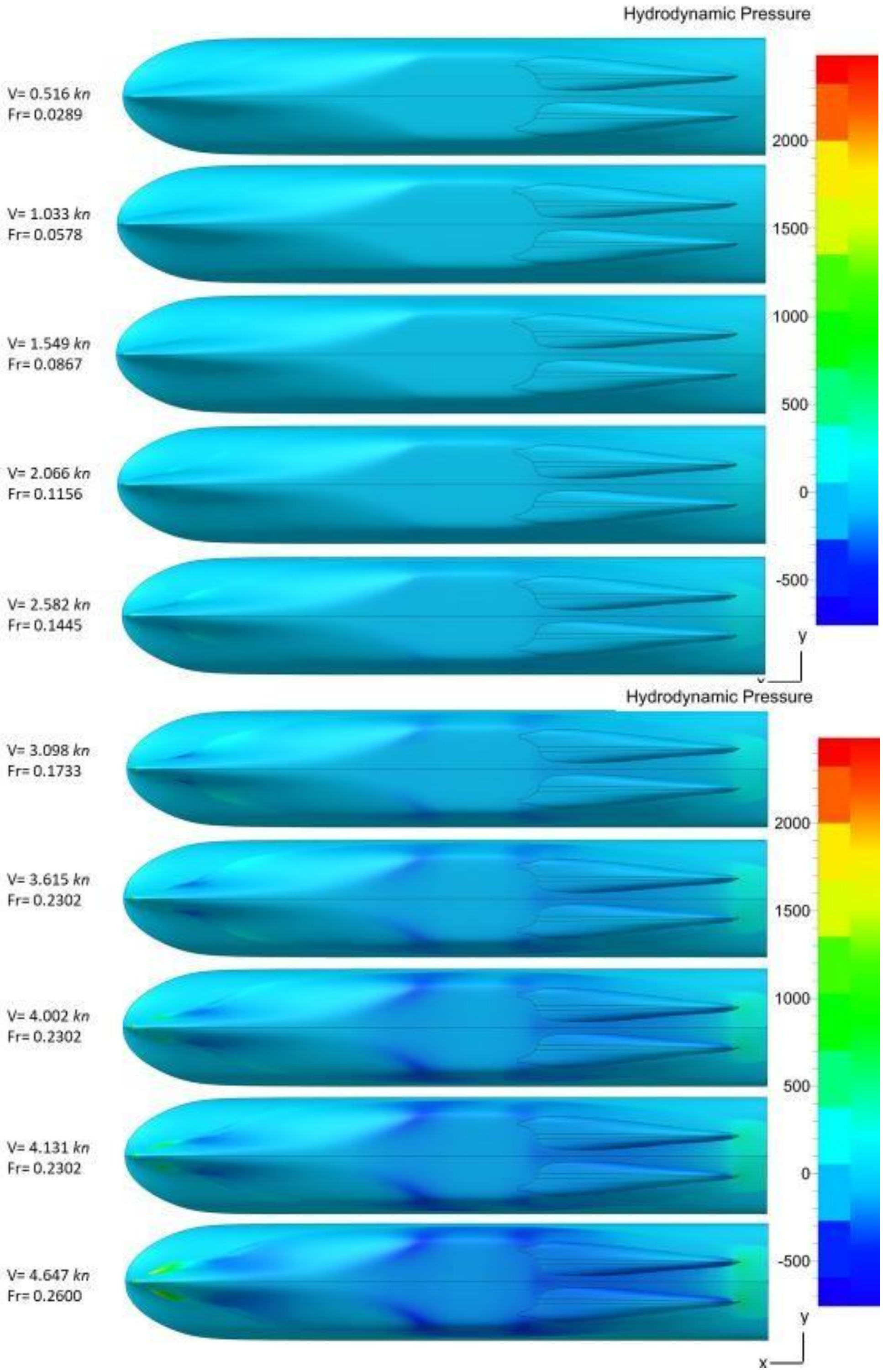

Figure 24. Hydrodynamic pressure, V_m= 0.516 kn to 4.648 kn, *Z*-plane bottom view

5. FULL SIZE MODEL OF THE FERRY

Accurate prediction of the resistance is one of the most important factors for the energy-efficient design of a ship. In general, the hydrodynamic performance of a full-scale ship could be achieved by model-scale simulation, but this method is strongly affected by the method used to extrapolate from a model scale to a ship scale. With the development of computing power, directly estimation of the ship resistance with full-scale CFD is an important approach. In this way, the need for extrapolation is eliminated.

In this study, CFD simulations were performed for the full-scale model of the ferry, for the speeds listed in Table 2. Two series of calculations were made, one is for the design draught of 5.25 m and the other is for the summer draught of 5.6 m.

5.1. Mesh generation

To generate the mesh, the procedure explained in section 4.1 was applied.

Figures 25. – 34. show some examples of the fine mesh for the full size model of the Ro-Pax ferry. For the model at the design draught the number of used cells was 1,377,466 with 1,446,503 vertices, and for the model at the summer draught the number of cells was 1,377,769 with 1,446,260 vertices.

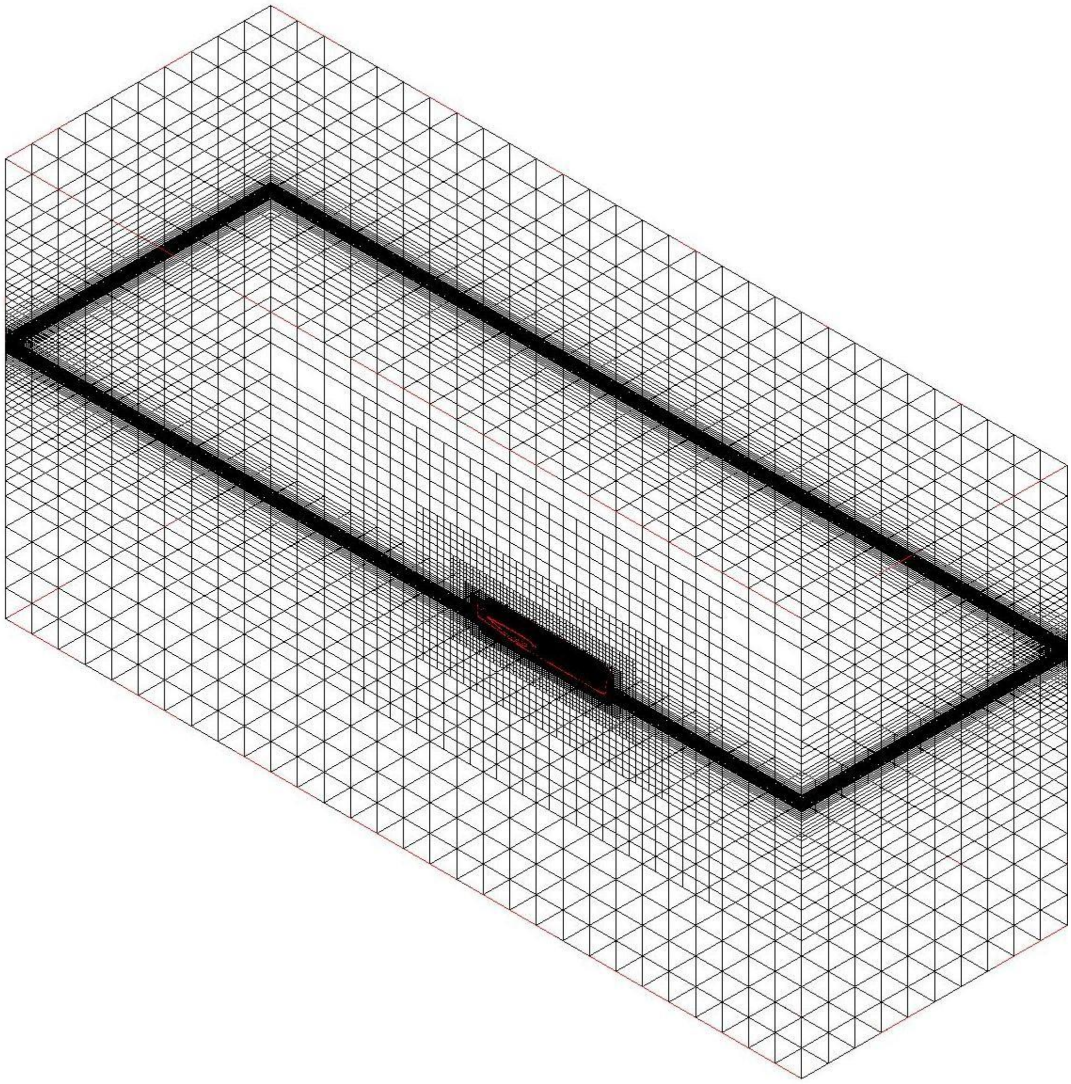

Figure 25. Fine mesh for the full scale model, *T*=5.25 m

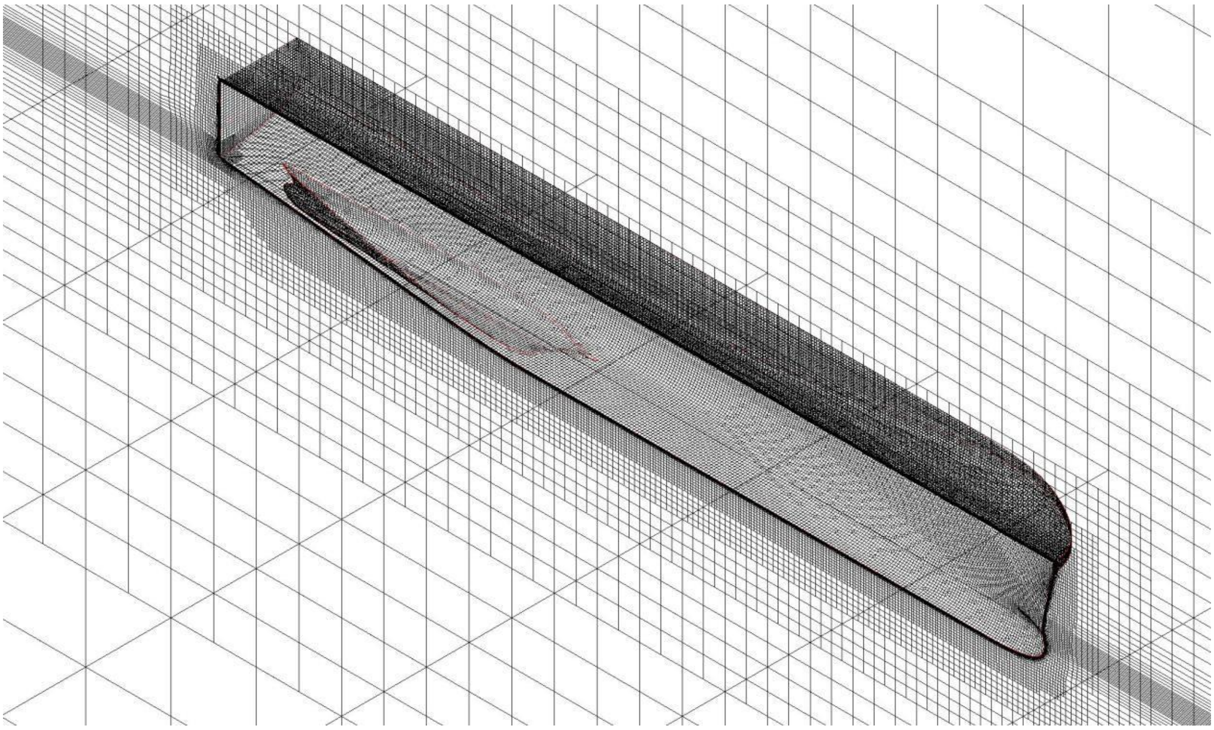

Figure 26. Fine mesh for the full scale model, *T*=5.25 m, zoomed view

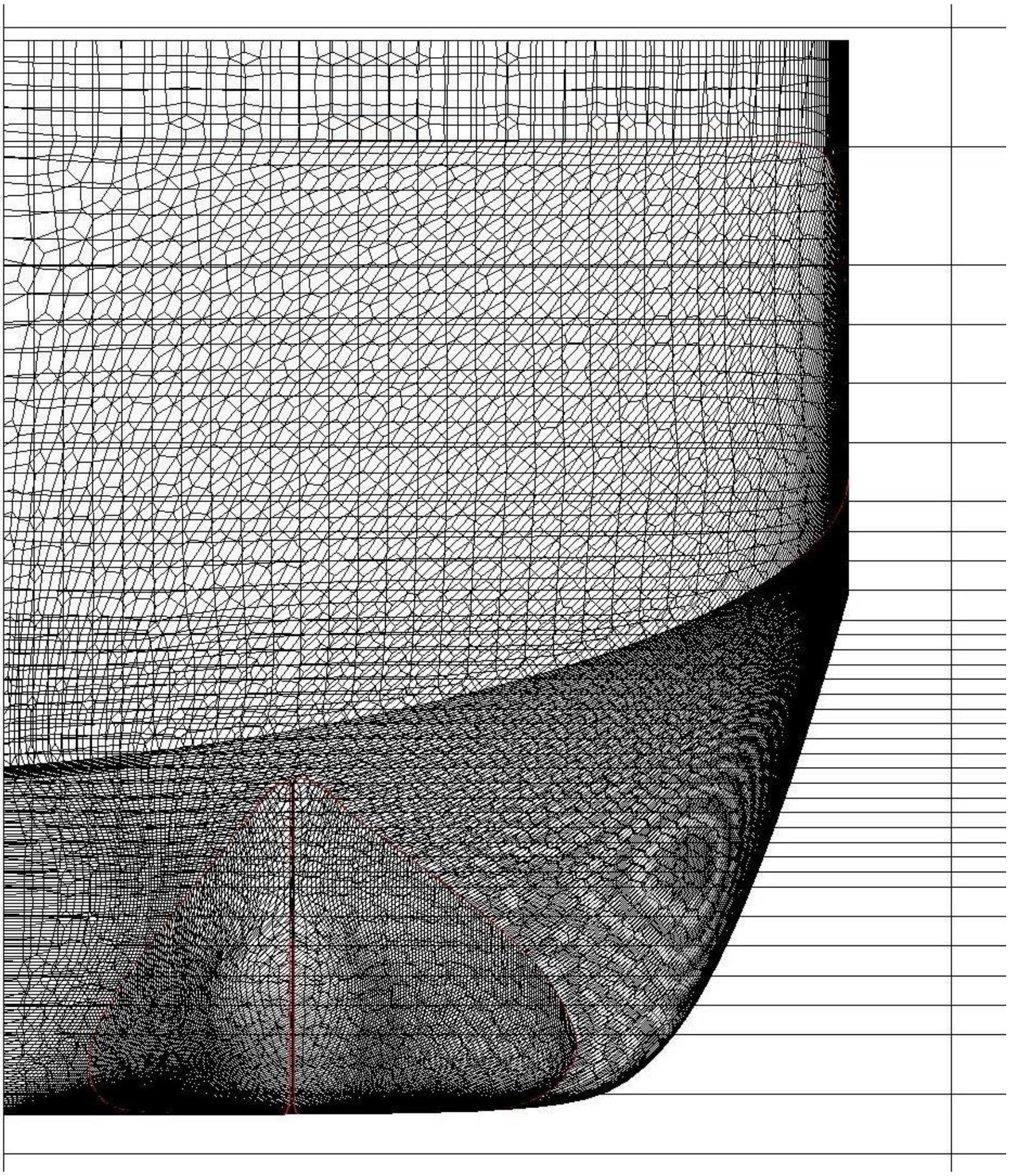

Figure 27. Fine mesh for the full scale model, *T*=5.25 m, *X*-plane view

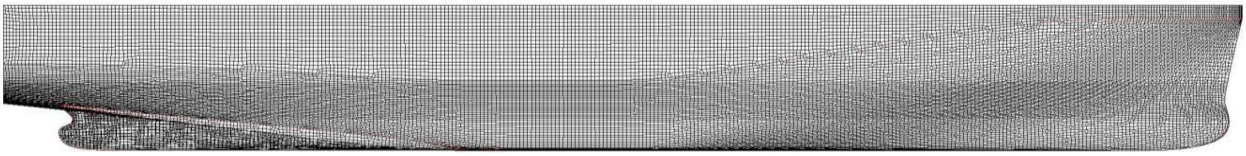

Figure 28. Fine mesh for the full scale model, *T*=5.25 m, *Y*-plane view

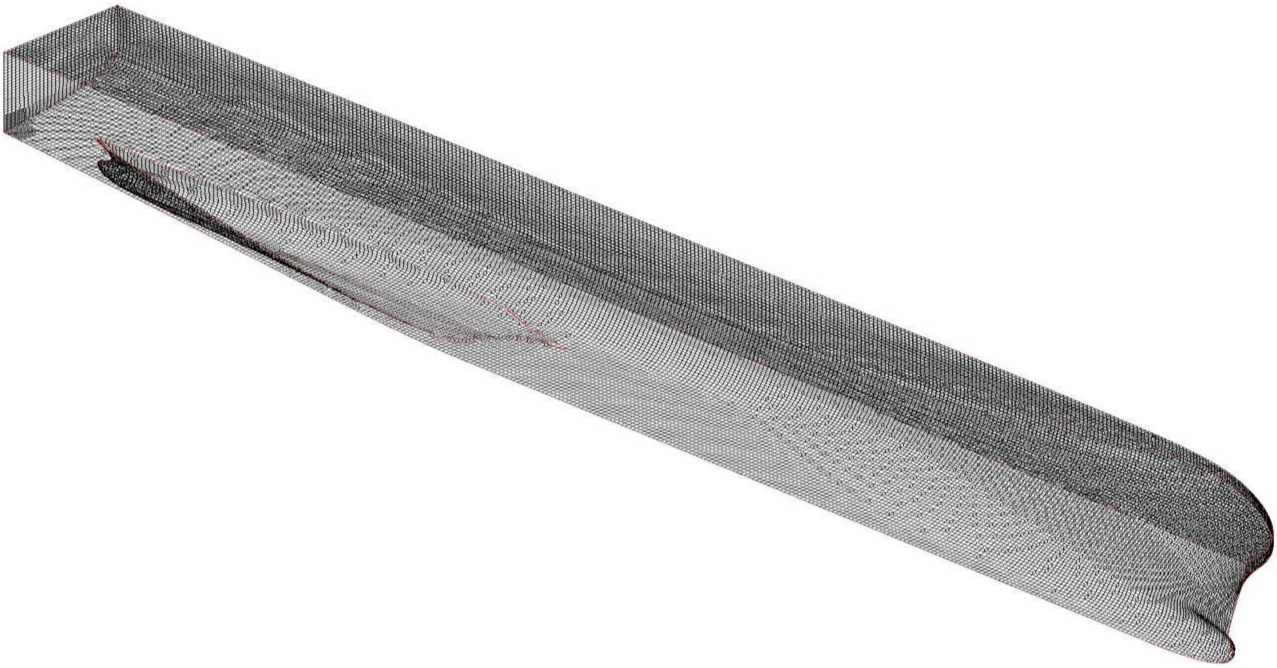

Figure 29. Fine mesh for the full scale model, *T*=5.25 m, isometric view

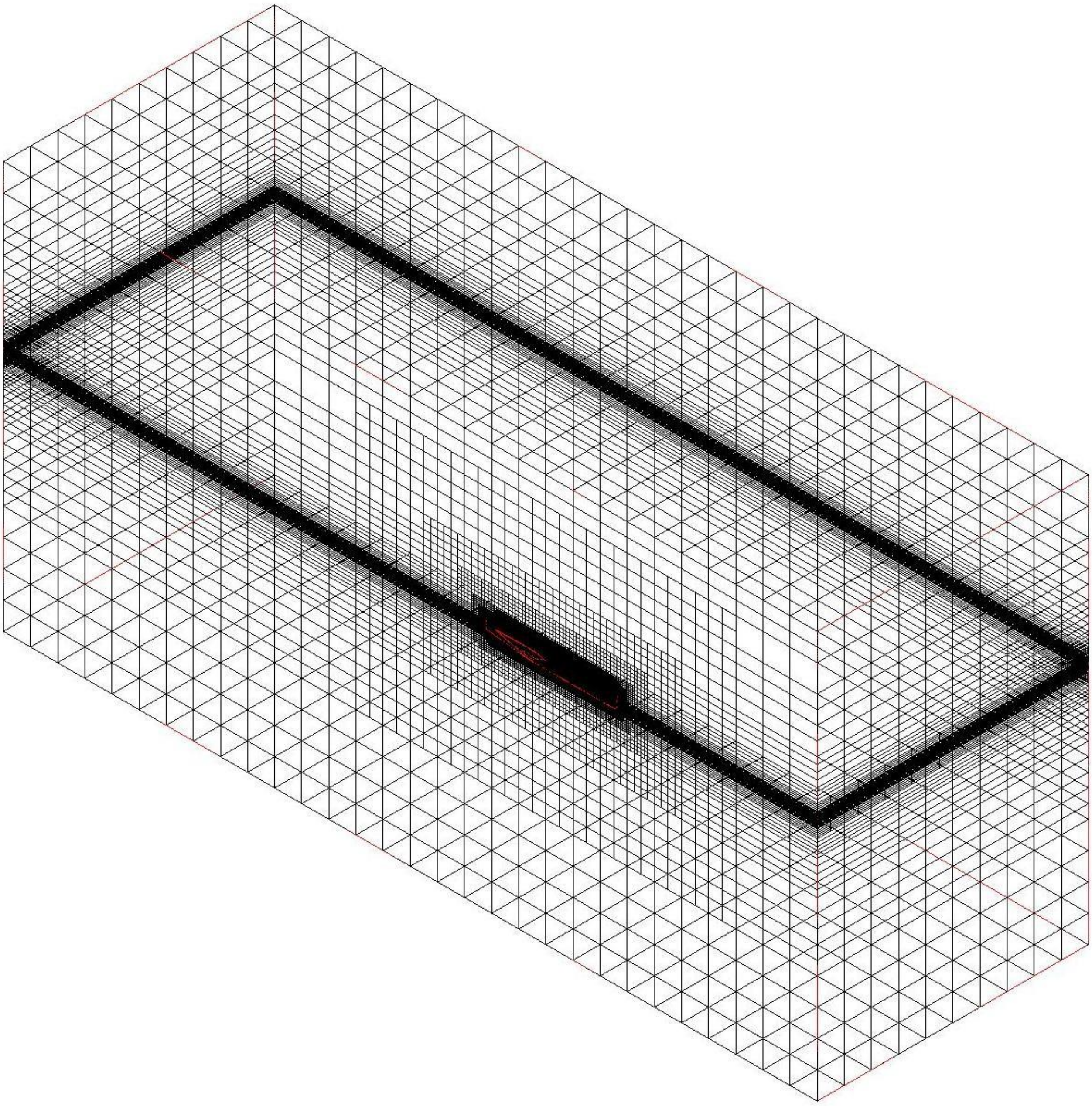

Figure 30. Fine mesh for the full scale model, *T*=5.6 m

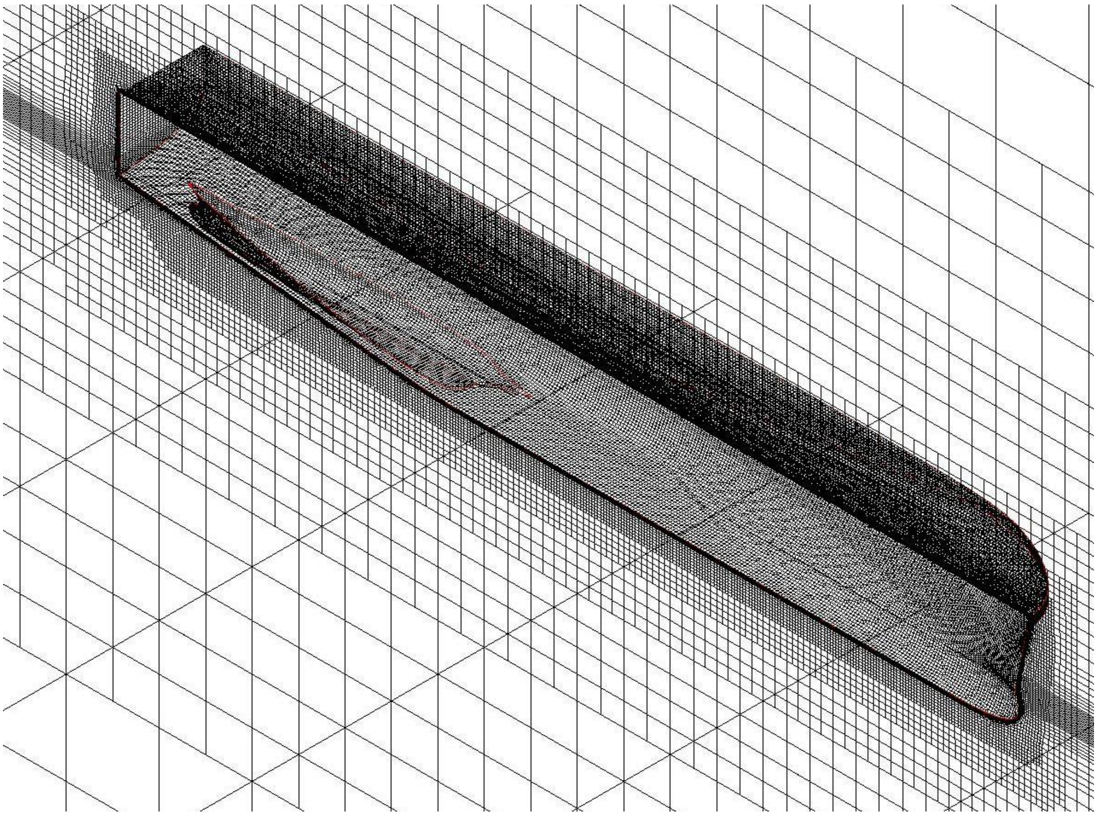

Figure 31. Fine mesh for the full scale model, *T*=5.6 m, zoomed view

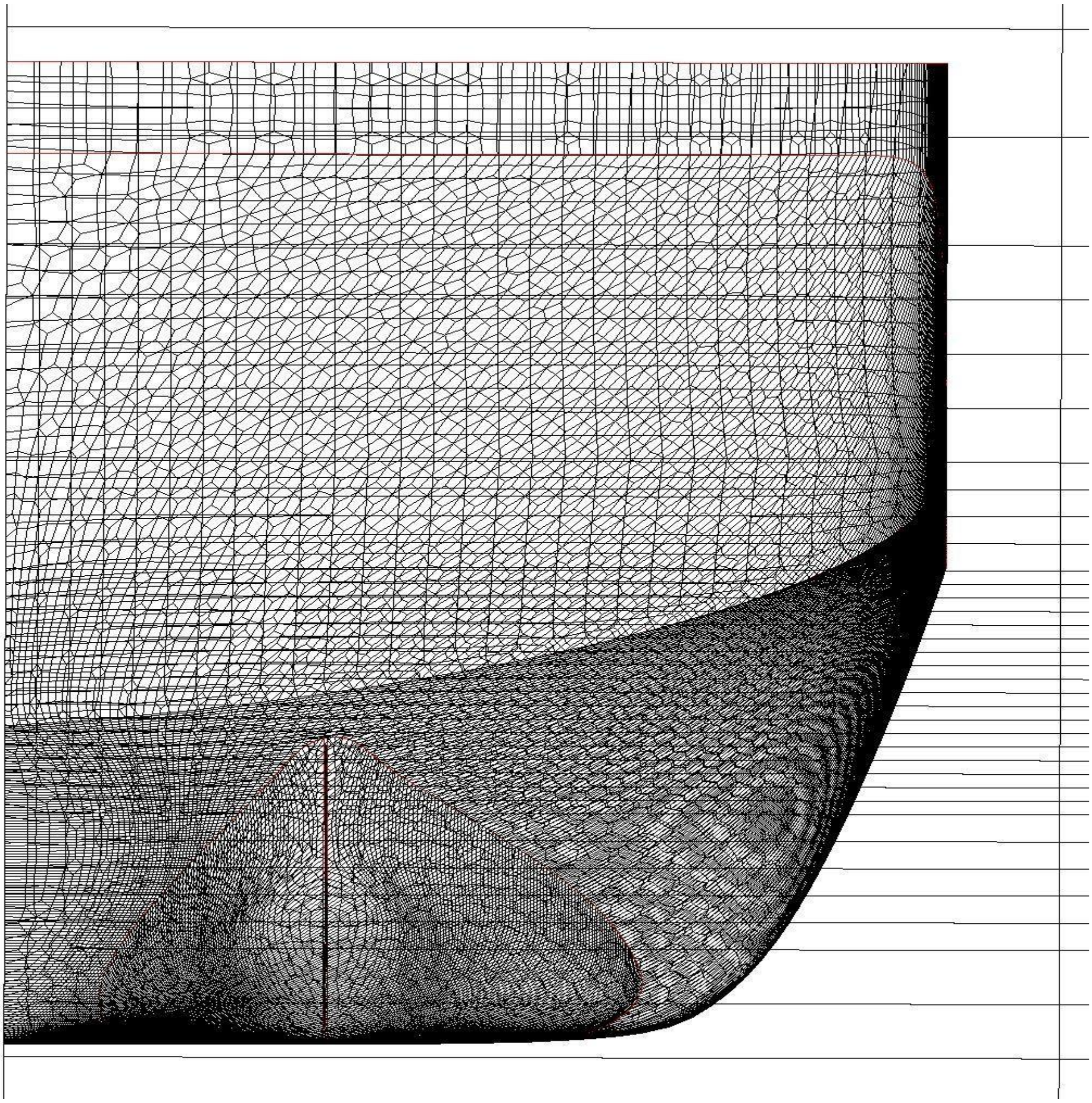

Figure 32. Fine mesh for the full scale model, *T*=5.6 m, *X*-plane view

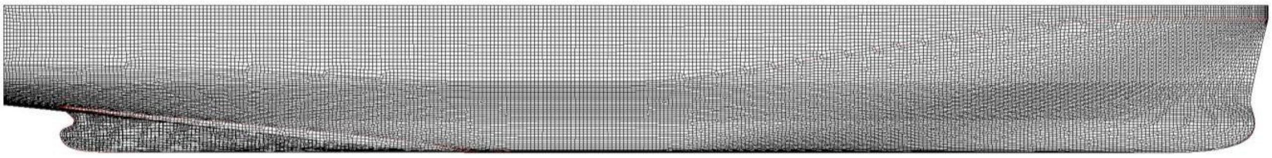

Figure 33. Fine mesh for the full scale model, *T*=5.6 m, *Y*-plane view

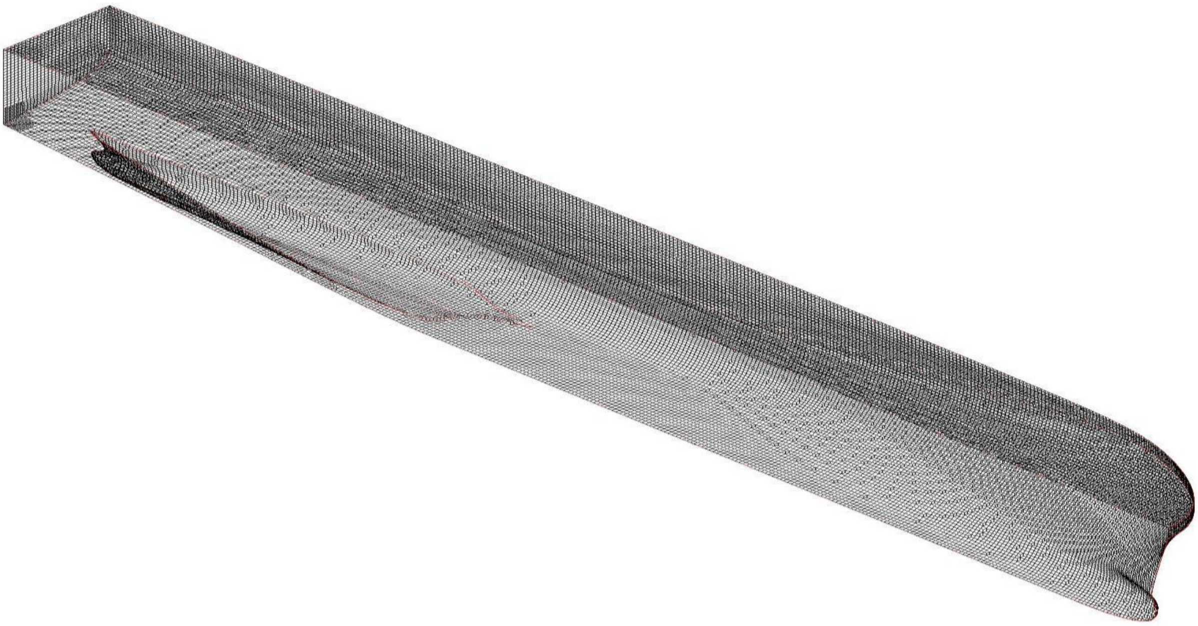

Figure 34. Fine mesh for the full scale model, *T*=5.6 m, isometric view

5.2. Computations

For the computations of the full scale model, the procedure explained in section 4.2 was applied.

The fluid characteristics were set as:

- water: density 1024.8 kg/m^3, dynamic viscosity 0.001077 x 10^{-3} N s/m^2,
- air: density 1.2 kg/m^3, dynamic viscosity 1.85 x 10^{-5} N s/m^2.

5.3. Results

In this section the results of extensive CFD analysis were presented. The results obtained for the design and summer draught are presented in sections 5.3.1 and 5.3.2.

5.3.1. Design draught *T* = 5.25 m

Table 6. shows the numerically obtained resistance for the corresponding Froude numbers at design draught of 5.25 m. The numerical values are graphically shown in the Figure 35.

Table 6. Resistance vs. Froude number

V_{KN}, knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0	15.5	16.0	18.0
V, m/s	1.029	2.058	3.087	4.116	5.144	6.173	7.202	7.974	8.231	9.260
Fr	0.029	0.058	0.087	0.116	0.144	0.173	0.202	0.224	0.231	0.260
Fx, kN	6.662	19.807	36.212	62.612	104.802	165.451	246.291	324.334	355.175	492.910

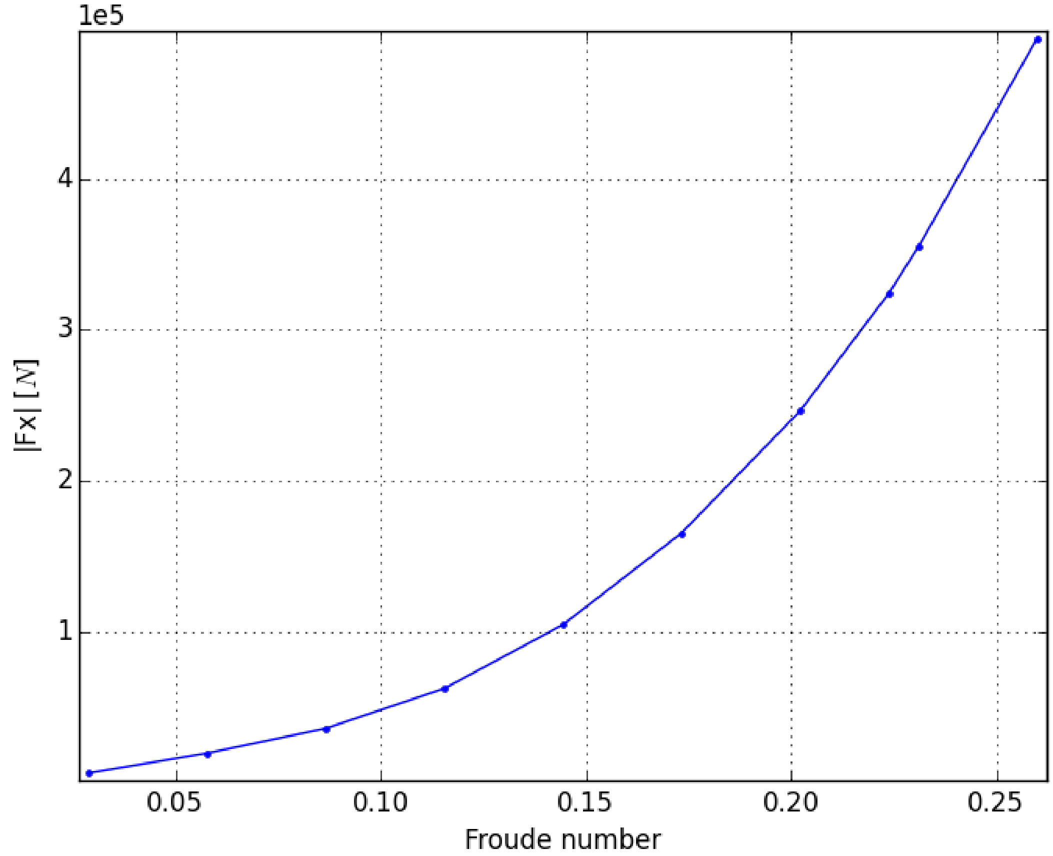

Figure 35. Resistance vs. Froude number

Figures 36. – 45. visually show some results obtained by CFD simulations. First, the results for the design speed of 15.5 knots and then the results for other speeds between 2.0 knots and 18.0 knots were shown.

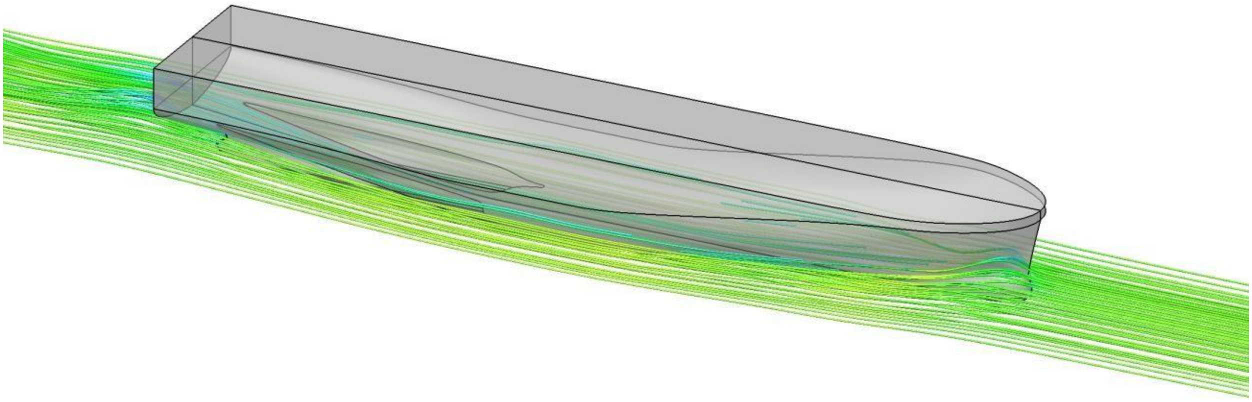

Figure 36. Relative velocity streamlines, design speed V= 15.5 knots

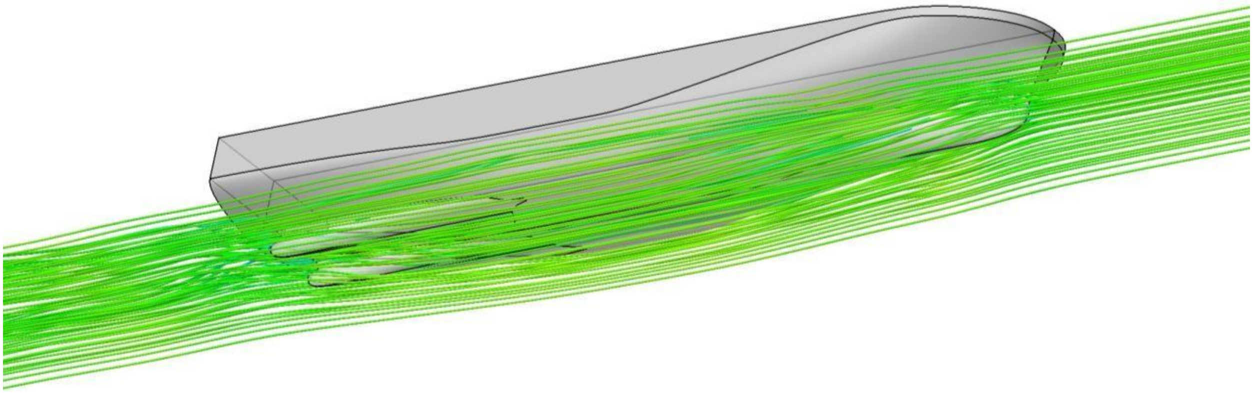

Figure 37. Relative velocity streamlines, design speed V= 15.5 knots

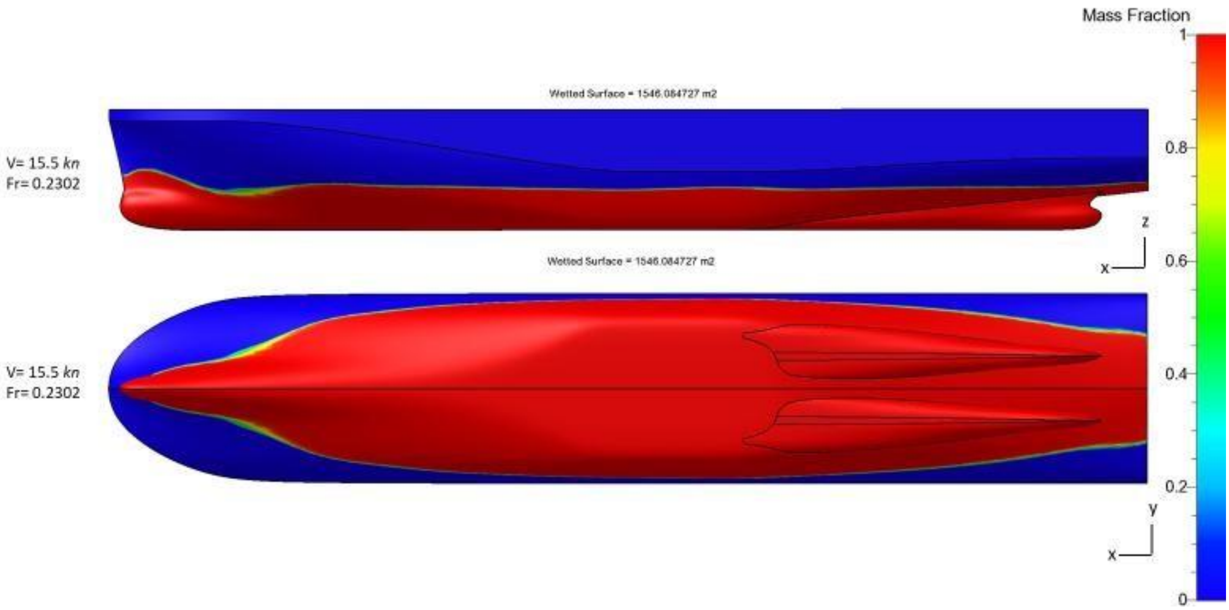

Figure 38. Wetted surface, design speed *V*= 15.5 knots

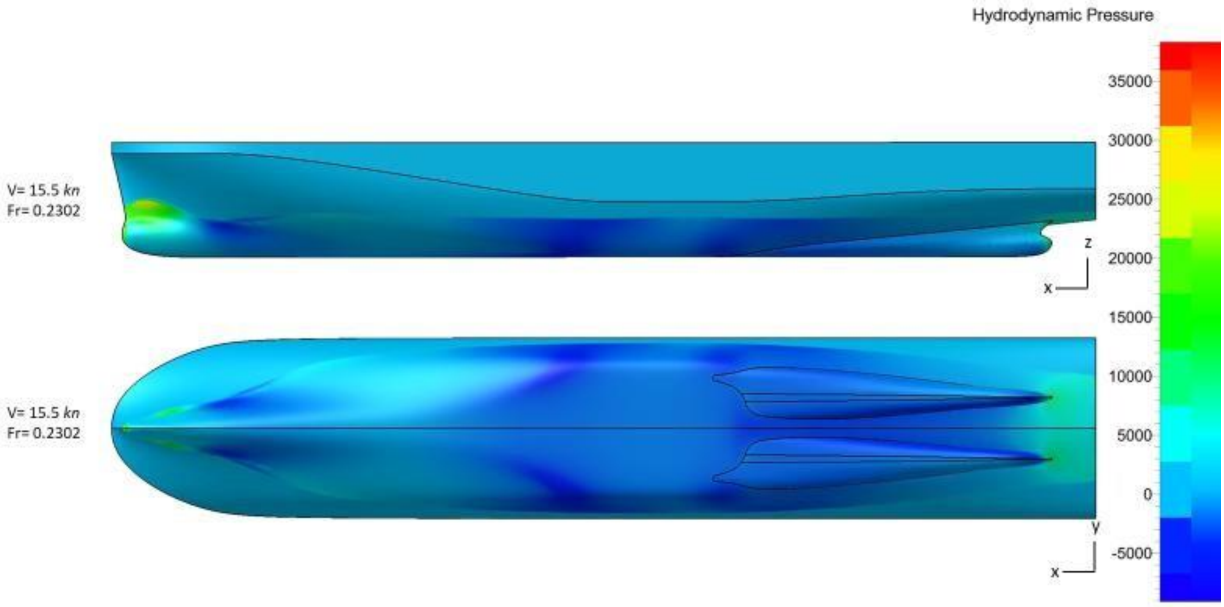

Figure 39. Hydrodynamic pressure, design speed *V*= 15.5 knots

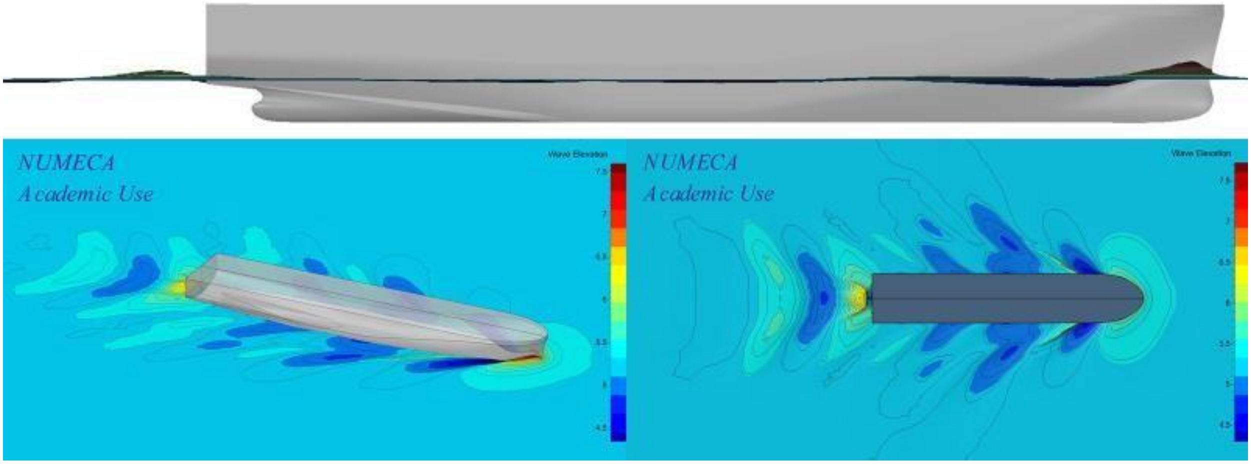

Figure 40. Wave elevations, design speed *V*= 15.5 knots

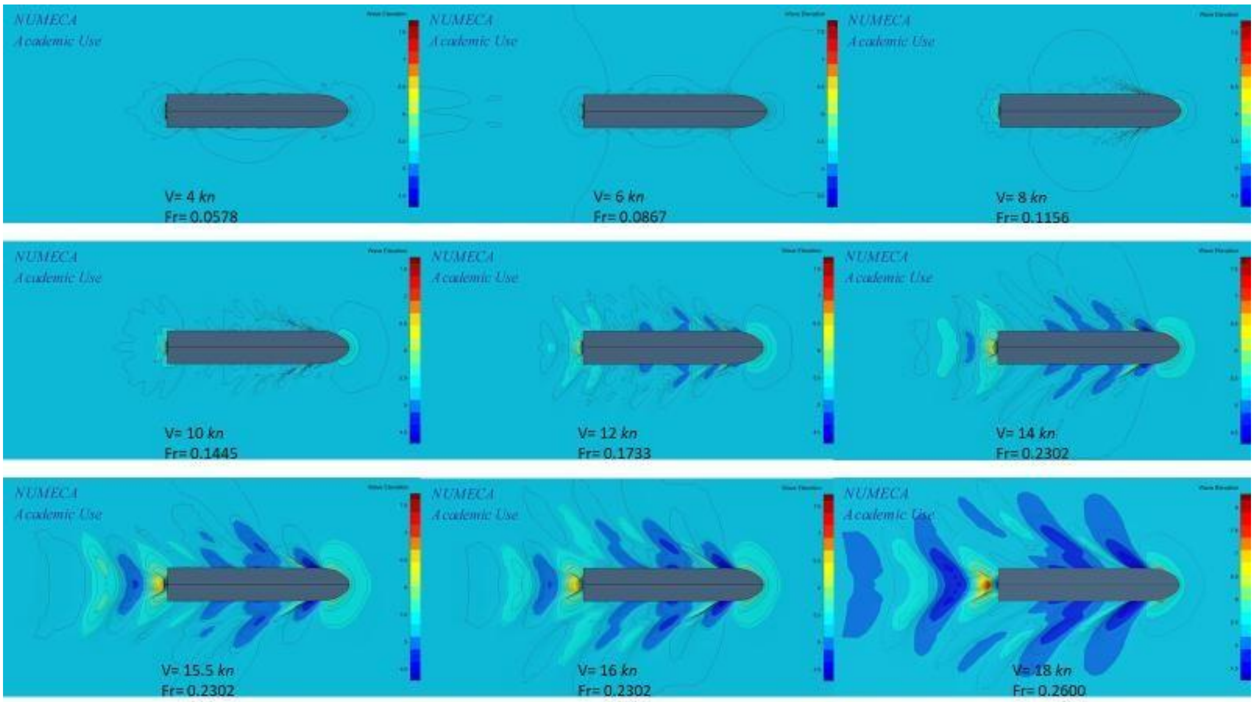

Figure 41. Wave elevations, *V*= 4.0 knots to 18.0 knots, *Y*-plane view

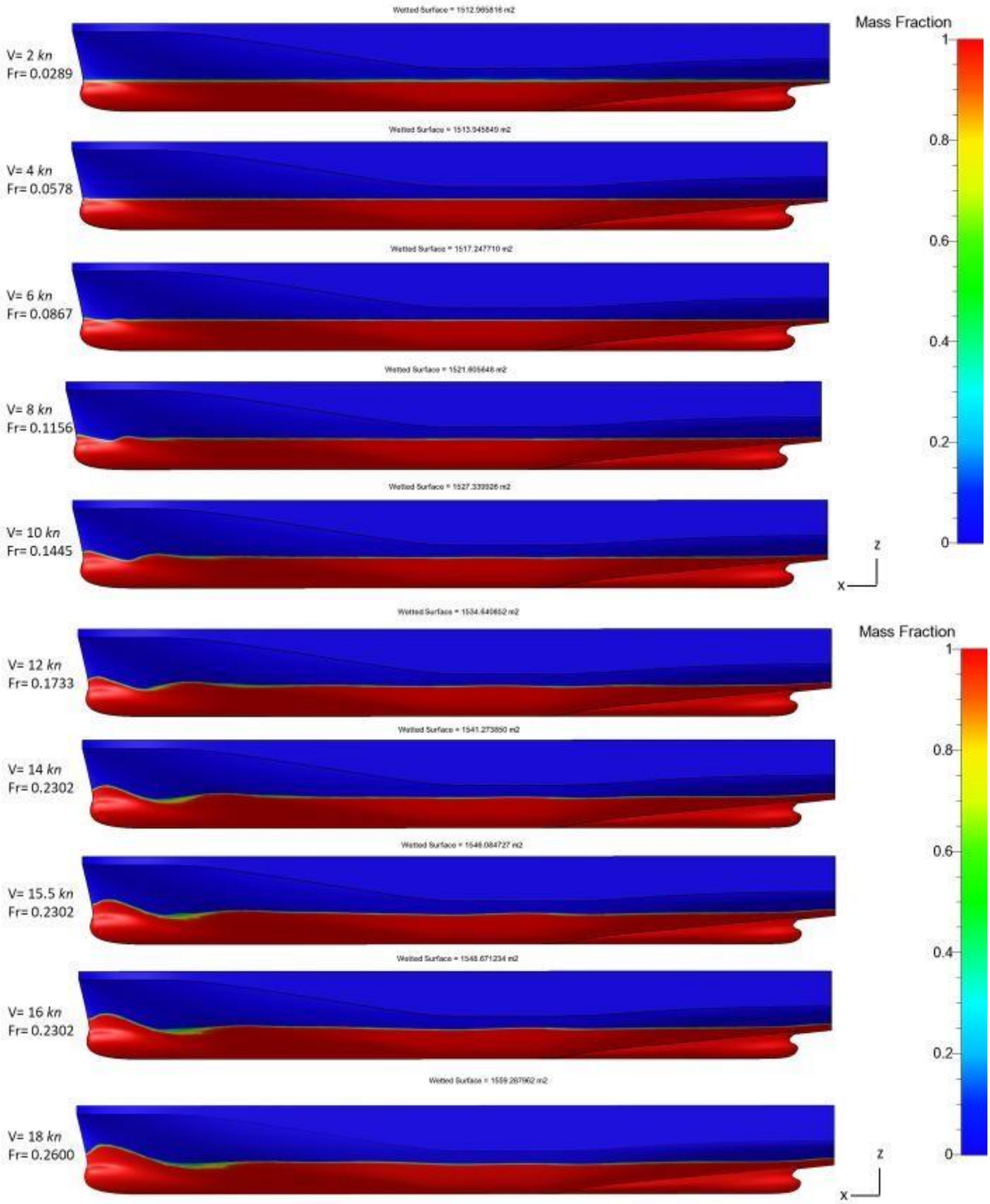

Figure 42. Wetted surface, *V*= 2.0 knots to 18.0 knots, *Y*-plane view

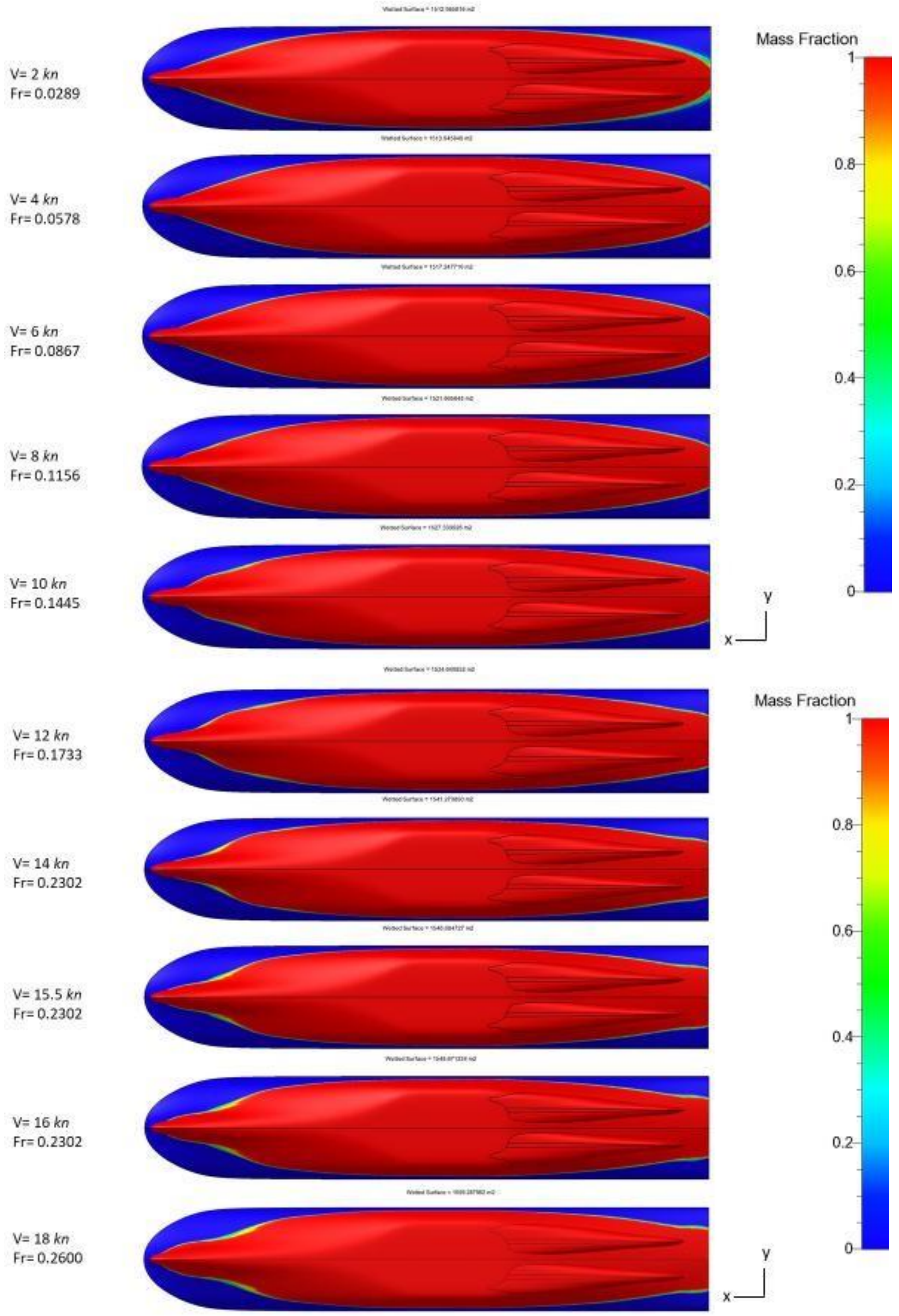

Figure 43. Wetted surface, *V*= 2.0 knots to 18.0 knots, *Z*-plane bottom view

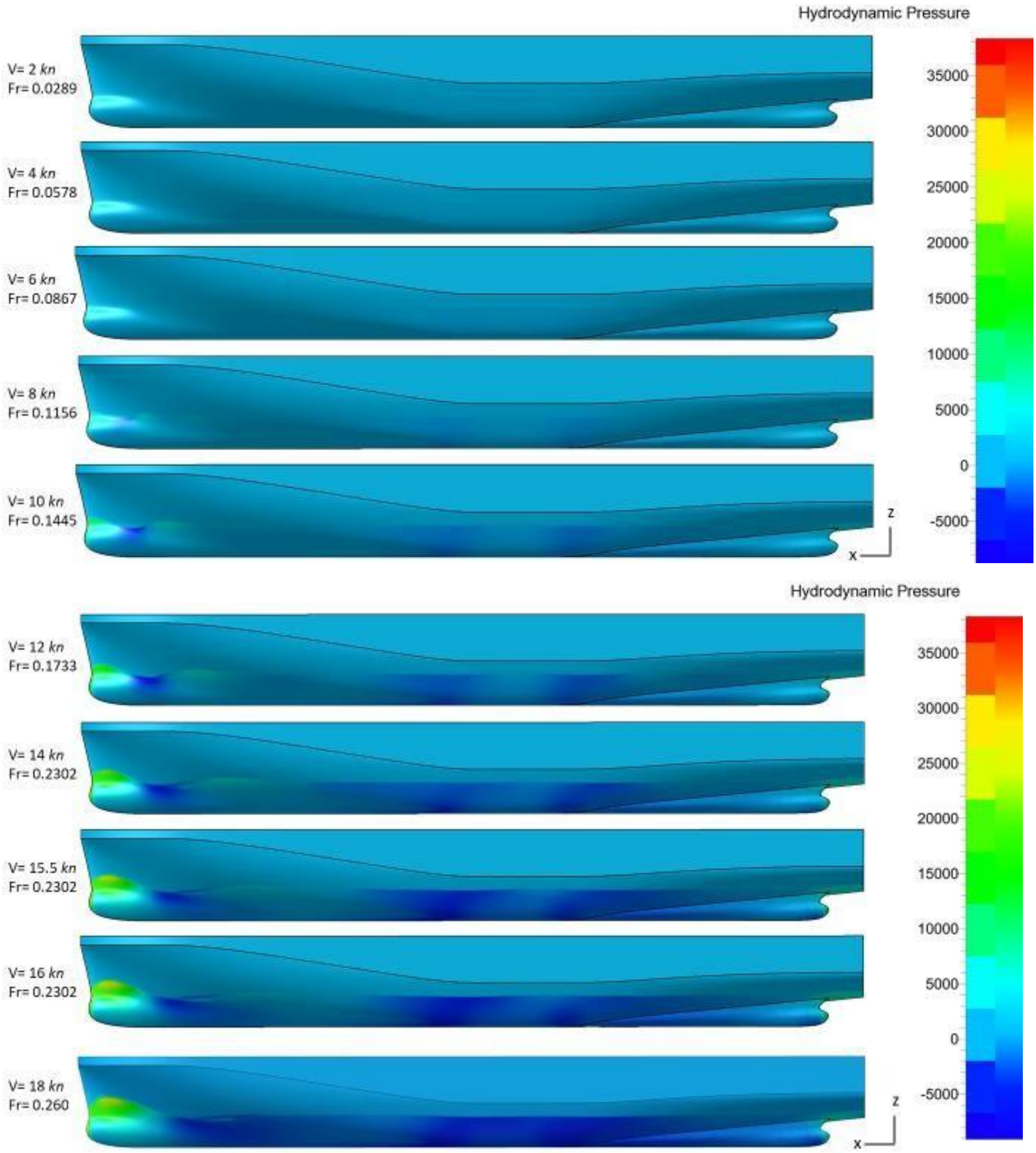

Figure 44. Hydrodynamic pressure, *V*= 2.0 knots to 18.0 knots, *Y*-plane view

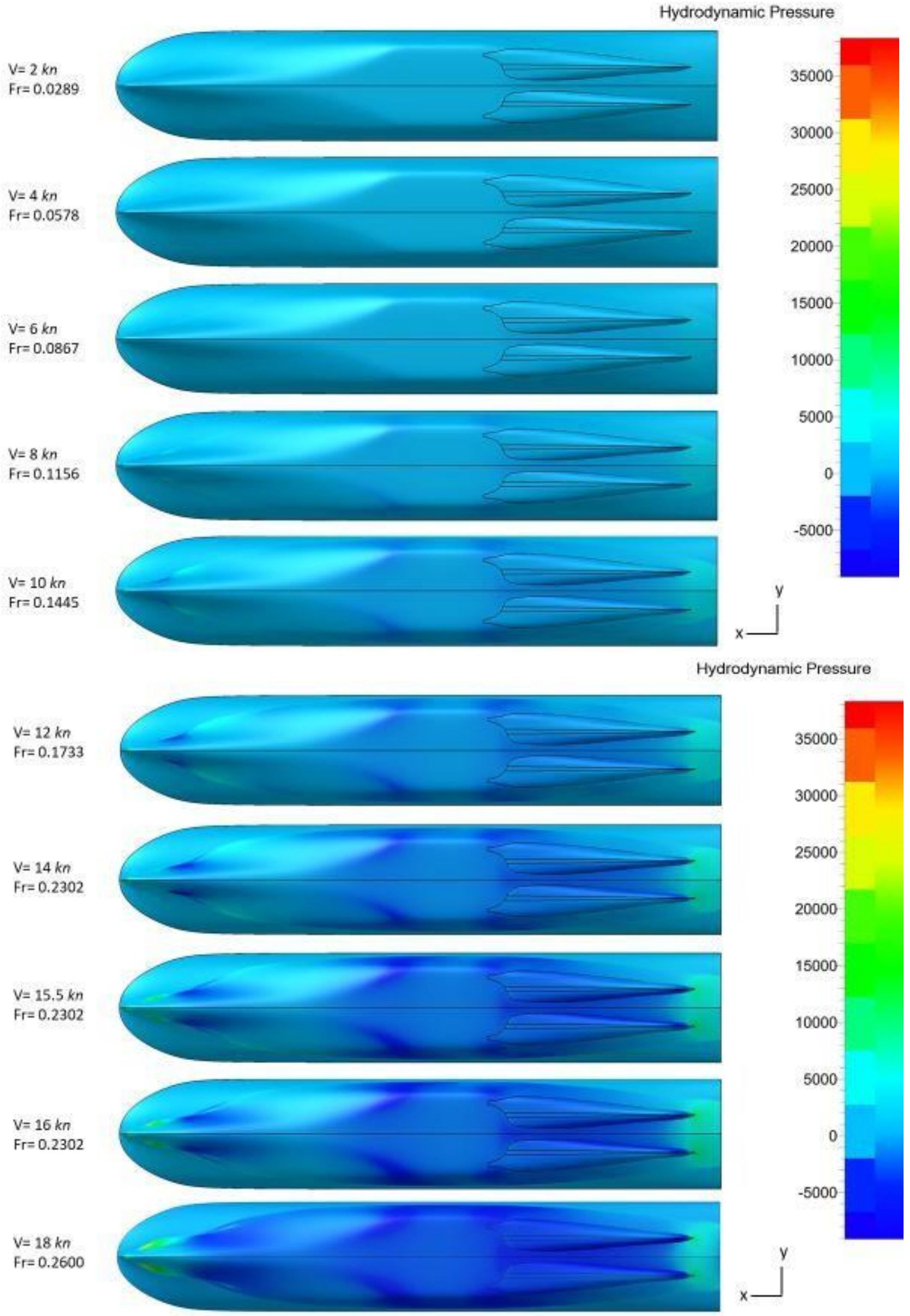

Figure 45. *Hydrodynamic pressure,* V= *2.0 knots to 18.0 knots,* Z*-plane bottom view*

5.3.2. Summer draught T = 5.6 m

Table 7. shows the numerically obtained resistance for the corresponding Froude numbers at summer draught of 5.6 m. The numerical values are graphically shown in the Figure 46.

Table 7. Resistance vs. Froude number

V_{KN}, knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0	15.5	16.0	18.0
V, m/s	1.029	2.058	3.087	4.116	5.144	6.173	7.202	7.974	8.231	9.260
Fr	0.029	0.058	0.087	0.116	0.144	0.173	0.202	0.224	0.231	0.260
Fx, kN	5.556	22.955	39.7	68.277	112.227	178.476	268.794	356.176	392.612	553.516

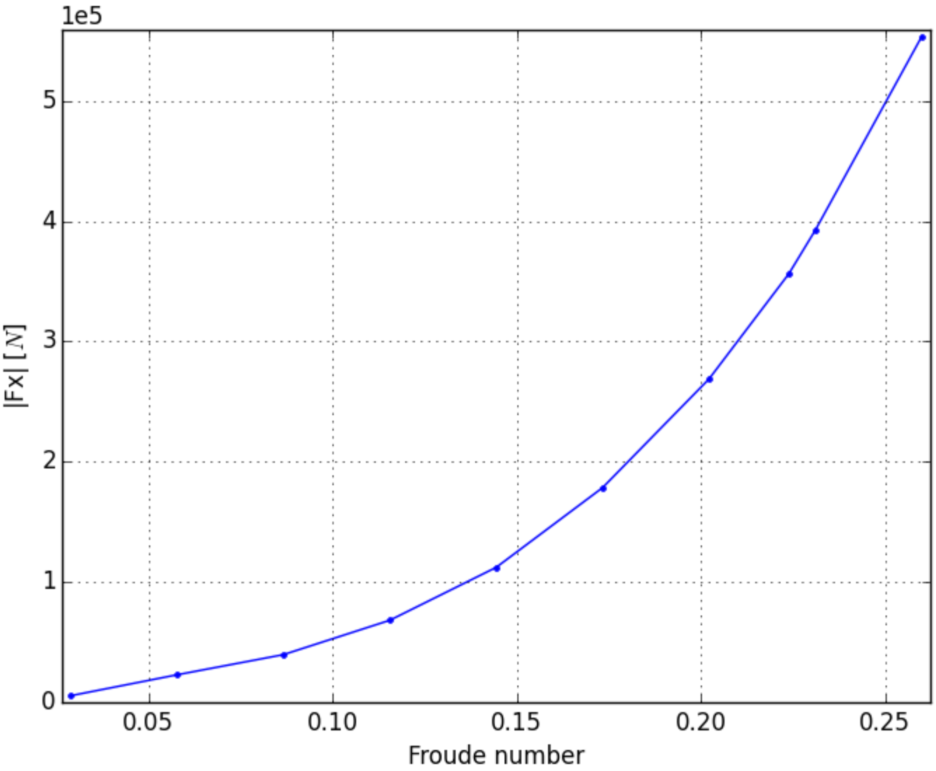

Figure 46. Resistance vs. Froude number

Figures 47. – 56. visually show some results obtained by CFD simulations. First, the results for the design speed of 15.5 knots and then the results for other speeds between 2.0 knots and 18.0 knots were shown.

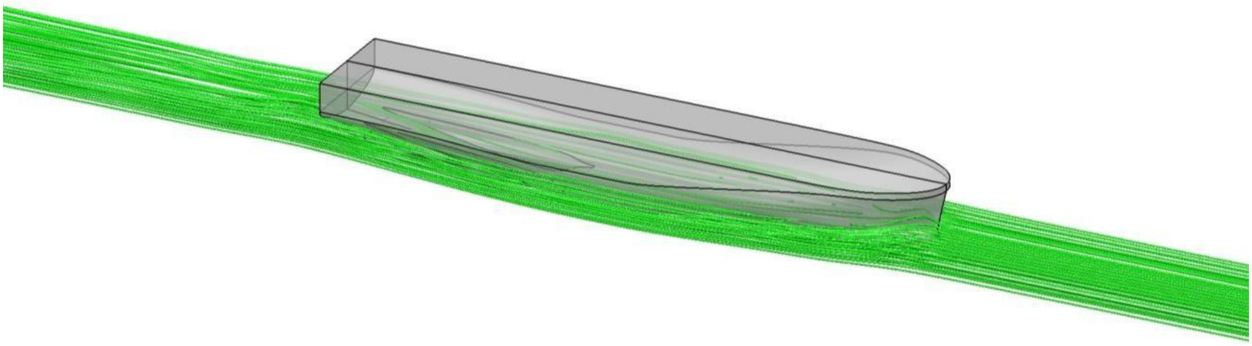

Figure 47. Relative velocity streamlines, design speed V= 15.5 knots

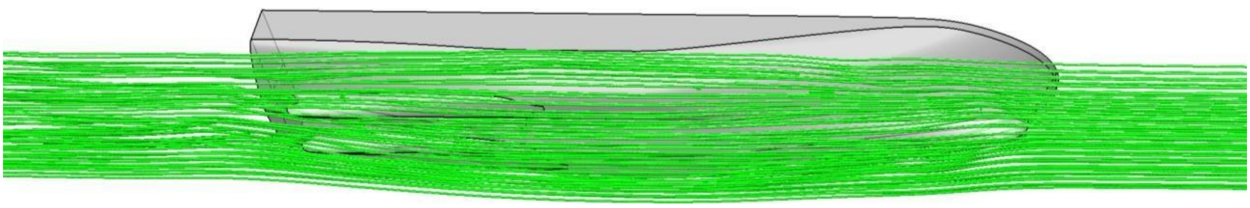

Figure 48. Relative velocity streamlines, design speed V= 15.5 knots

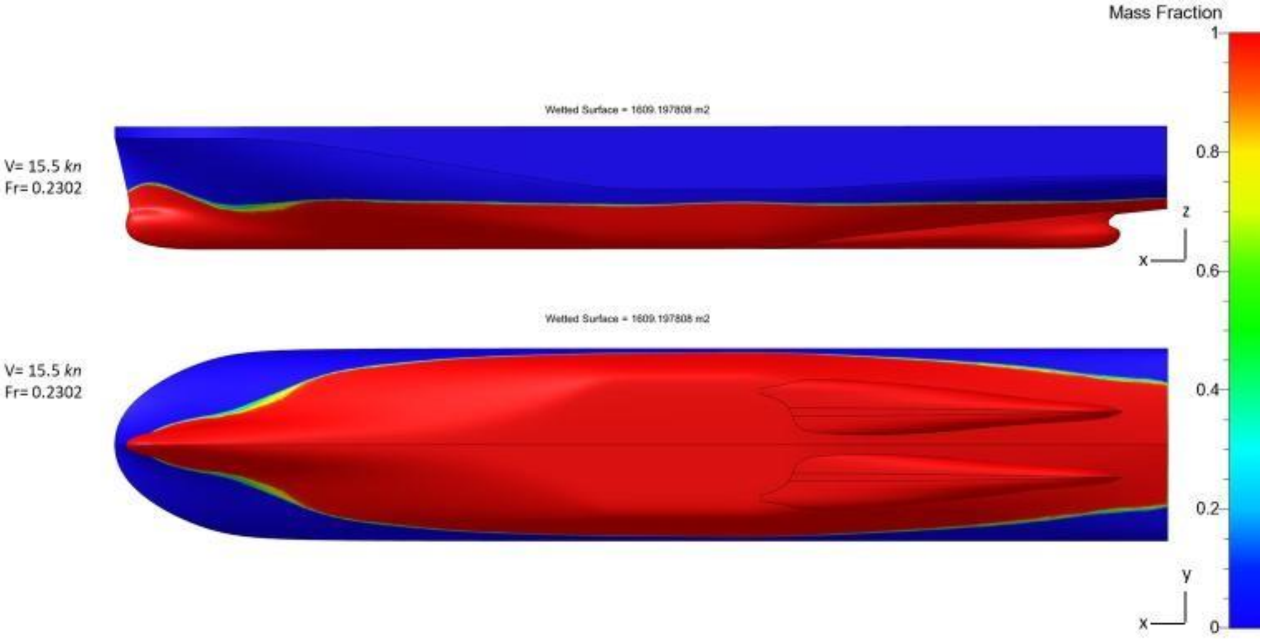

Figure 49. Wetted surface, design speed *V*= 15.5 knots

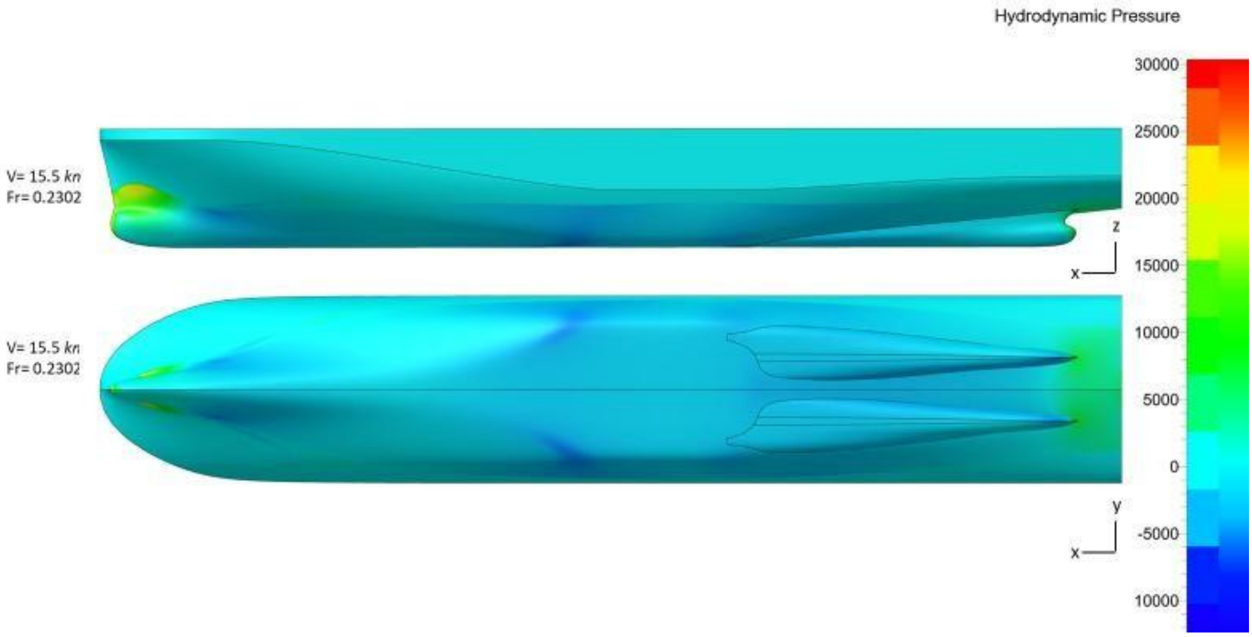

Figure 50. Hydrodynamic pressure, design speed *V*= 15.5 knots

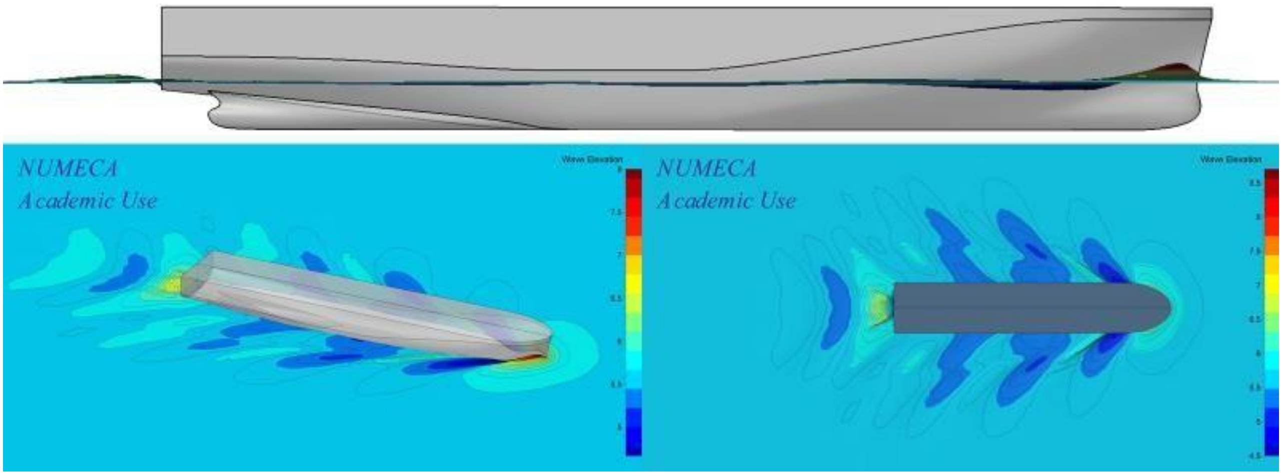

Figure 51. Wave elevations, design speed *V*= 15.5 knots

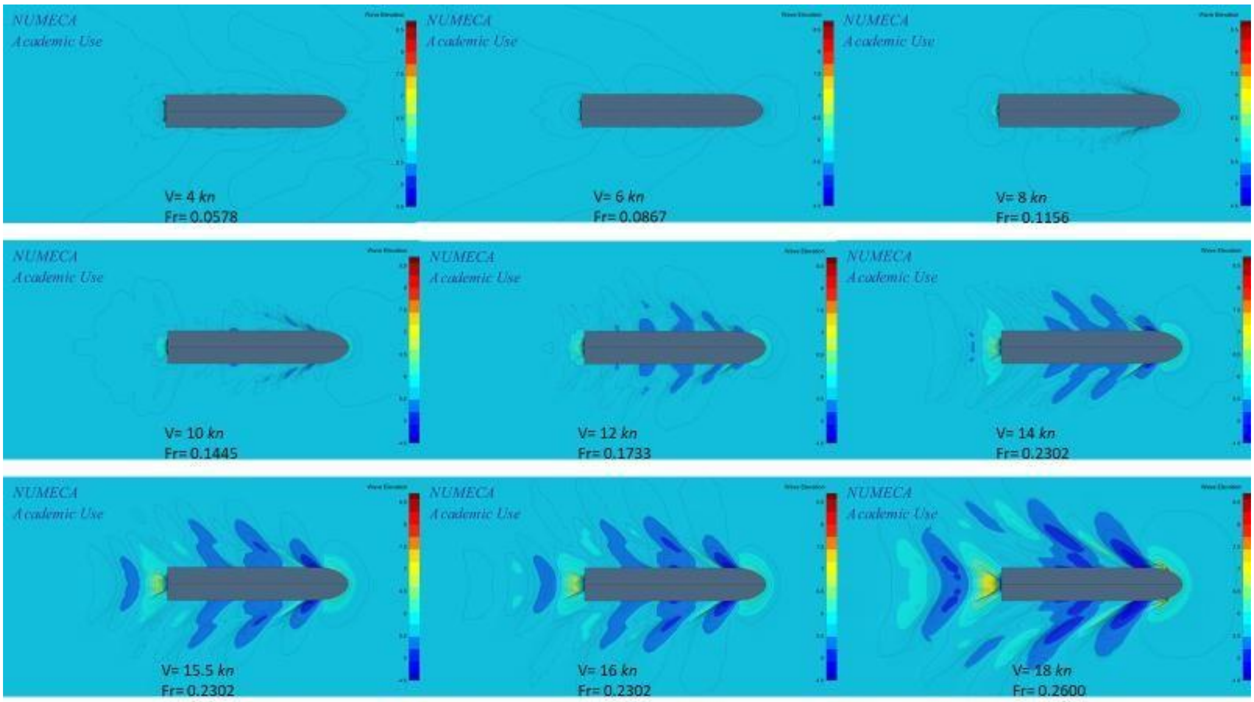

Figure 52. Wave elevations, *V*= 4.0 knots to 18.0 knots, *Y*-plane view

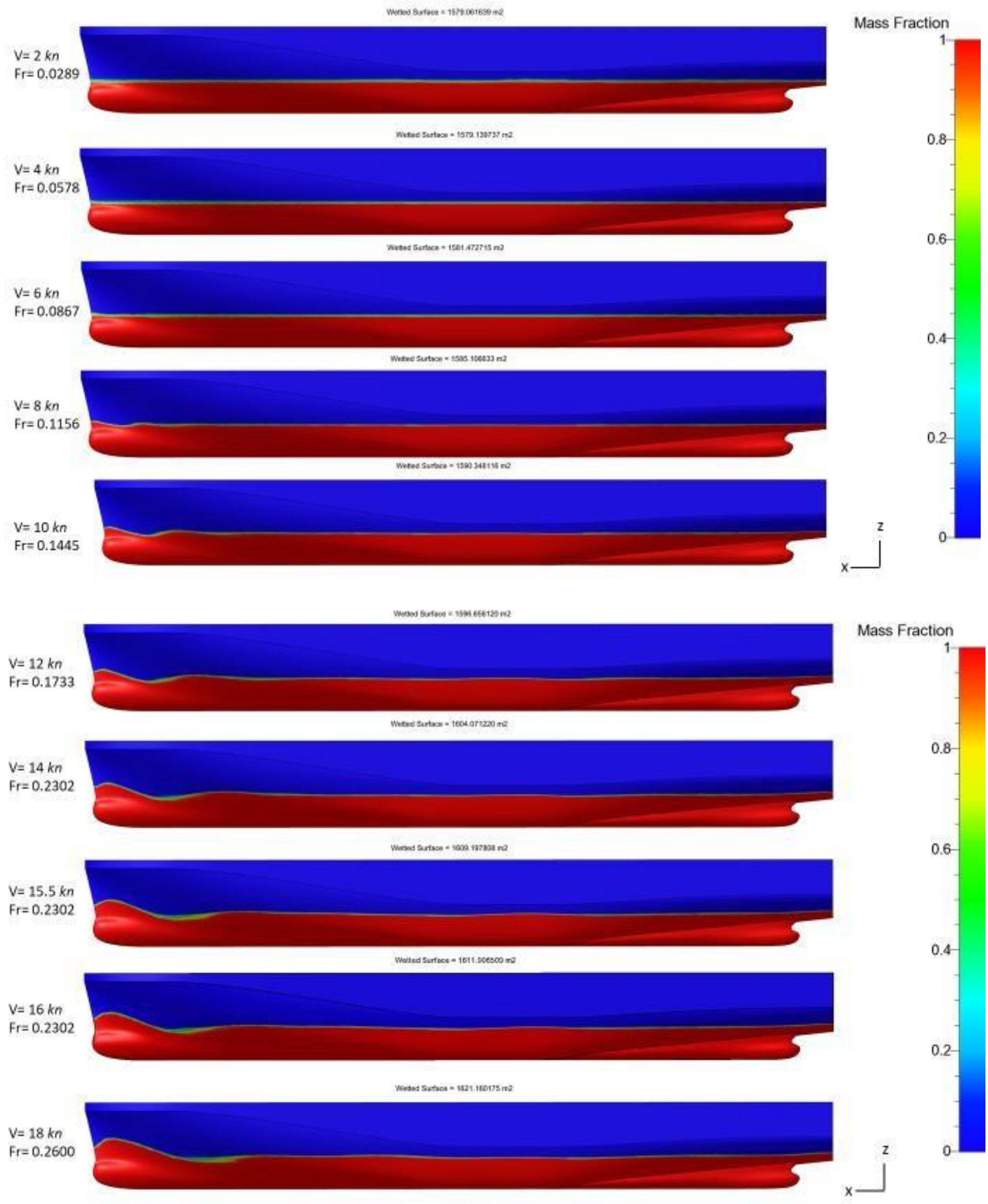

Figure 53. Wetted surface, *V*= 2.0 knots to 18.0 knots, *Y*-plane view

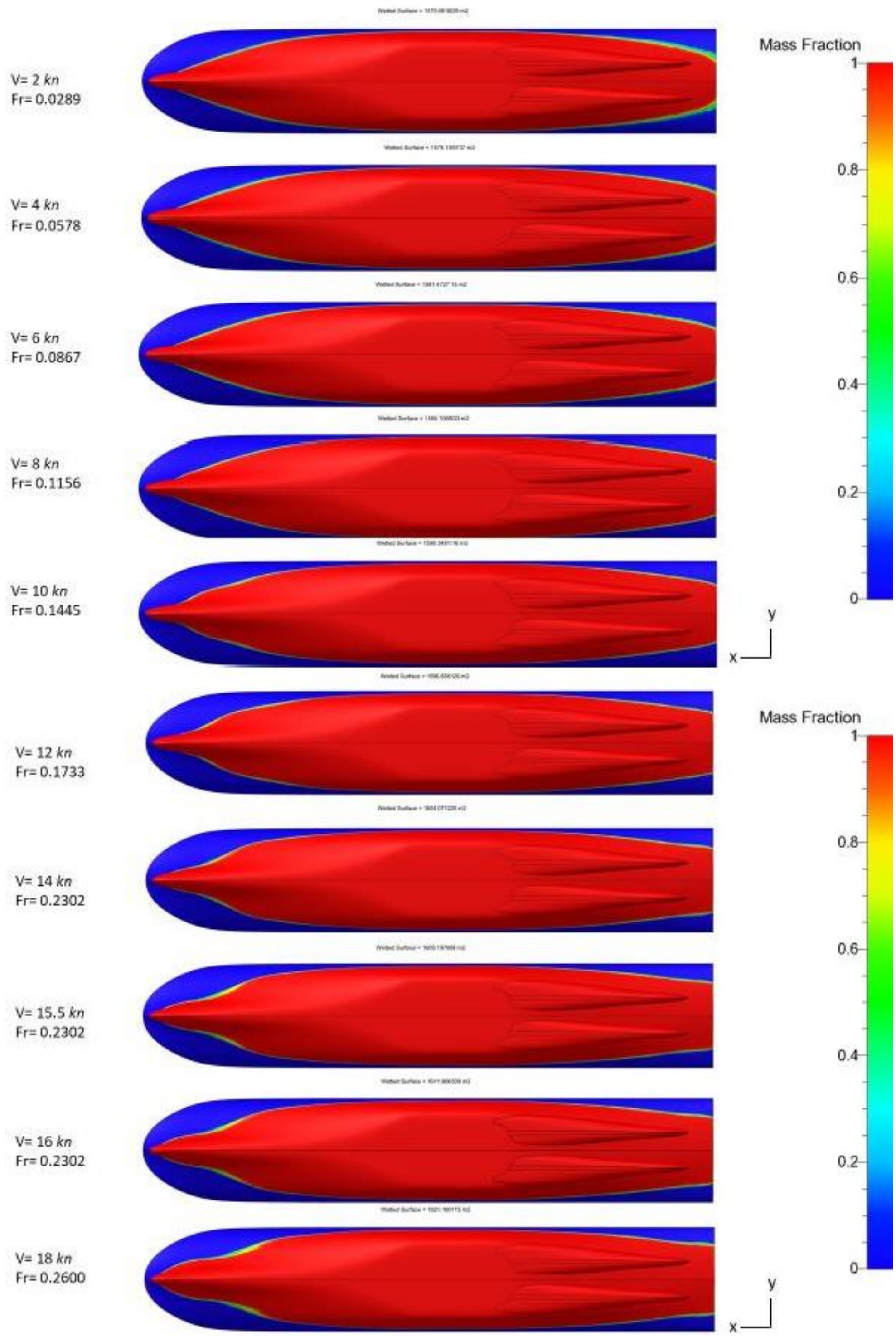

Figure 54. Wetted surface, *V*= 2.0 knots to 18.0 knots, *Z*-plane bottom view

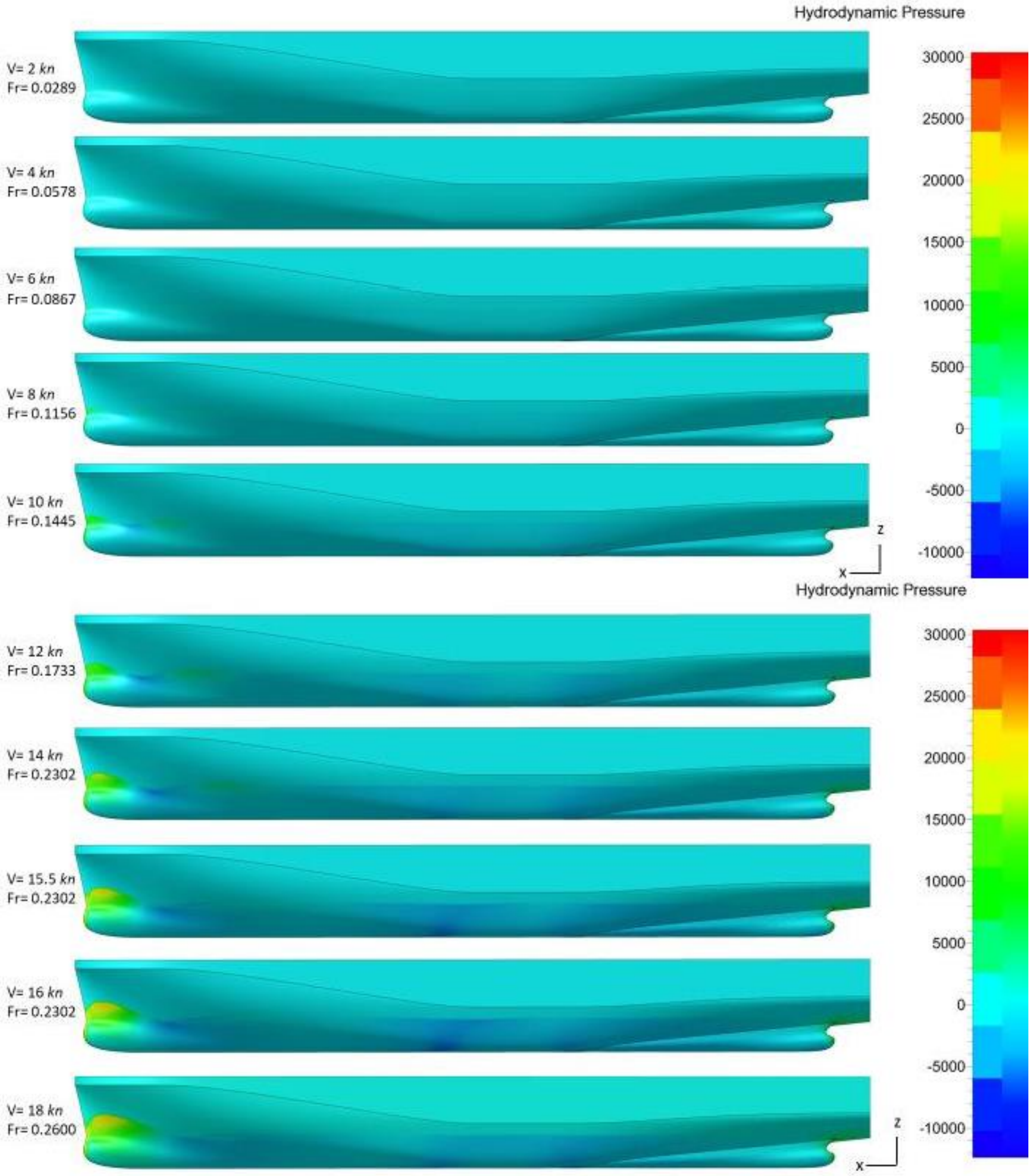

Figure 55. Hydrodynamic pressure, *V*= 2.0 knots to 18.0 knots, *Y*-plane view

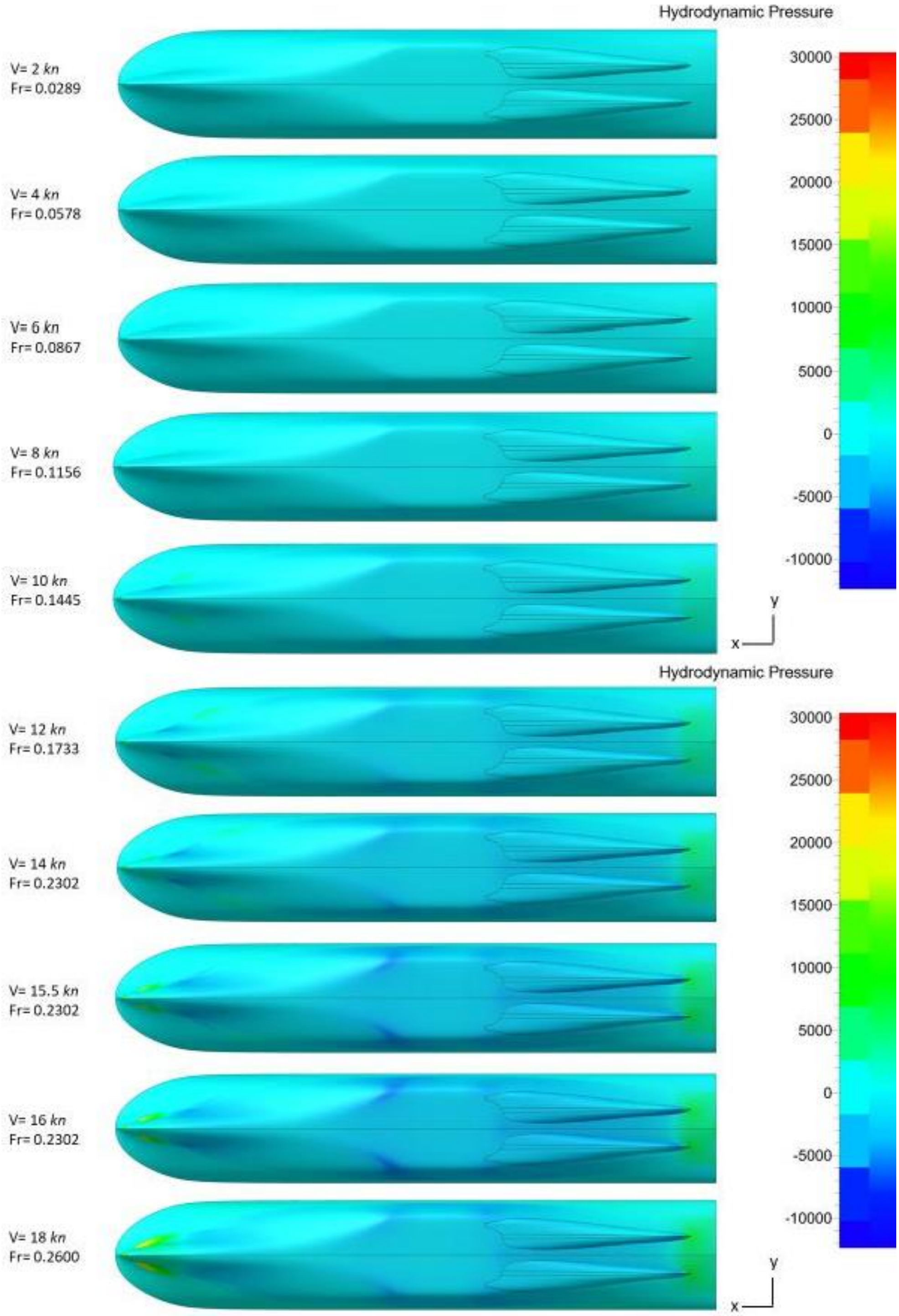

Figure 56. Hydrodynamic pressure, *V*= 2.0 knots to 18.0 knots, *Z*-plane bottom view

6. THE POWER-SPEED DIAGRAM

This estimation of the power-speed is made after the CFD analysis of the Ro-Pax ferry is accomplished. Table 8 shows the results related to ship resistance for the draught T=5.25 m (design draught). The resistance obtained from CFD analysis, the appendage resistance, the total resistance and the effective power are shown. The appendage resistance was estimated according to the well-known Holtrop method [4, 5, 6]. As the appendages, the ferry has fin stabilisers, bilge keels and the opening of the bow thruster tunnel.

Table 8. shows the numerically obtained resistance, the appendage resistance and the effective power for the corresponding Froude numbers at design draught of 5.25 m. The numerical values are graphically shown in the Figure 35.

Table 8. Resistance and effective power, T=5,25 m

V_{KN}, knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0	15.5	16.0	18.0
V, m/s	1.029	2.058	3.087	4.116	5.144	6.173	7.202	7.974	8.231	9.260
Fr	0.033	0.066	0.099	0.132	0.164	0.197	0.230	0.224	0.231	0.260
R_T(CFD), kN	6.662	19.807	36.212	62.612	104.802	165.451	246.291	324.334	355.175	492.910
R_{APP}, kN	0.361	1.319	2.824	4.851	7.384	10.412	13.925	16.874	17.916	22.379
R_T, kN	7.023	21.126	39.036	67.463	112.186	175.863	260.216	341.208	373.091	515.289
P_E, kW	7.2	43.5	120.5	277.6	577.1	1085.7	1874.1	2720.8	3071.0	4771.6

The estimation of propulsion characteristics was made for the following conditions:

Engine:

- *MCR*: 2 x 4000 kW
- Rate of revolution: N_M = 750.0 min^{-1}

Propeller:

- Z = 4
- D = 3.80 m (maximum that can still be fitted)

- P = 4.80 m
- A_E/A_O = 0.55
- Rate of revolution N_P = 150.0 min^{-1}

The design point of one propeller is defined by the following parameters:

- P_D = 3920.0 kW (100% of one propeller)
- N_P = 150.0 min^{-1} (100% of one propeller)
- V_{KN}= 18.0 knots (taken as maximum speed)

The obtained results are shown in Table 9.

Table 9. Propulsion characteristics

V_{KN}, knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0	15.5	16.0	18.0
V, m/s	1.029	2.058	3.087	4.116	5.144	6.173	7.202	7.974	8.231	9.260
Fr	0.033	0.066	0.099	0.132	0.164	0.197	0.230	0.224	0.231	0.260
w	0.061	0.061	0.061	0.061	0.061	0.061	0.061	0.061	0.061	0.061
t	0.168	0.168	0.168	0.168	0.168	0.168	0.168	0.168	0.168	0.168
V_A, m/s	0.966	1.932	2.898	3.865	4.831	5.797	6.763	7.487	7.729	8.695
η_H	0.886	0.886	0.886	0.886	0.886	0.886	0.886	0.886	0.886	0.886
P_T, kW	8.2	49.1	136.0	313.4	651.4	1225.3	2115.2	3070.7	3465.9	5385.2
T, kN (2 props.)	8.441	25.392	46.919	81.086	134.839	211.374	312.760	410.106	448.427	619.338
T, kN (1 prop.)	4.220	12.696	23.459	40.543	67.420	105.687	156.380	205.053	224.213	309.669
N_P , min^{-1}	17.53	32.59	46.64	61.83	78.23	95.940	114.09	128.36	133.37	153.57
η_O	0.670	0.708	0.703	0.735	0.733	0.725	0.718	0.711	0.708	0.698
η_R	0.970	0.970	0.970	0.970	0.970	0.970	0.970	0.970	0.970	0.970
η_S	0.98	0.98	0.98	0.98	0.98	0.98	0.98	0.98	0.98	0.98
η_P	0.565	0.596	0.592	0.619	0.617	0.610	0.604	0.599	0.596	0.588
P_B, kW (1 engine)	6.4	36.5	101.7	224.1	467.7	889.2	1550.6	2271.9	2574.9	4059.2
P_B, kW (2 engines)	12.8	72.9	203.4	448.2	935.4	1778.4	3101.2	4543.9	5149.7	8118.5

Figure 57. shows the corresponding power-speed diagram.

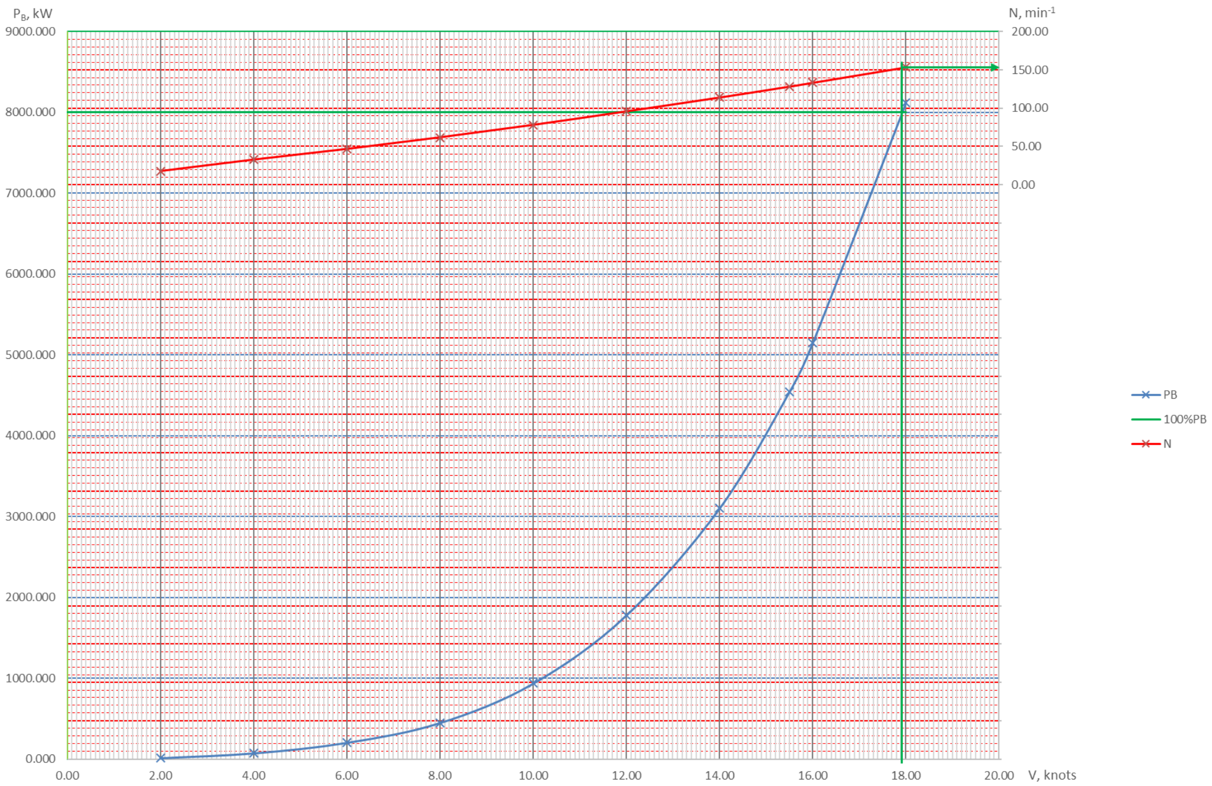

Figure 57. Power-speed diagram

The following values of the speed of the ferry and the rate of revolution of the propeller of one propulsion unit were obtained from the diagram.

- N_P = 153 min^{-1}
- V_{KN} = 17.92 kn

Based on the obtained results, it can be concluded that with the envisaged main engines, the ferry would meet all speed requirements.

7. CONCLUSION

One of the main goals of the METRO project is the development of new hybrid double-ended ferry, which may be suitable for the transportation of passengers and vehicles between ports in Italy and Croatia in the Northern Adriatic.

In this report the results of the CFD analysis of the Ro-Pax ferry hull form are presented. The CFD computations were done with FINE™/Marine, NUMECA's Flow INtegrated Environment for marine applications. Three series of simulations were made: one for a scale model of the ferry and the other two for a full size ferry for two different draughts (design and maximum). The simulations were done for the hulls in upright conditions, for a series of different speeds.

This obtained results for the full-scale Ro-Pax ferry show that the designed hull form with bulbous bow, transom stern, twin skegs and is well-chosen and that it would ensure both good seaworthiness together with the maneuverability and the favorable resistance. Furthermore, for one case of the propeller design point for which it was assumed that the maximum continuous engine power is delivered to the propeller at the maximum propeller speed, a speed of almost 18 knots was obtained. On the basis of the obtained results it can be confirmed that the developed hybrid Ro-Pax ferry with the selected main engines, would meet all ferry speed requirements.

NOMENCLATURE

A_E/A_O- - expanded blade area ratio

B - breadth (ferry), m

B_m - breadth (scale model), m

D - propeller diameter, m

D - depth (ferry), m

Fr - Froude number (ferry)

Fr_m - Froude number (scale model)

F_X - resistance (ferry), kN

F_{Xm} - resistance (scale model), kN

L_{OA} - length over all (ferry), m

L_{OAm} - length over all (scale model), m

L_{PP} - length between perpendiculars (ferry), m

L_{PPm} - length between perpendiculars (scale model), m

MCR - maximum continuos rating, kW

N_M - engine rate of revolution, min^{-1}

N_P - propeller rate of revolution, min^{-1}

P - propeller pitch, m

P_B - engine power, kW

P_D - delivered power, kW

T - thrust (ferry), kN

t - thrust deduction factor

T - draught (ferry), m

T_m	-	draught (scale model), m
V	-	speed (ferry), m/s
V_A	-	speed of advance of the propeller, m/s
V_{KN}	-	speed (ferry), knot
V_m	-	speed (scaled model), m/s
w	-	wake fraction coefficient
X, Y, Z	-	coordinate axes
Z	-	number of propeller blades
η_H	-	hull efficiency
η_O	-	open water *propeller* efficiency
η_P	-	propulsive coefficient
η_R	-	relative rotative efficiency
η_S	-	gear and shaft efficiency
λ	-	model scale

REFERENCES

[1] …, FINE™/Marine, https://www.numeca.com/product/fine-marine

[2] …, OMNIS™/Hexpress, https://www.numeca.com/product/omnis-hexpress

[3] …, CFView™, https://www.environmental-expert.com/software/cfview-computational-flow-visualisation-software-235505

[4] Holtrop, J., A statistical analysis of performance test results, approximate power prediction method, Netherlands Ship Model Basin, NSMB, Wageningen, Publication No. 540, Published in: International Shipbuilding Progress, ISP, Volume 24, Nr 270, 1977

[5] Holtrop, J., Mennen, G.G.J., A statistical power prediction method, Netherlands Ship Model Basin, NSMB, Wageningen, Publication No. 689, Published in: International Shipbuilding Progress, ISP, Volume 29, Nr 335, 1982

[6] Holtrop, J., A statistical re-analysis of resistance and propulsion data, Netherlands Ship Model Basin, NSMB, Wageningen, Publication No. 769, Published in: International Shipbuilding Progress, ISP, Volume 31, Nr 363, 1984