

## WP 3 / Activity 3.3

# A Computational Fluid Dynamics Analysis of a Double-Ended Ferry Hull Form

Prepared by:

University of Rijeka - Faculty of Engineering  
Tehnomont Shipyard Pula Ltd

## Summary

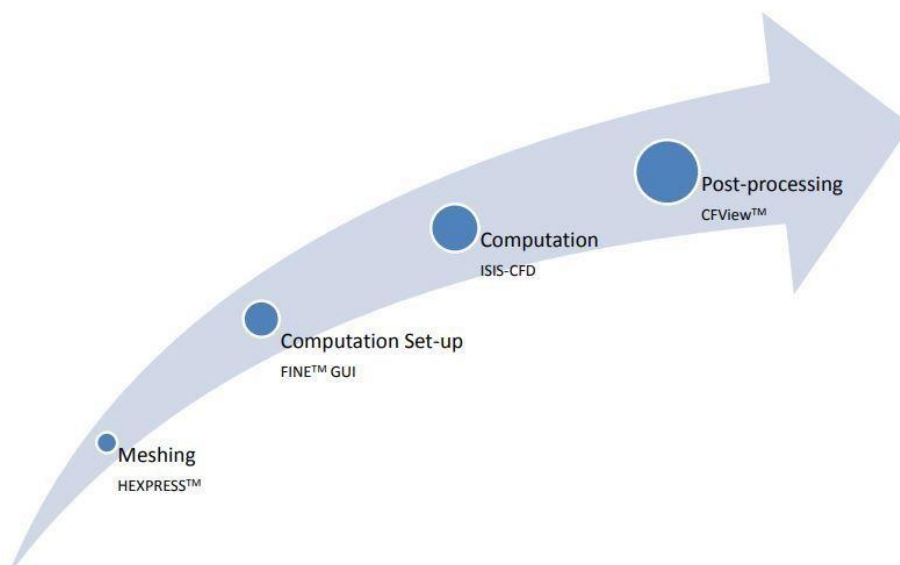
<b>1. INTRODUCTION .....</b>	<b>2</b>
<b>2. DOUBLE ENDED FERRY .....</b>	<b>4</b>
<b>3. PREPARATION OF A 3D MODEL FOR CFD SIMULATIONS .....</b>	<b>7</b>
<b>4. SCALE MODEL OF THE FERRY .....</b>	<b>11</b>
4.1. Mesh generation .....	12
4.2. Computations .....	17
4.3. Results .....	18
<b>5. FULL SIZE MODEL OF THE FERRY .....</b>	<b>29</b>
5.1. Mesh generation .....	29
5.2. Computations .....	38
5.3. Results .....	38
5.3.1. Design draught $T = 2.3$ m .....	38
5.3.2. Maximum draught $T = 2.5$ m .....	48
<b>6. THE POWER-SPEED DIAGRAM .....</b>	<b>60</b>
<b>7. CONCLUSION .....</b>	<b>63</b>
<b>NOMENCLATURE .....</b>	<b>64</b>
<b>REFERENCES .....</b>	<b>66</b>

## 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 double-ended ferry. The numerical simulations (CAD import – meshing – computations – visualization) were performed with FINE™/Marine, NUMECA’s Flow InItegrated 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 maximum). The simulations were done for the hulls in upright conditions, for a series of different speeds. The workflow of the procedure is presented in 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. DOUBLE-ENDED FERRY

The main particulars of the double-ended ferry hull are shown in the Table 1.

*Table 1. Double-ended ferry hull main particulars*

$L_{OA}$	101.9 m
$L_{PP}$	92.7 m
$B$	20,0 m
$T_{design}$	2.3 m
$T_{scantling}$	2.5 m
$D_{to\ main\ deck}$	3.8 m
$D_{to\ passenger\ deck}$	9.4 m

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

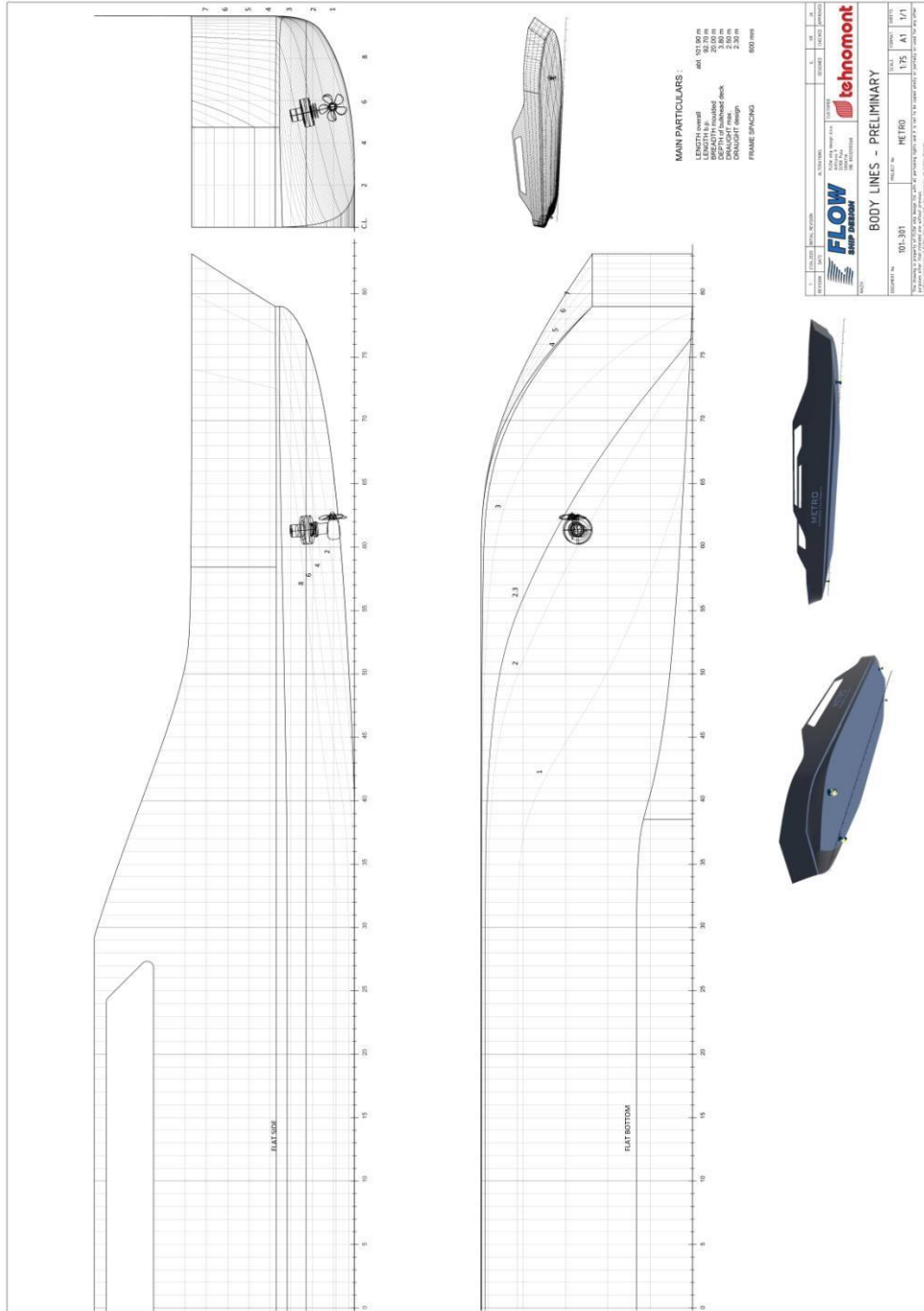


Figure 2. Double-ended ferry hull lines

The estimated double-ended ferry speed is 12 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
$V$ , m/s	1.0289	2.0578	3.0867	4.1156	5.1444	6.1733	7.2022
$Fr$	0.0329	0.0658	0.0987	0.1316	0.1644	0.1973	0.2302

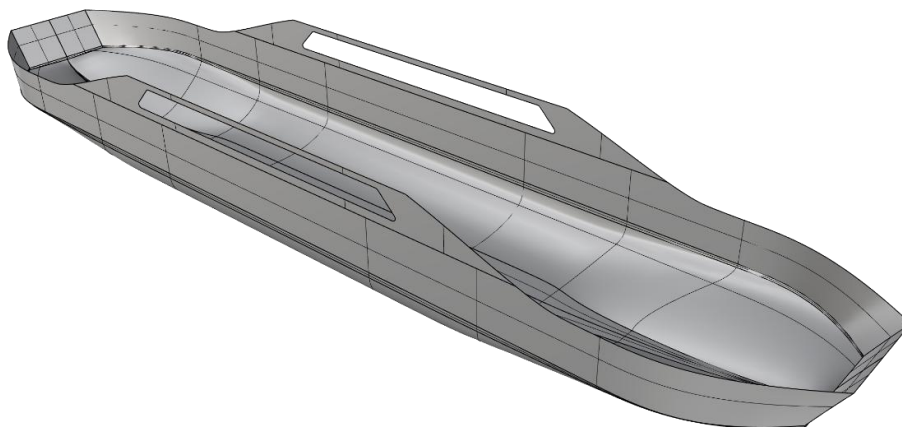
The speeds range from 2.0 knots (1.0289 m/s) to 14.0 knots (7.2022 m/s), and these speeds correspond to Froude numbers ranging from 0.0329 to 0.2302.

### 3. PREPARATION OF A 3D MODEL FOR CFD SIMULATIONS

The basic model of the double-ended 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. Therefore, additional modifications were made to the model in order to close the surfaces of the hull, particularly at the main deck level.

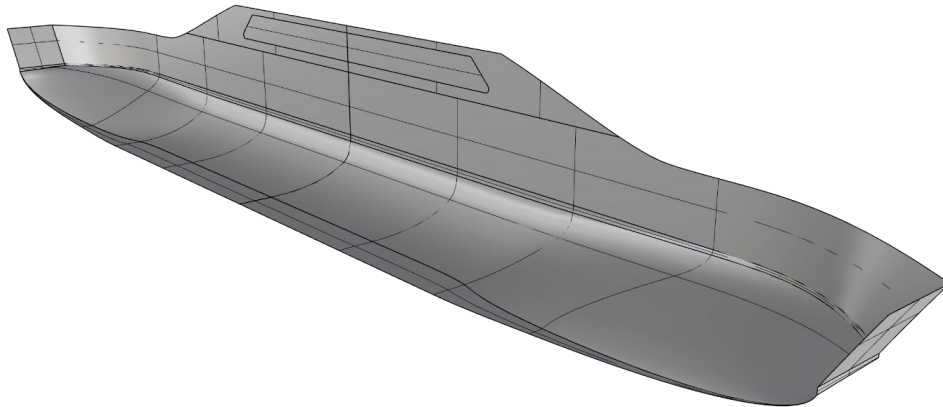
Figure 3. shows the initial 3D model, while Figures 4 to 8 briefly show the steps of processing the initial model until obtaining the final 3D model ready to be imported into the CFD software.



*Figure 3. Initial 3D model of the double-ended ferry*

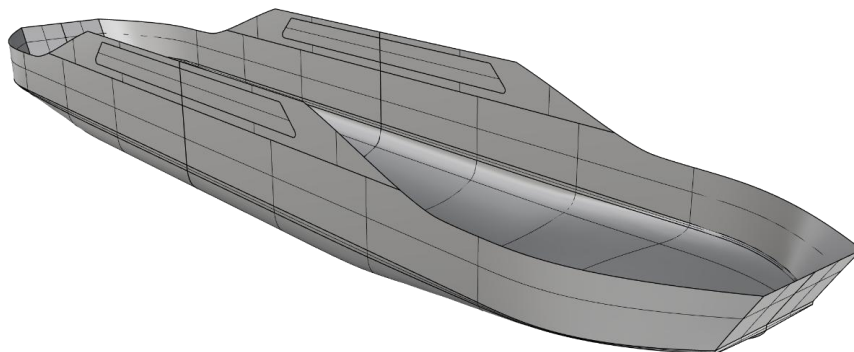


Step 1: Since the hull, i.e. the model, is considered as symmetrical about the centerline, the model is longitudinally divided in half in order to make all corrections only on one half of the model. The closed side-openings can be noticed in Figure 4.



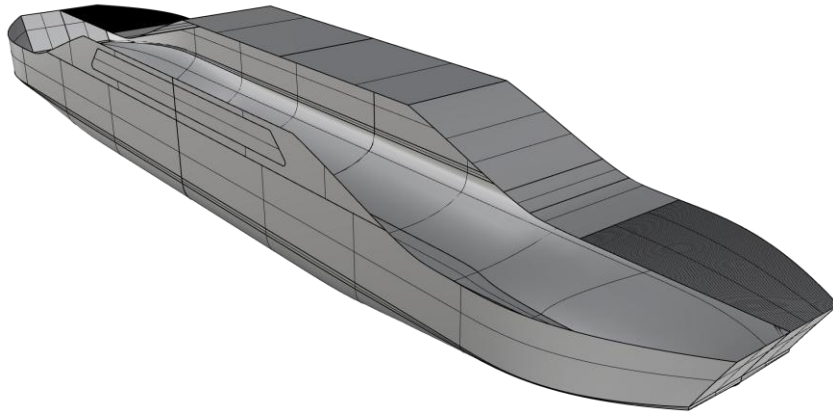
*Figure 4. Process of closing the 3D model – Step 1*

Step 2: After closing the side openings and doing other corrections, the half model body was mirrored to get the complete model, Figure 5.



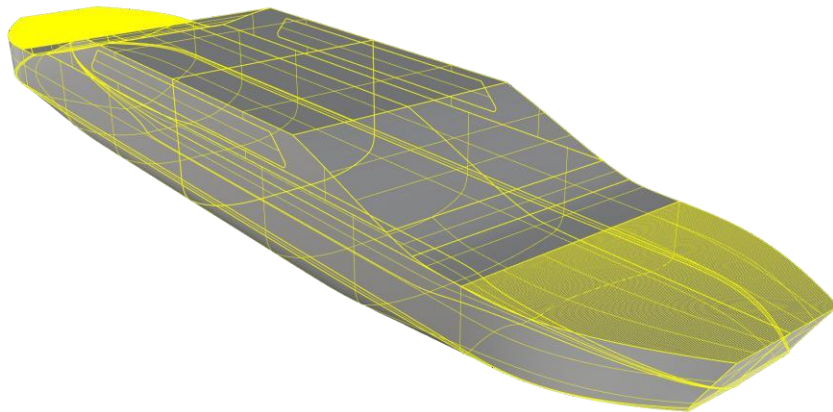
*Figure 5. Process of closing the 3D model – Step 2*

Step 3: Model needs to be fully closed so it was closed from above, Figure 6. Only one half of the cover surfaces are presented. These surfaces are subsequently mirrored.

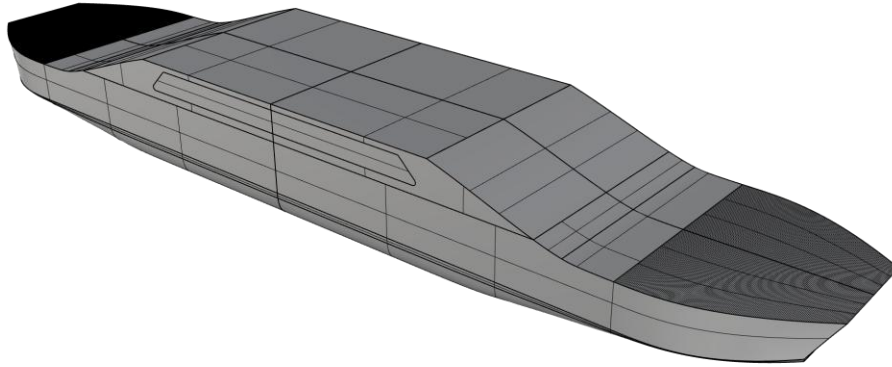


*Figure 6. Process of closing the 3D model – Step 3*

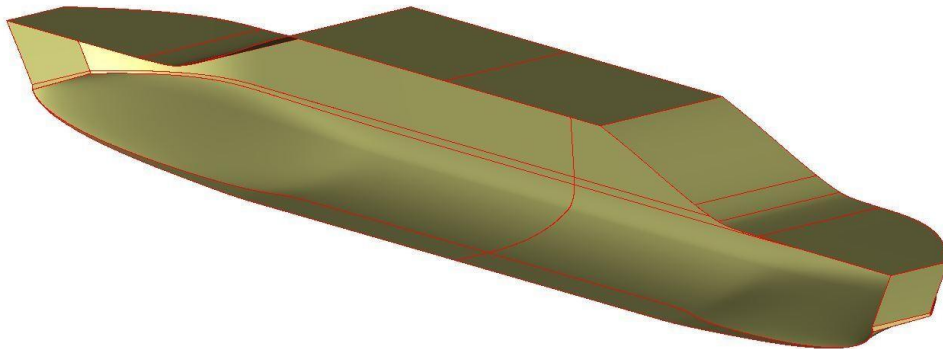
Step 4: After mirroring the cover surfaces, the last step consists of re-checking that there are no gaps between surfaces which can finally be joined in order to get the model as a single solid, Figure 7.



*Figure 7. Process of closing the 3D model – Step 4*



*Figure 8.* 3D model of the double-ended ferry as a solid, ready to be imported in the CFD software



*Figure 9.* 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 9. 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  $\lambda = 10$  was chosen and the obtained main particulars of the model are presented in Table 3.

Table 3. Double-ended ferry model hull main particulars

$L_{OAm}$	10.19 m
$L_{PPm}$	9.27 m
$B_m$	2.0 m
$T_{m,design}$	0.23 m
$T_{m,scantling}$	0.25 m
$D_{m,to\ main\ deck}$	0.38 m
$D_{m,to\ passenger\ deck}$	0.94 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

$V_m, \text{ m/s}$	0.3254	0.6507	0.9761	1.3014	1.6268	1.9521	2.2775
$V_{KNm}, \text{ knot}$	0.6325	1.2649	1.8974	2.5299	3.1623	3.7947	4.4272
$Fr_m$	0.0329	0.0658	0.0987	0.1316	0.1644	0.1973	0.2302

For the model, the speeds range from 0.3254 m/s to 2.2775 m/s, and these values correspond to a variation of the Froude number from 0.0329 to 0.2302. Due to the dynamic similarity, the values of the Froude numbers for the ferry and its model are equal.

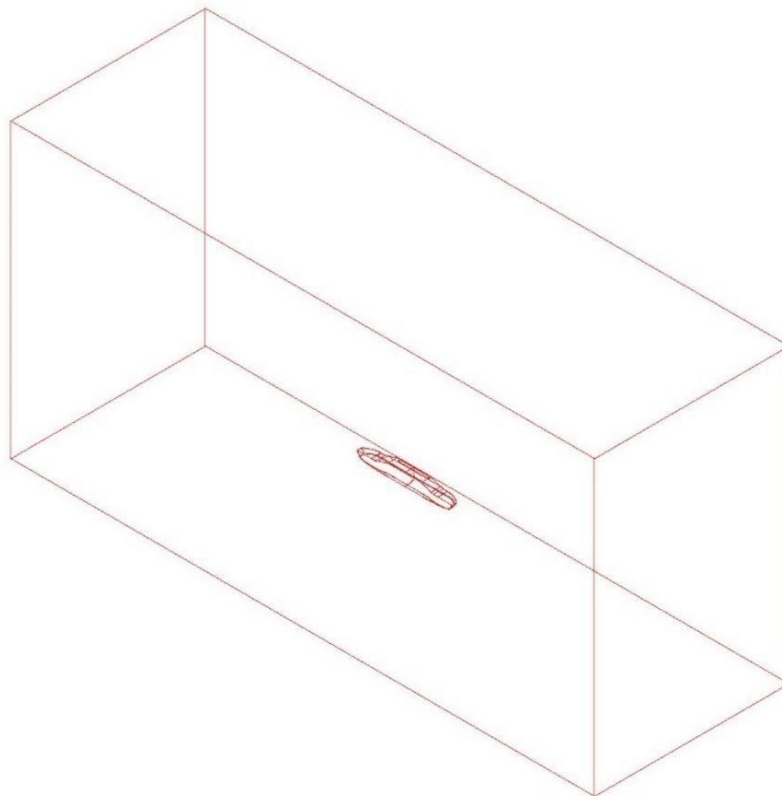
The CFD computations were done for the draught of 0.23 m which corresponds to a ferry design draught.

#### 4.1. Mesh generation

In order to form an appropriate mesh, the “Parasolid” model is loaded into HEXPRESS™. 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 10.



*Figure 10. Domain box*

The mesh generation in HEXPRESS™ is done in five-steps, Figure 11:

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.

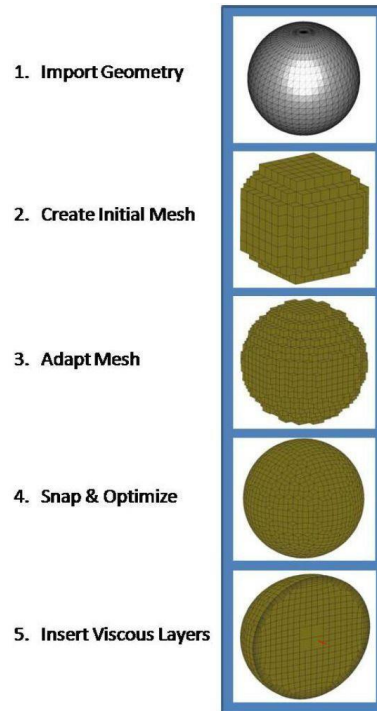
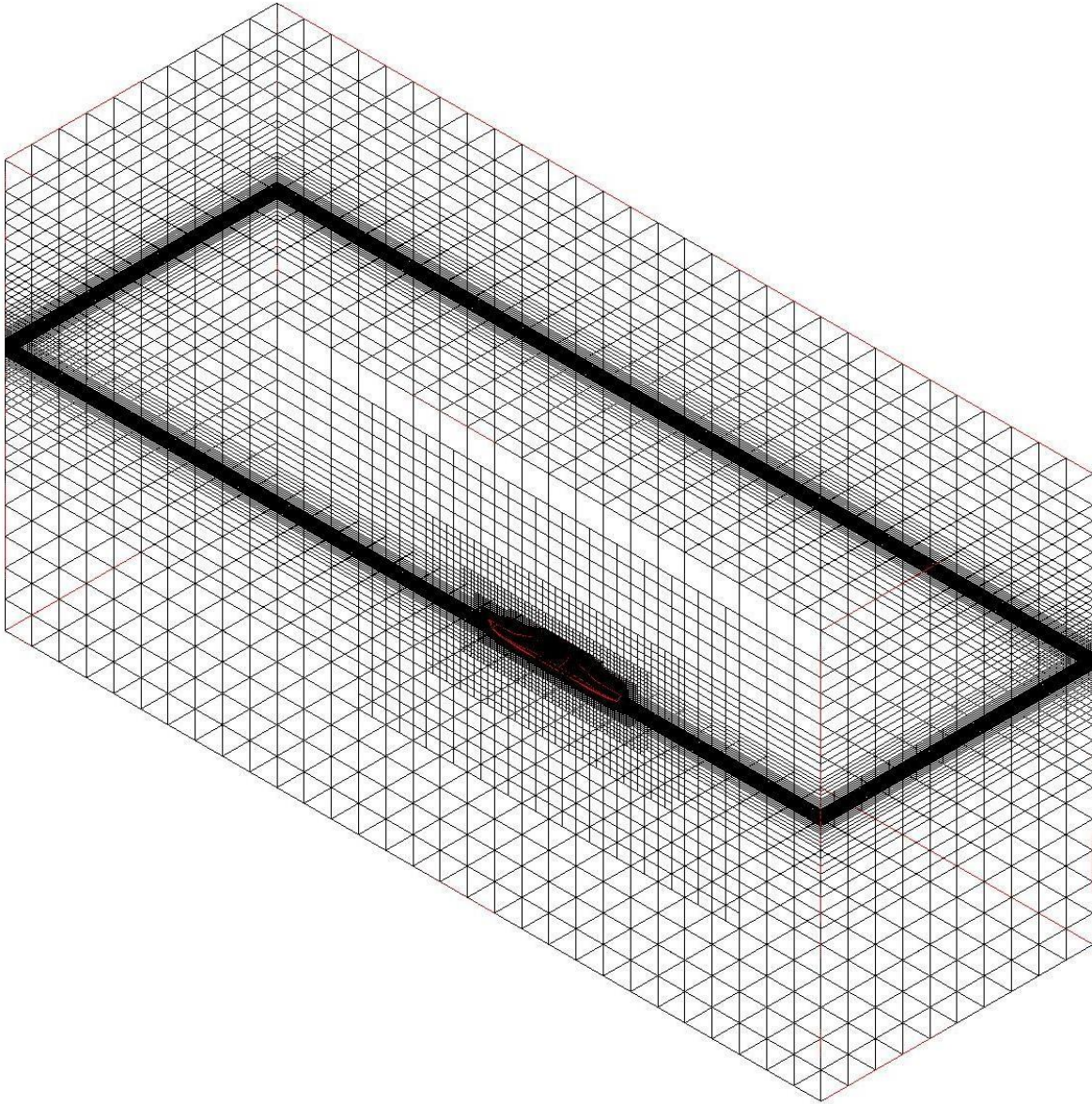


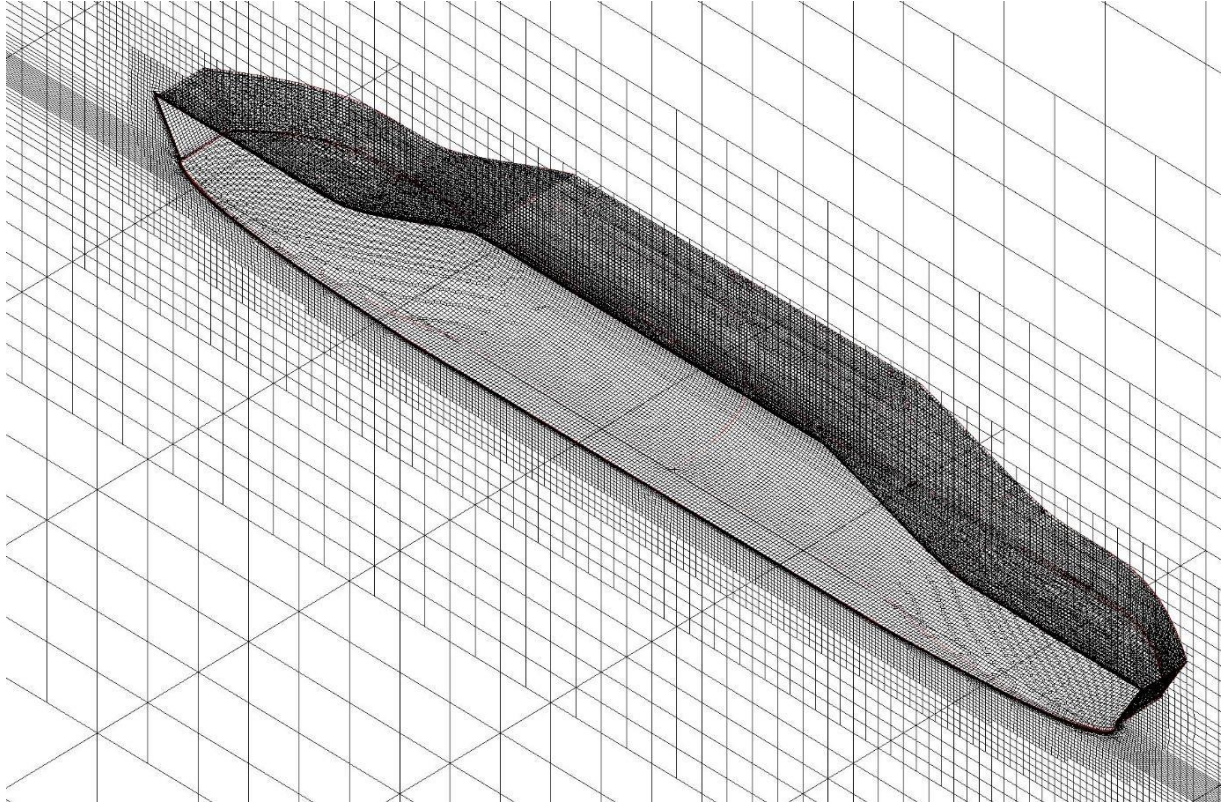
Figure 11. The process of creating a network in HEXPRESS™

Figures from 12 to 13 show some examples of the fine mesh for the scale model of the double-ended ferry. The number of used cells was 1,034,471 with 1,100,354 vertices.



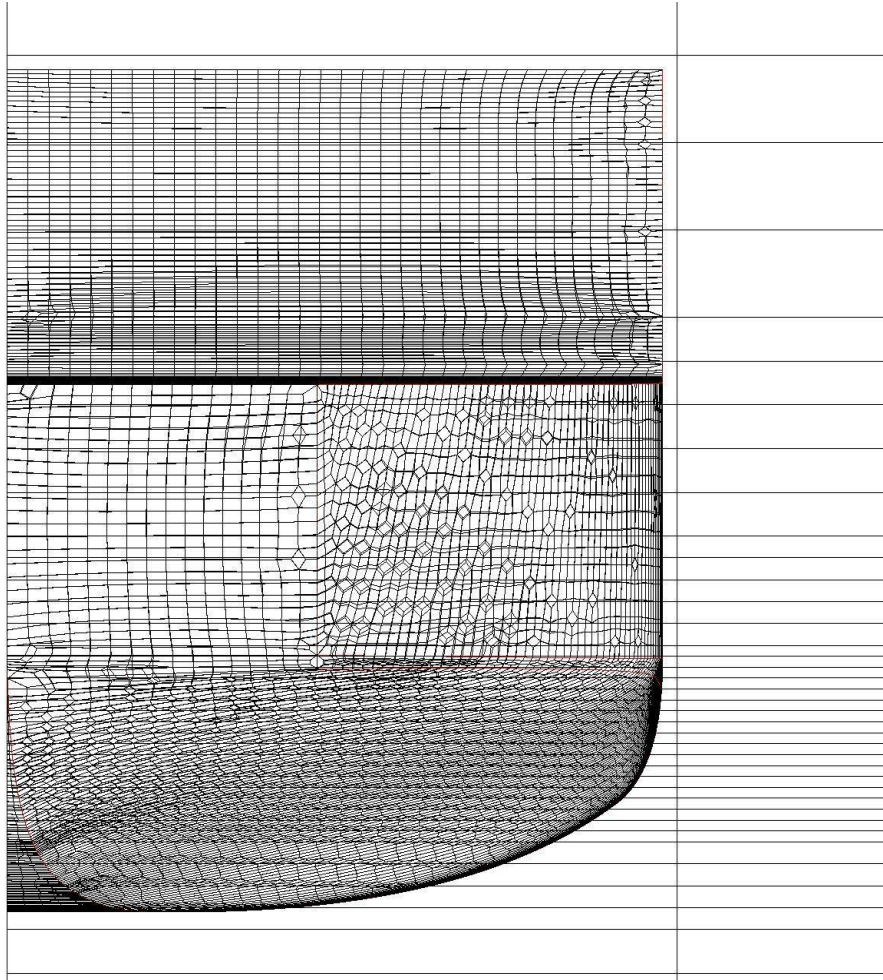


*Figure 12.* Fine mesh for the scale model

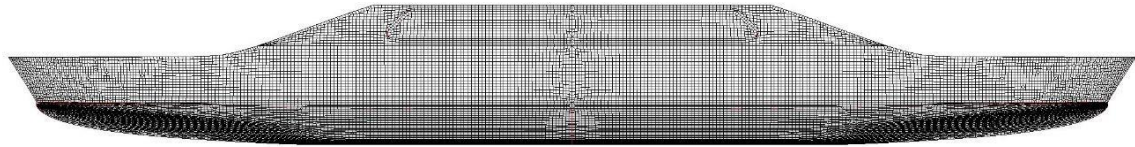


*Figure 13. Fine mesh for the scale model, zoomed view*

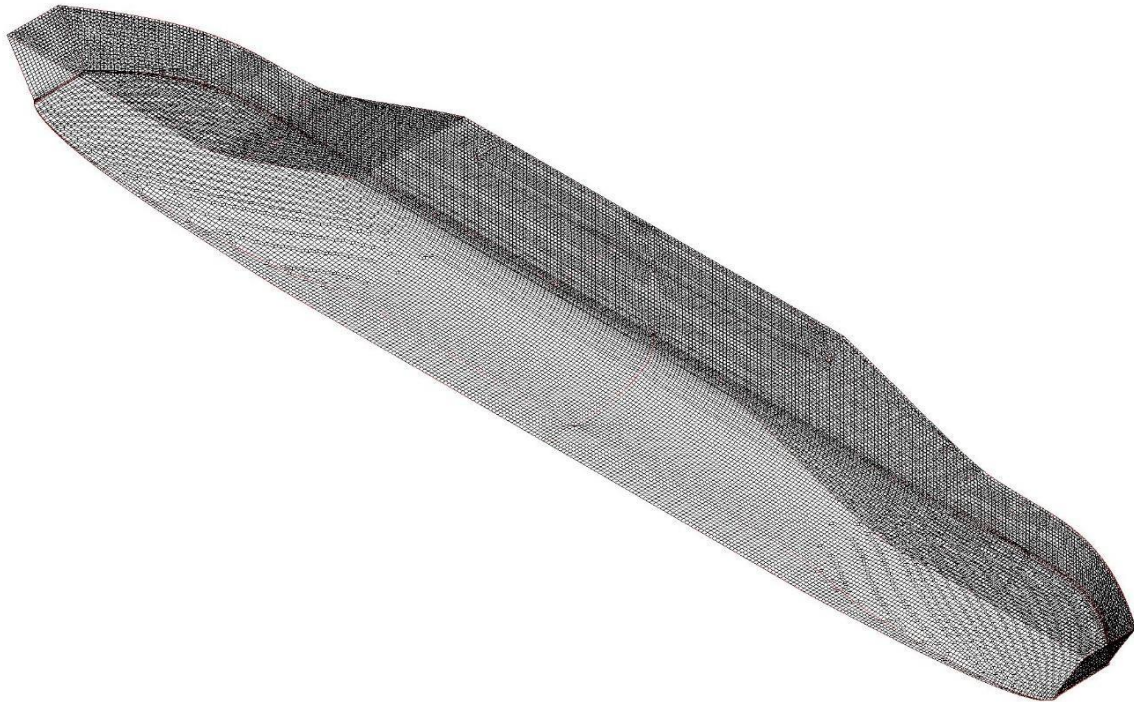




*Figure 14. Fine mesh for the scale model, X-plane view*



*Figure 15.* Fine mesh for the scale model, Y-plane view



*Figure 16.* Fine mesh for the scale model, isometric view

## 4.2. Computations

The boundary conditions shown in Figure 17 are set for the calculations.

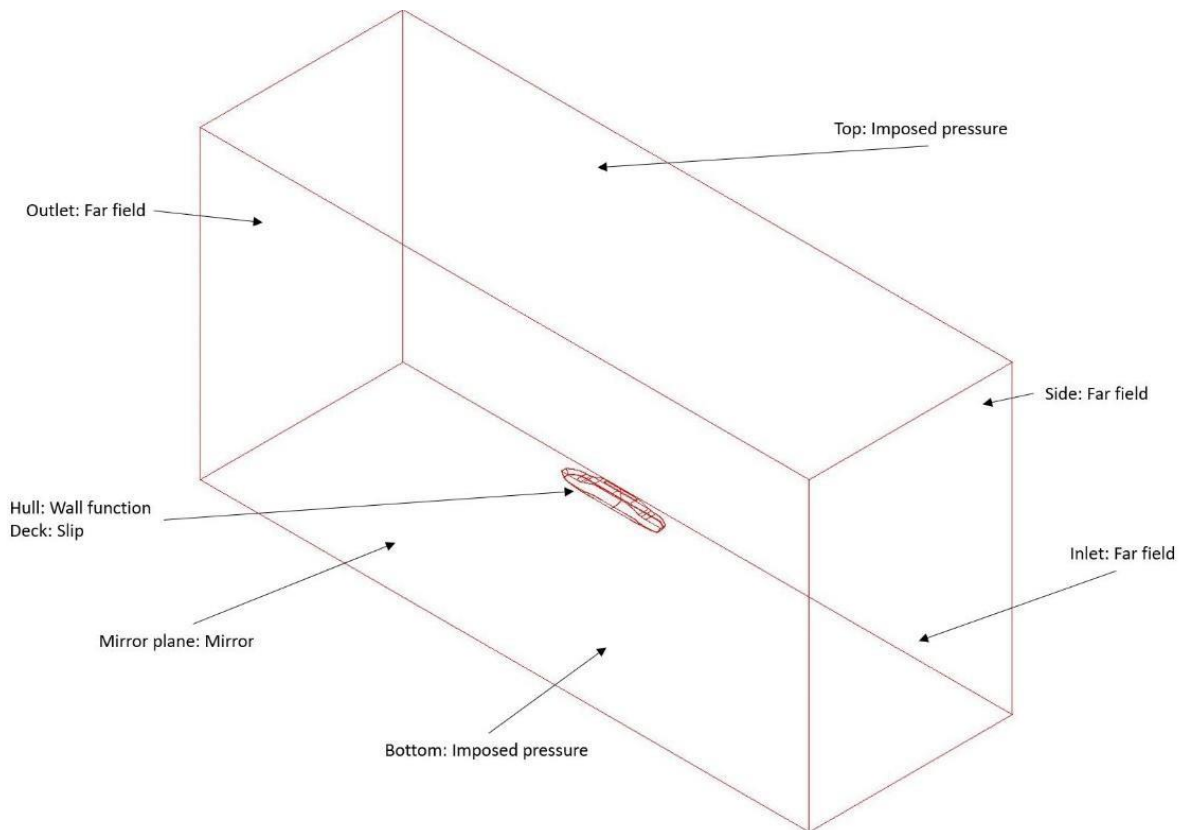


Figure 17. Boundary conditions

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

### 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 0.23 m. The numerical values are graphically shown in the Figure 18.

Table 5. Resistance vs. Froude number

$V_m$ , m/s	0.3254	0.6507	0.9761	1.3015	1.6268	1.9522	2.2775
$Fr_m$	0.0329	0.0658	0.0987	0.1316	0.1644	0.1973	0.2302
$Fx_m$ , N	5.463	13.695	26.558	45.518	71.954	108.955	160.438

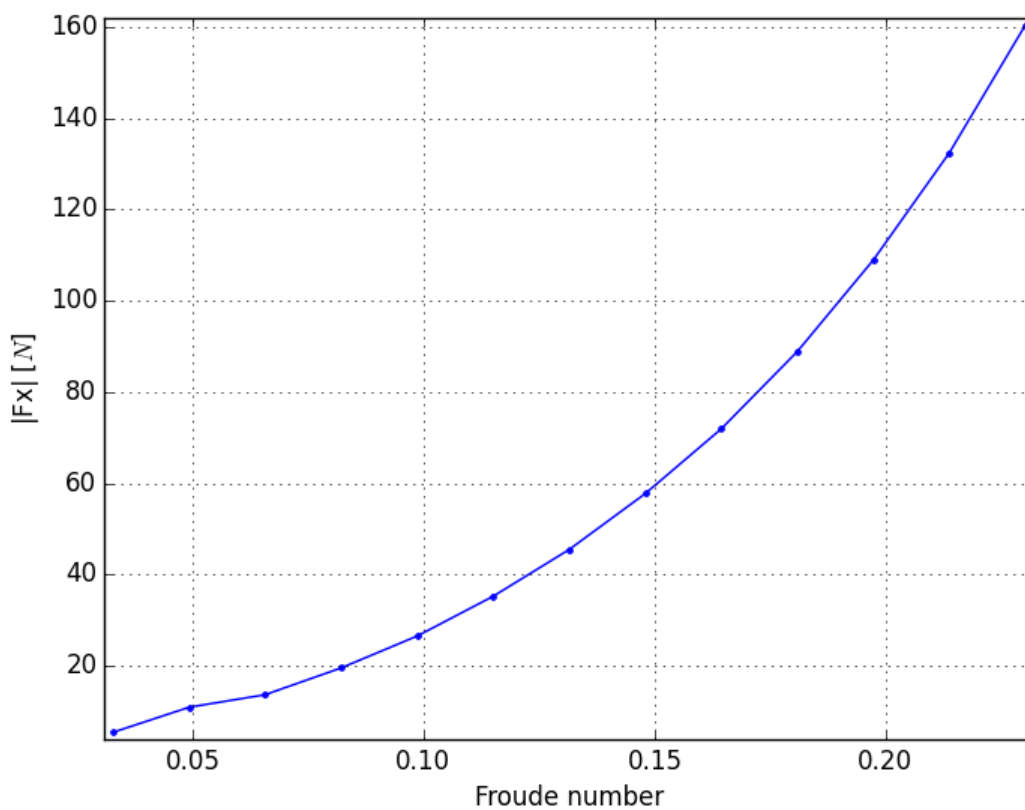
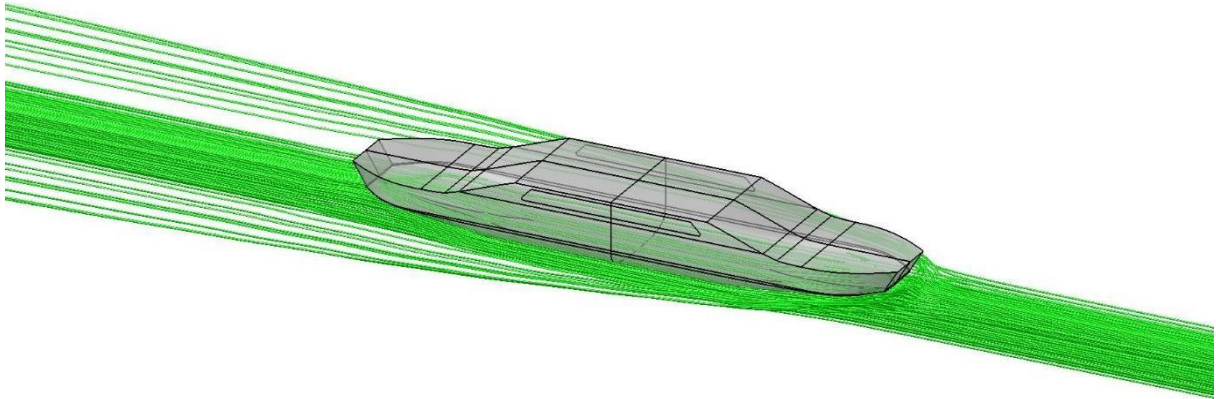


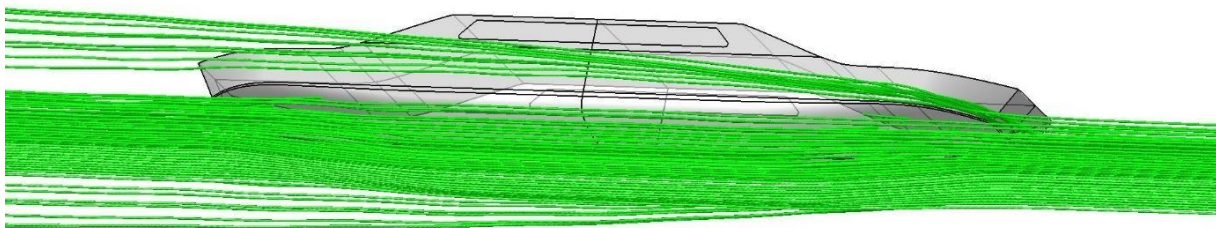
Figure 18. Resistance vs. Froude number

Figures 19. - 28. visually show some results obtained by CFD simulations. First, the results for the speed of 1.9522 m/s that corresponds to a ferry design speed and then the results for other speeds between 0.3254 m/s and 2.2775 m/s were shown.





*Figure 19.* Relative velocity streamlines,  $V_m = 1.955$  m/s



*Figure 20.* Relative velocity streamlines,  $V_m = 1.955$  m/s

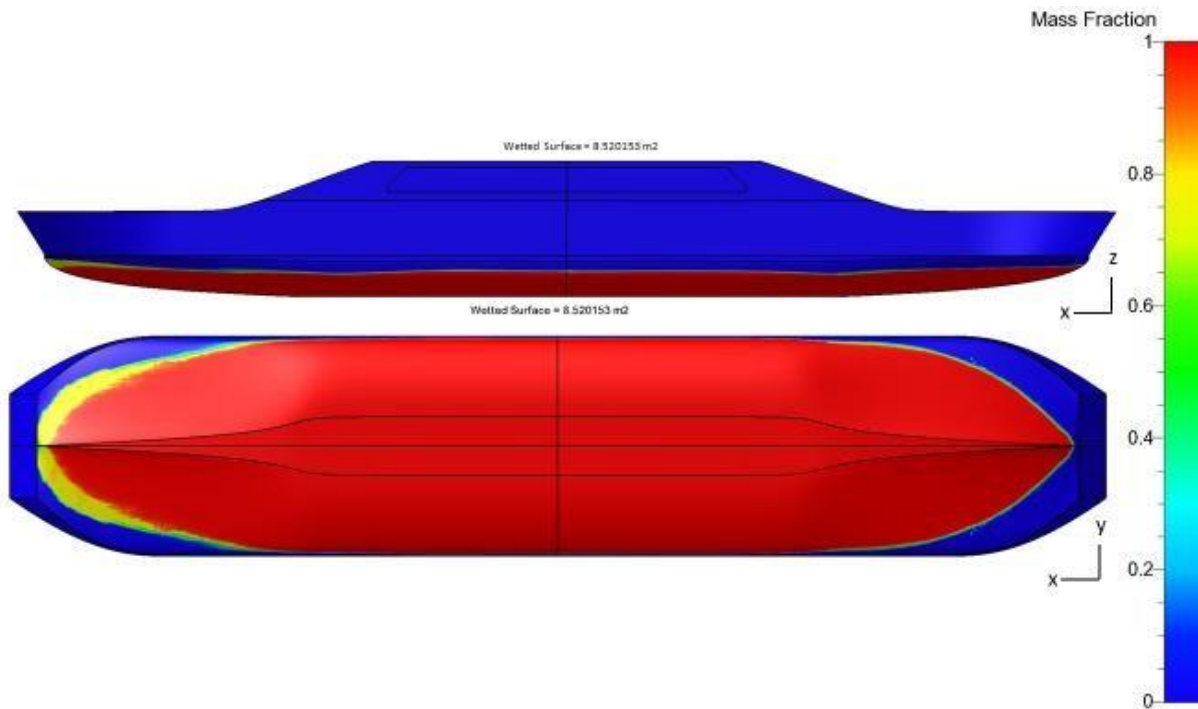


Figure 21. Wetted surface,  $V_m = 1.955$  m/s

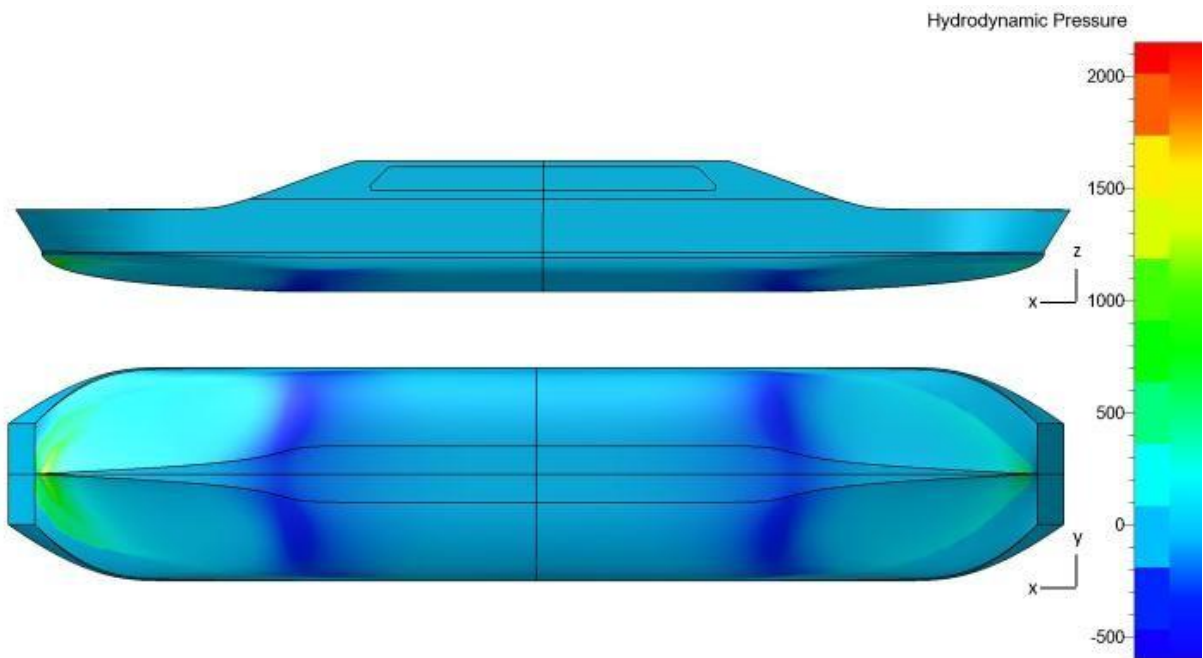


Figure 22. Hydrodynamic pressure,  $V_m = 1.955$  m/s

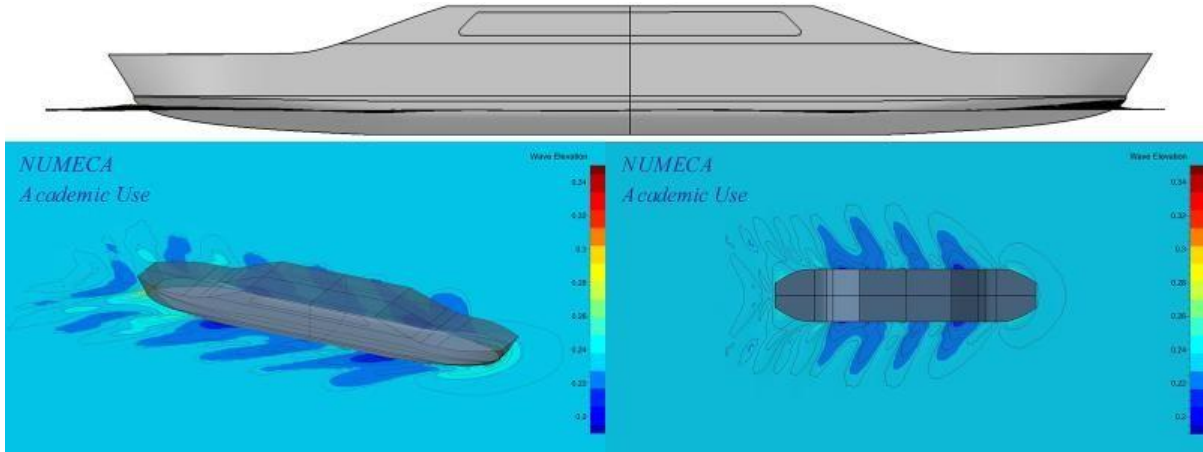


Figure 23. Wave elevations,  $V_m = 1.955$  m/s

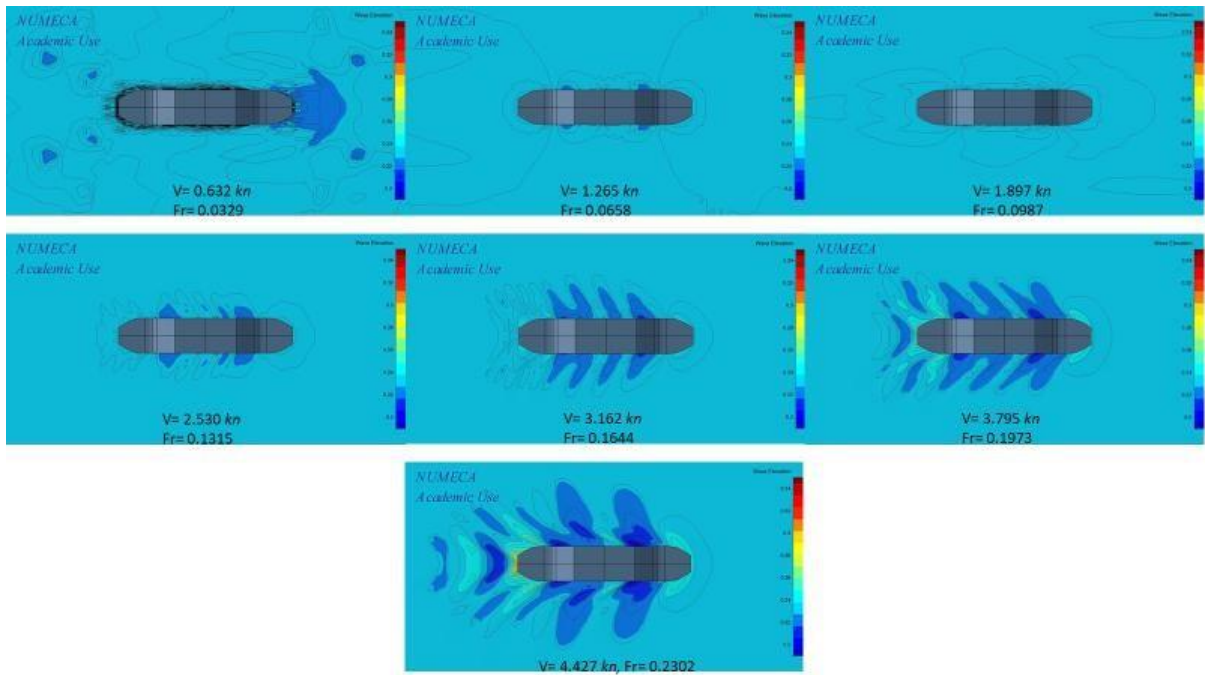


Figure 24. Wave elevations,  $V_m = 0.3254$  m/s to 2.2775 m/s, Y-plane view

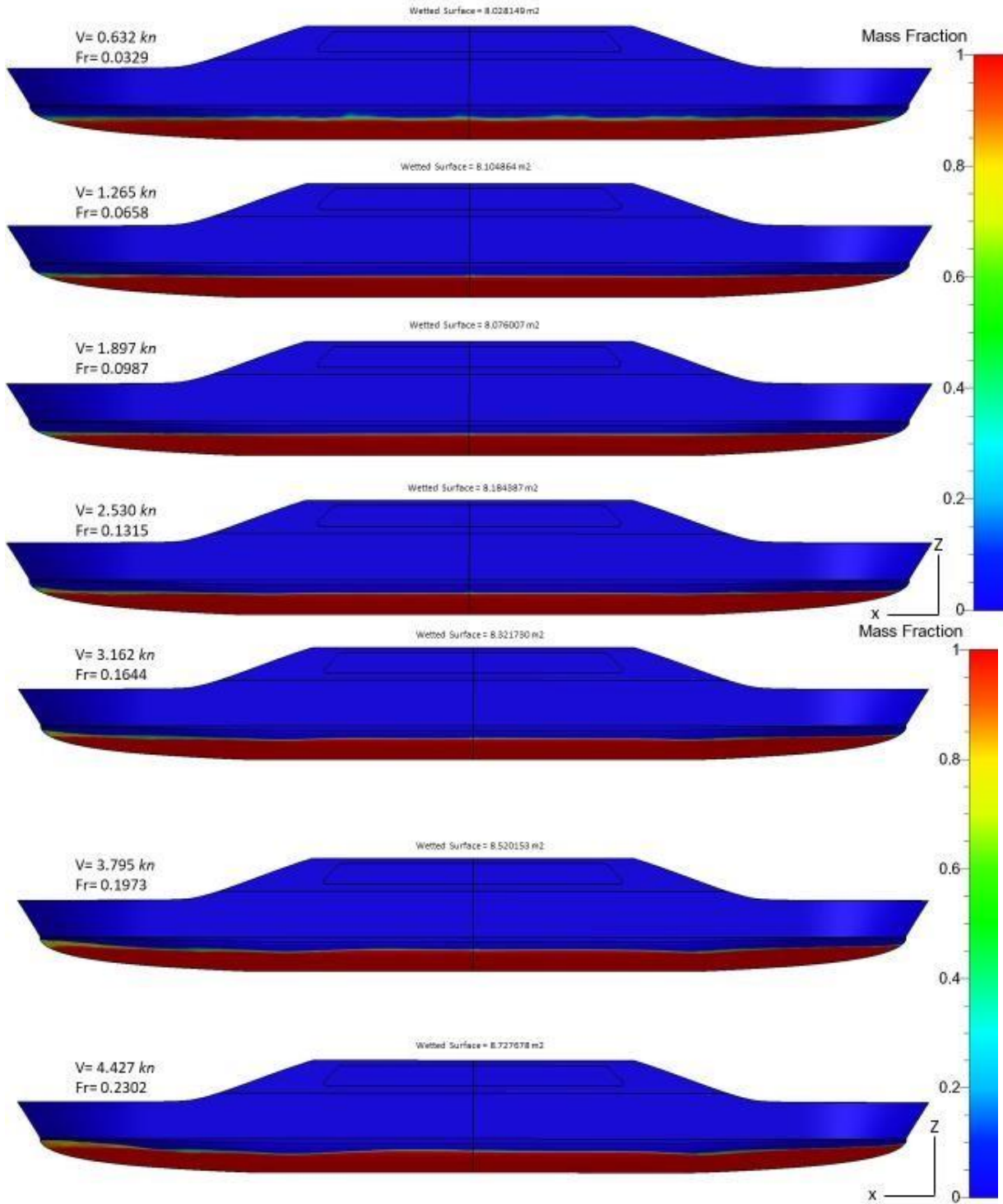
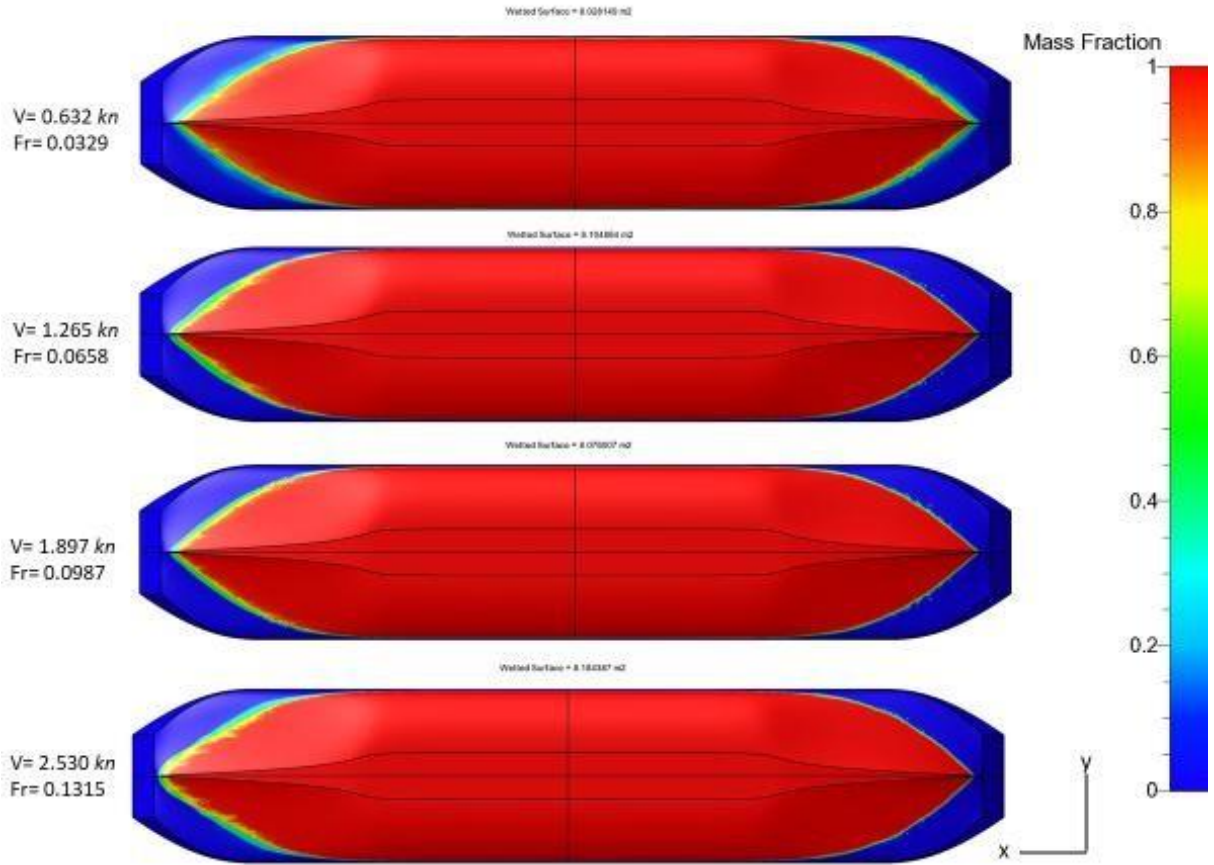


Figure 25. Wetted surface,  $V_m = 0.3254$  m/s to 2.2775 m/s, Y-plane view





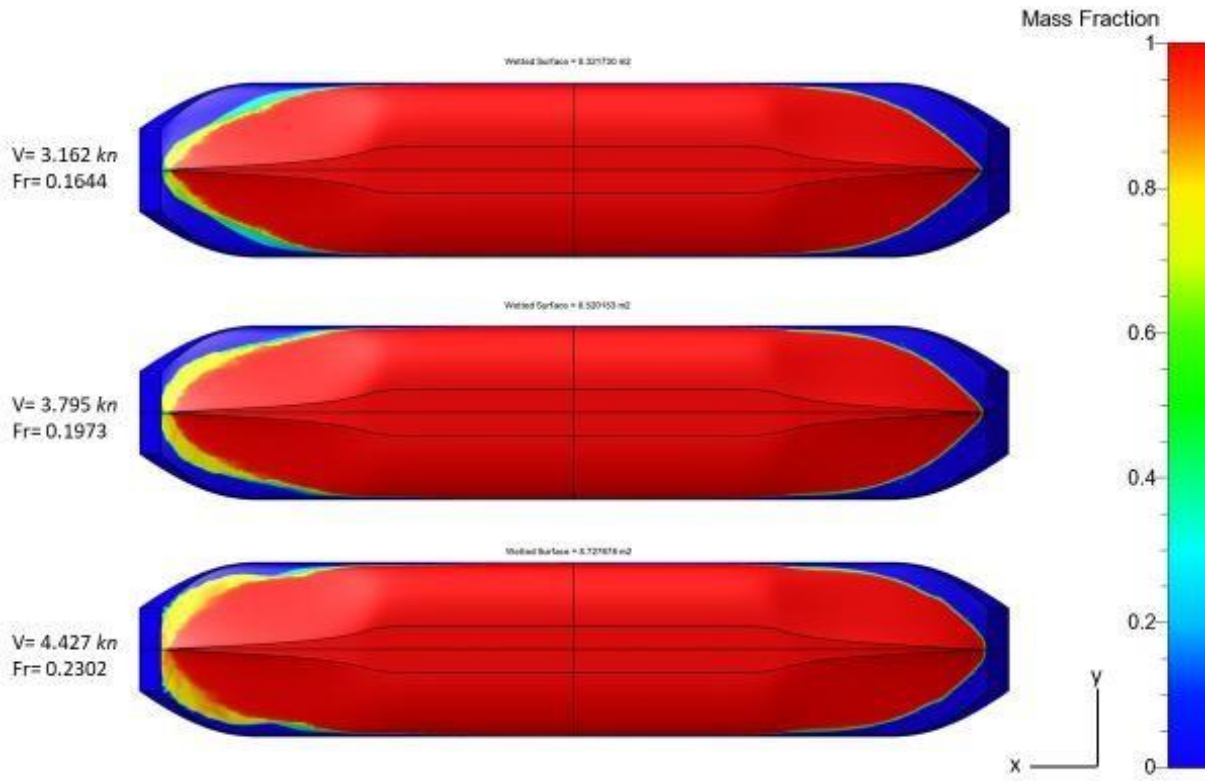


Figure 26. Wetted surface,  $V_m = 0.3254$  m/s to 2.2775 m/s, Z-plane bottom view

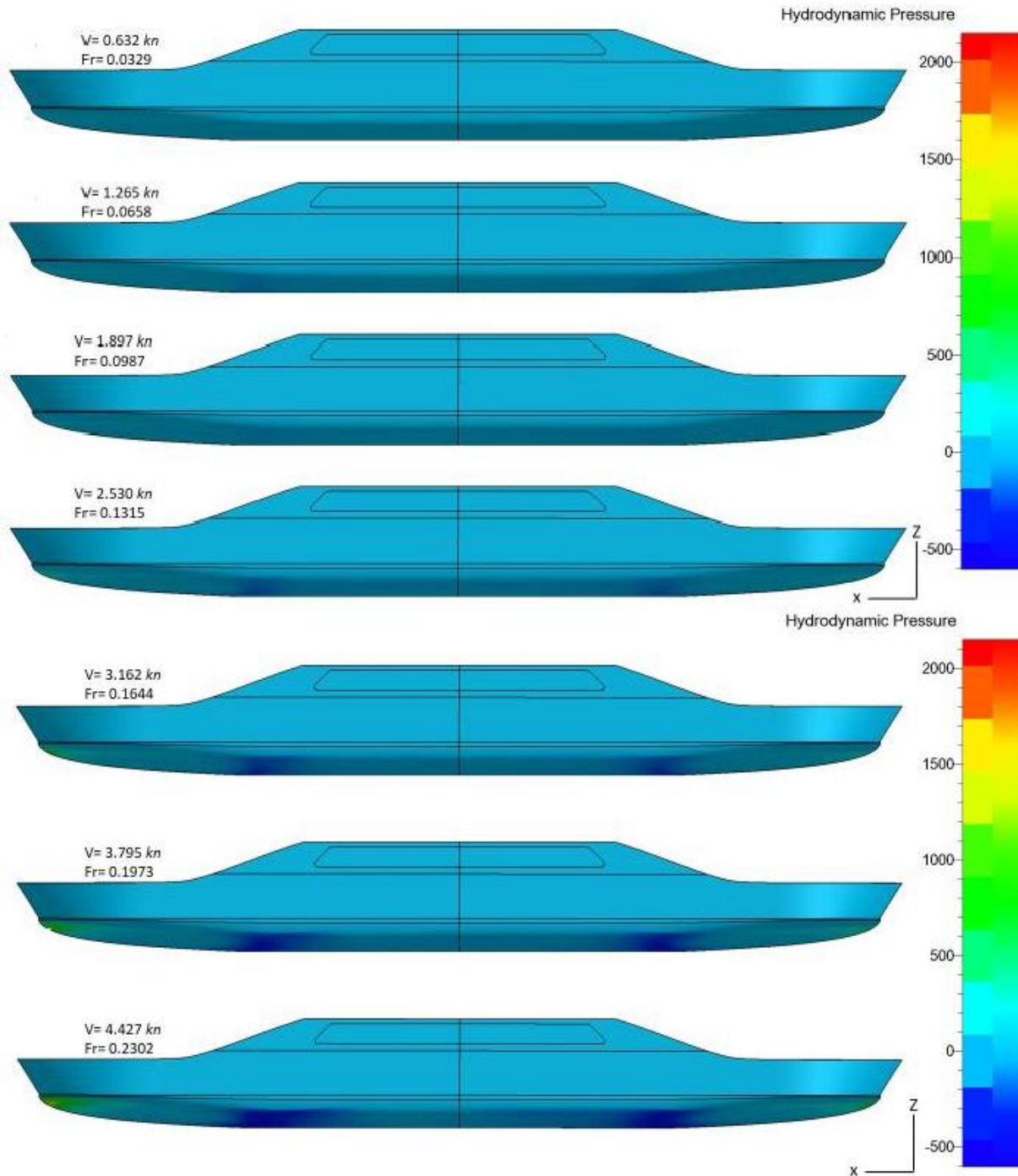
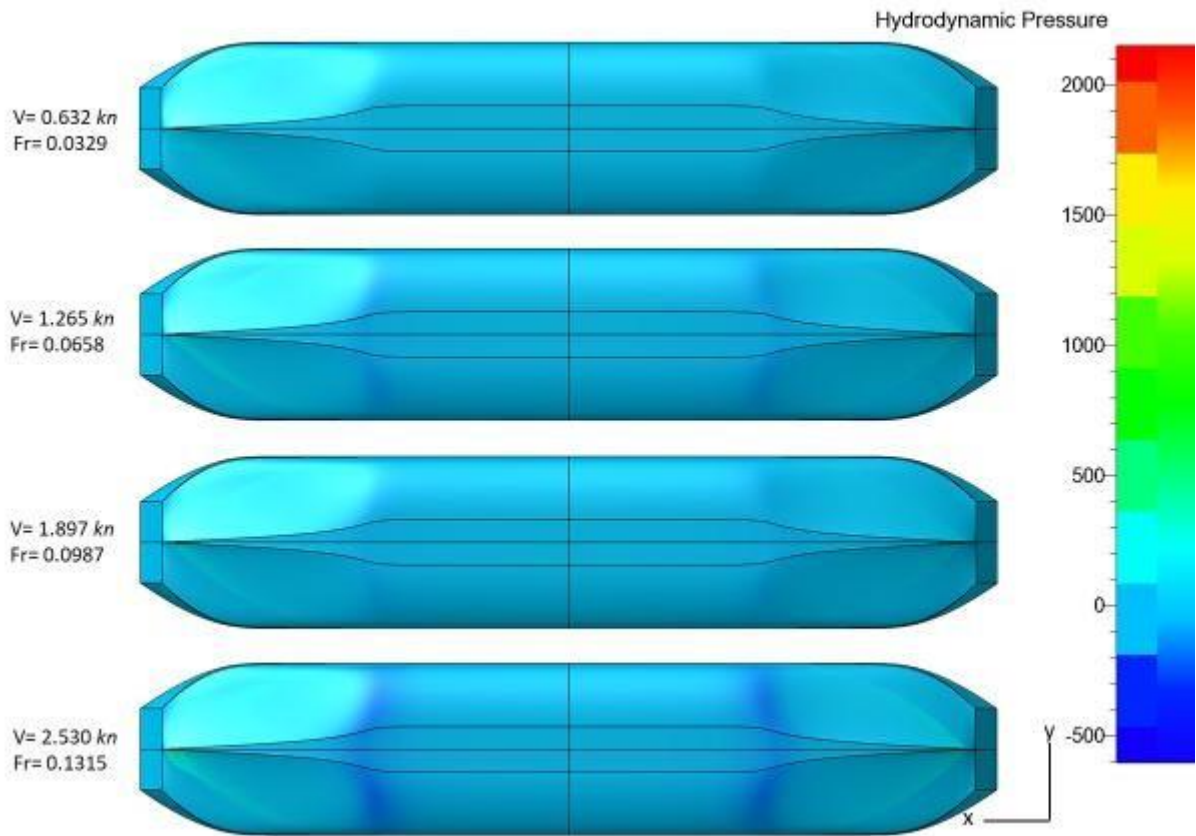


Figure 27. Hydrodynamic pressure,  $V_m = 0.3254$  m/s to 2.2775 m/s, Y-plane view



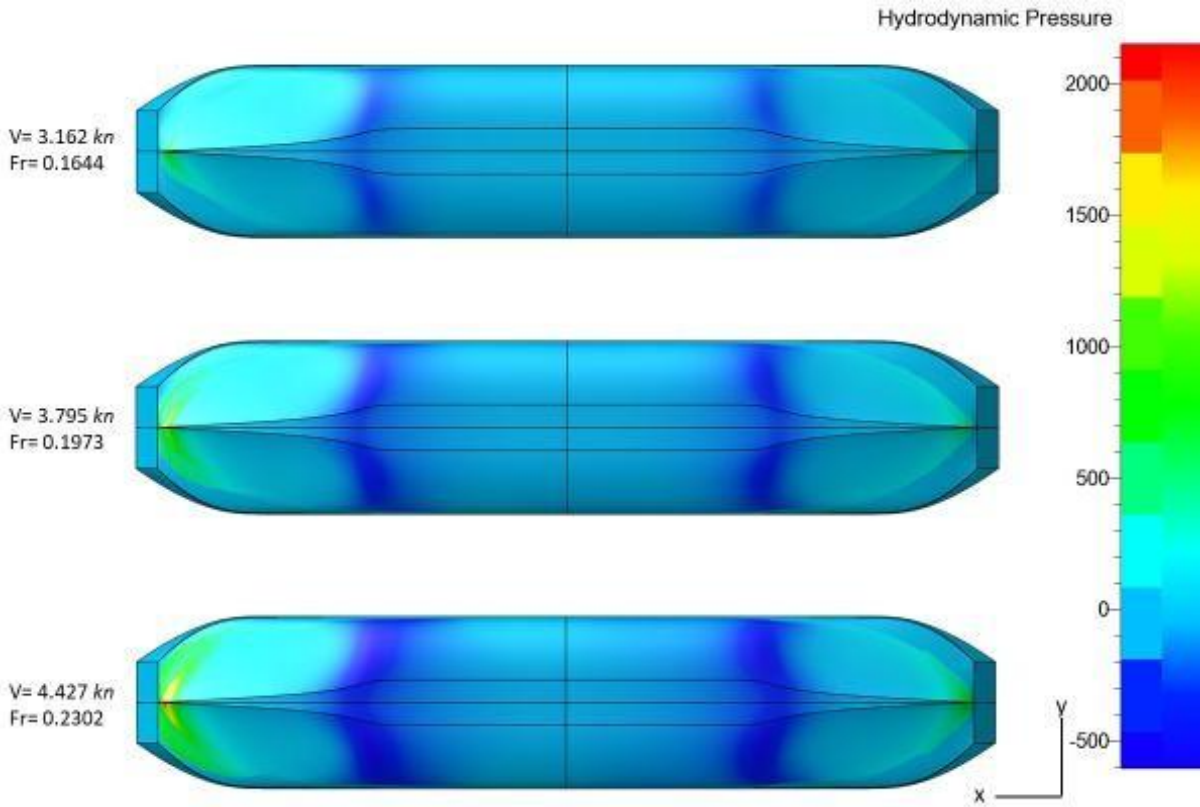


Figure 28. Hydrodynamic pressure,  $V_m = 0.3254$  m/s to 2.2775 m/s, Z-plane bottom view



## 5. FULL SIZE MODEL OF THE FERRY

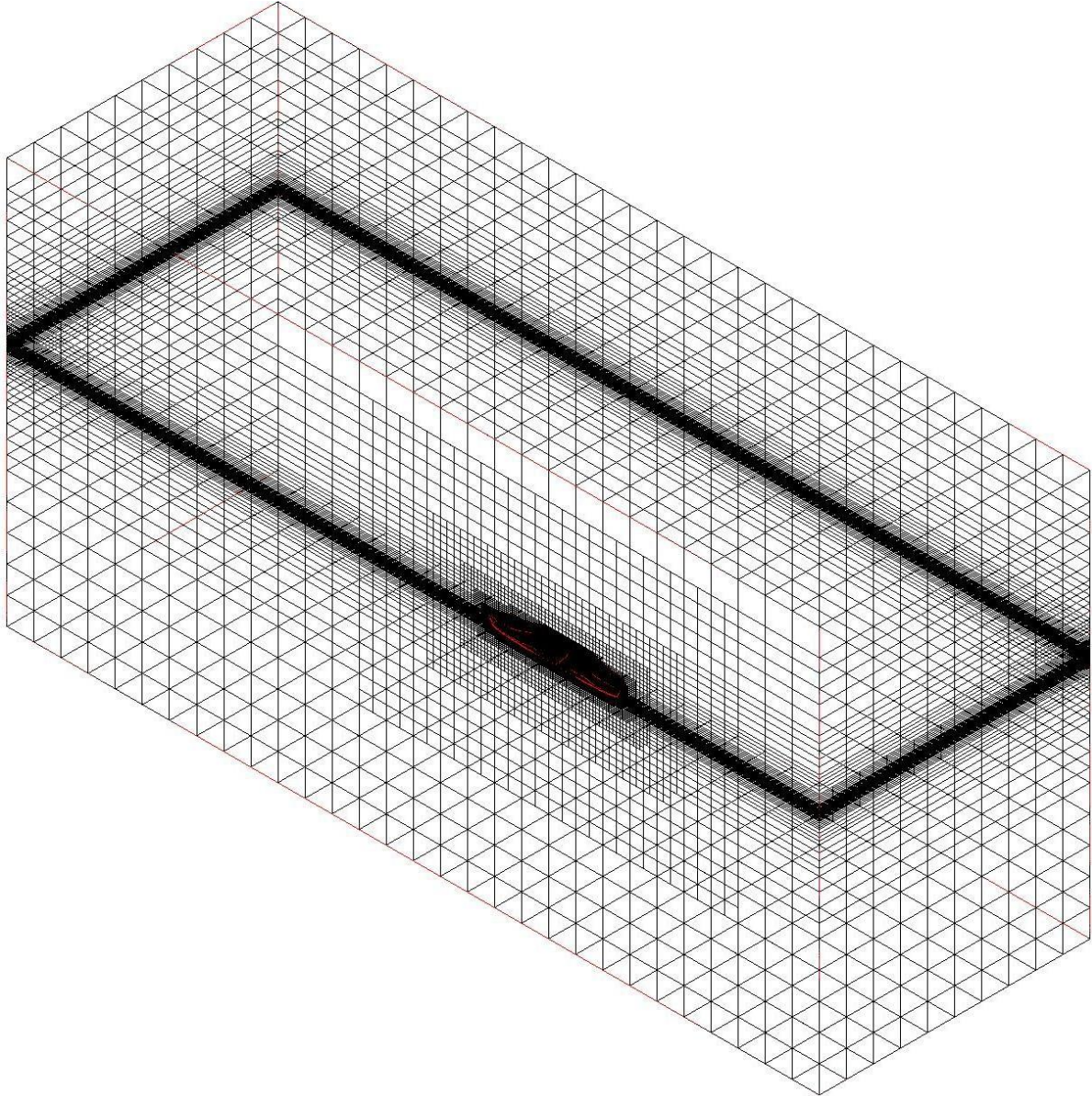
Accurate prediction of the resistance is one of the most important factors for the energy-efficient design of a ship. In general, the hydrodynamic performance of a full-scale ship could be achieved by model-scale simulation, but this method is strongly affected by the method used to extrapolate from 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 2.3 m and the other is for the maximum draught of 2.5 m.

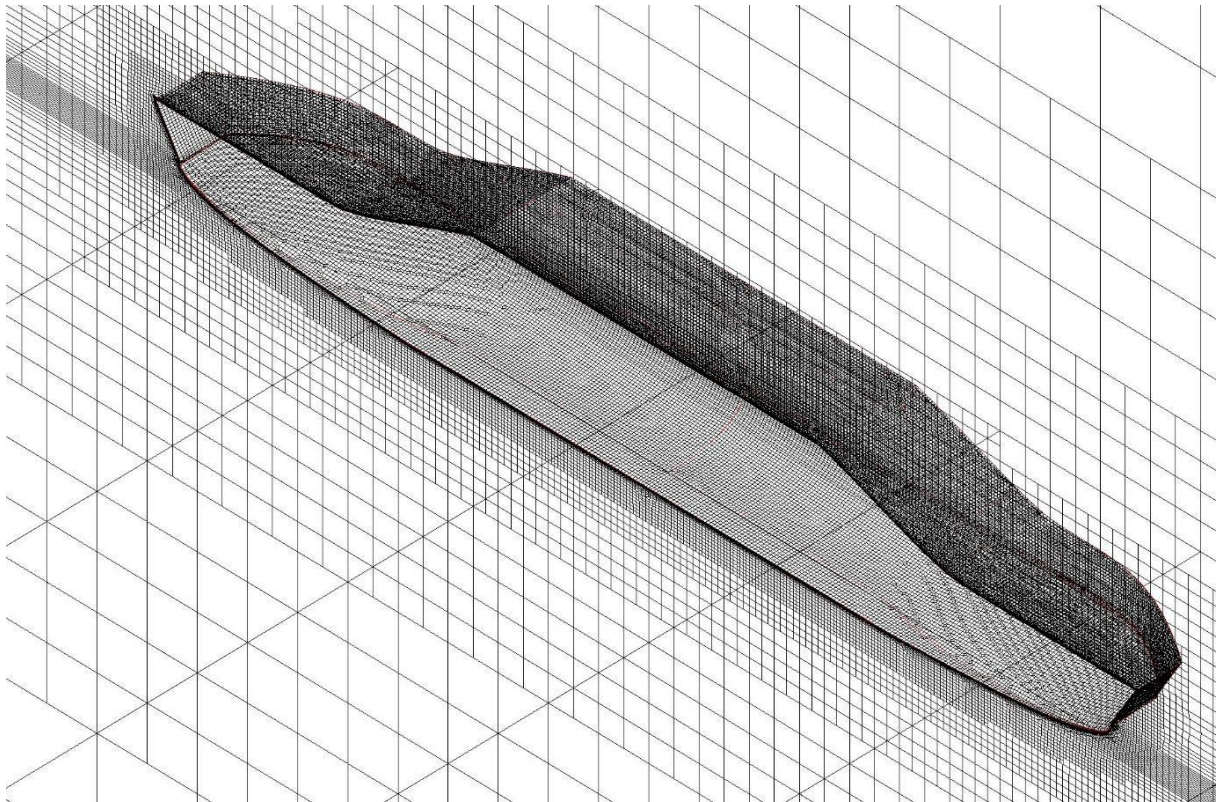
### 5.1. Mesh generation

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

Figures 29. – 39. show some examples of the fine mesh for the full size model of the double-ended ferry. For the model at the design draught the number of used cells was 1,354,651 with 1,424,455 vertices, while for the model at the maximum draught the number of used cells was 1,347,162 with 1,428,015 vertices.

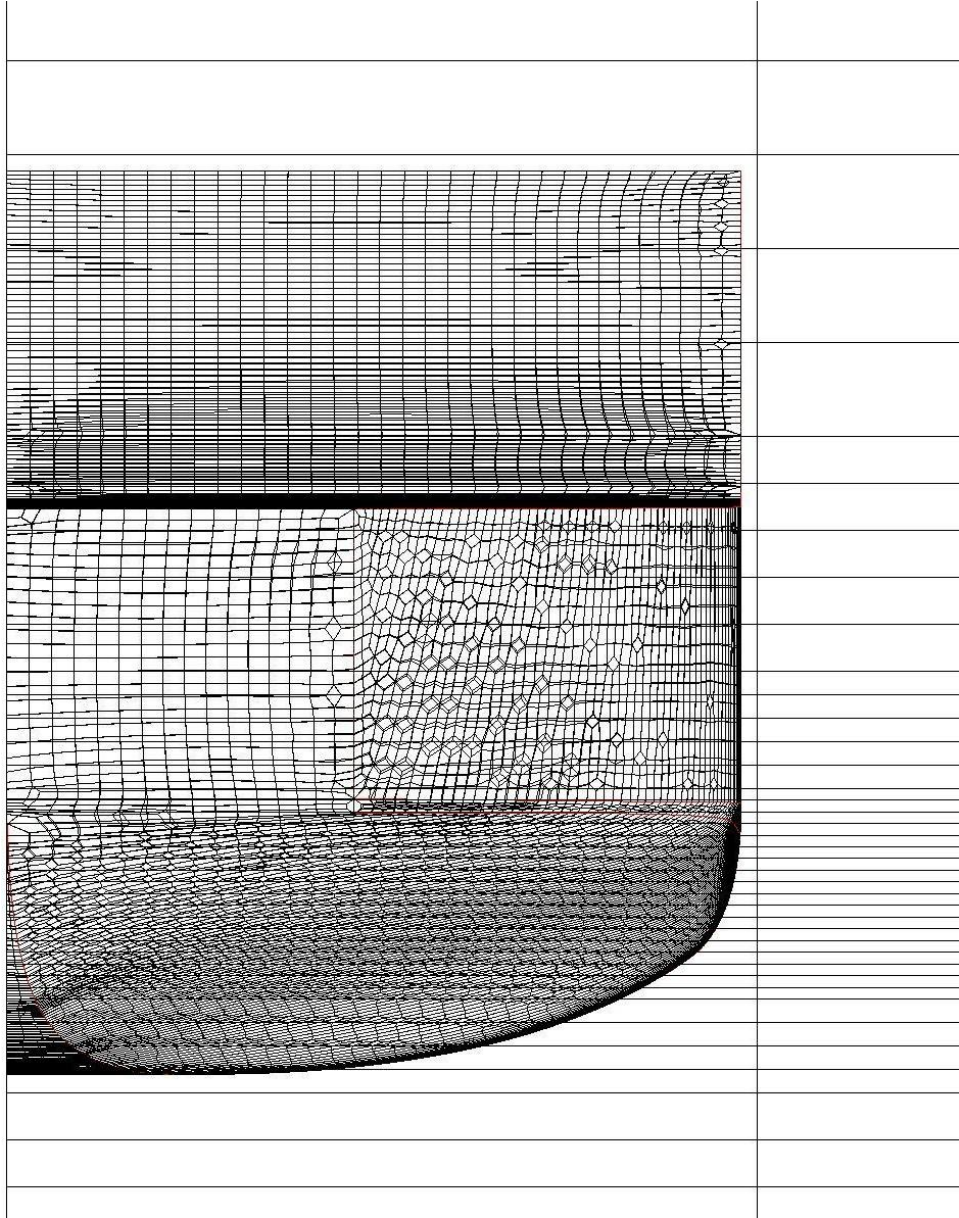


*Figure 29. Fine mesh for the full scale model,  $T=2.3$  m*

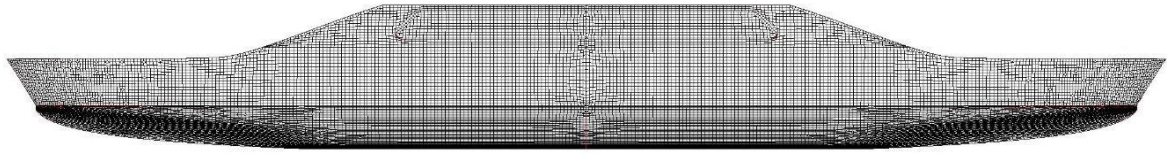


*Figure 30. Fine mesh for the full scale model,  $T=2.3$  m, zoomed view*

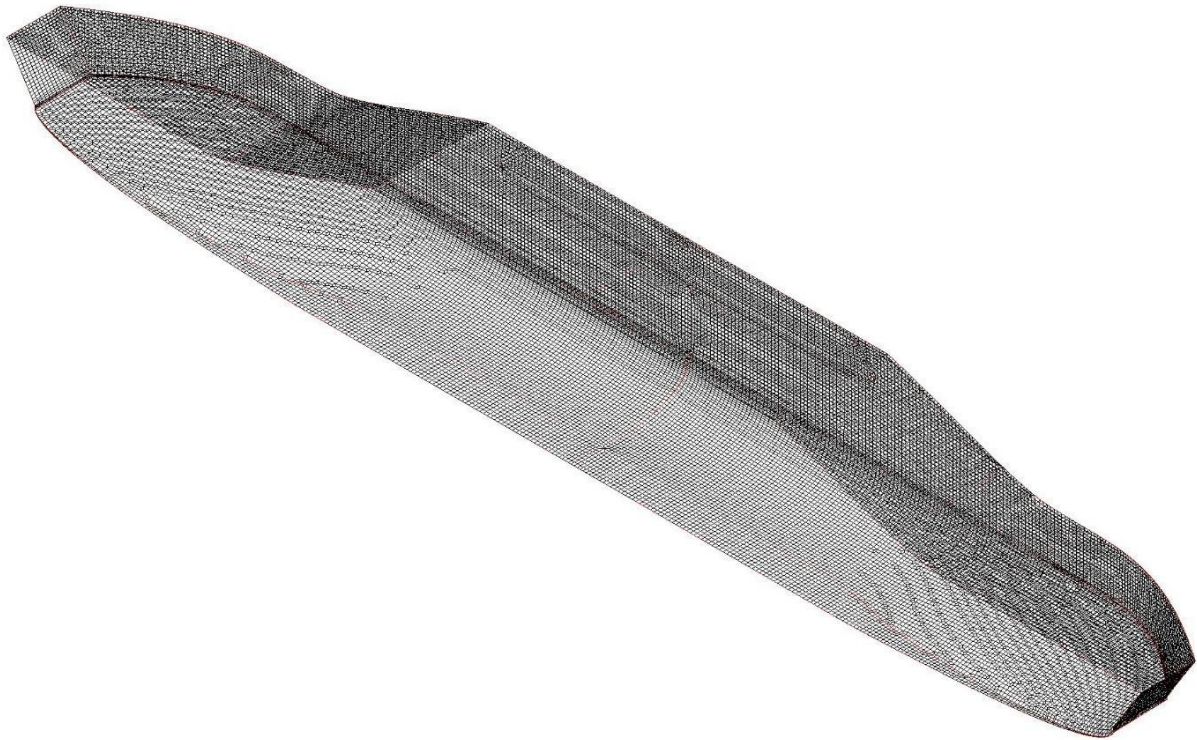




*Figure 31. Fine mesh for the full scale model,  $T=2.3$  m, X-plane view*

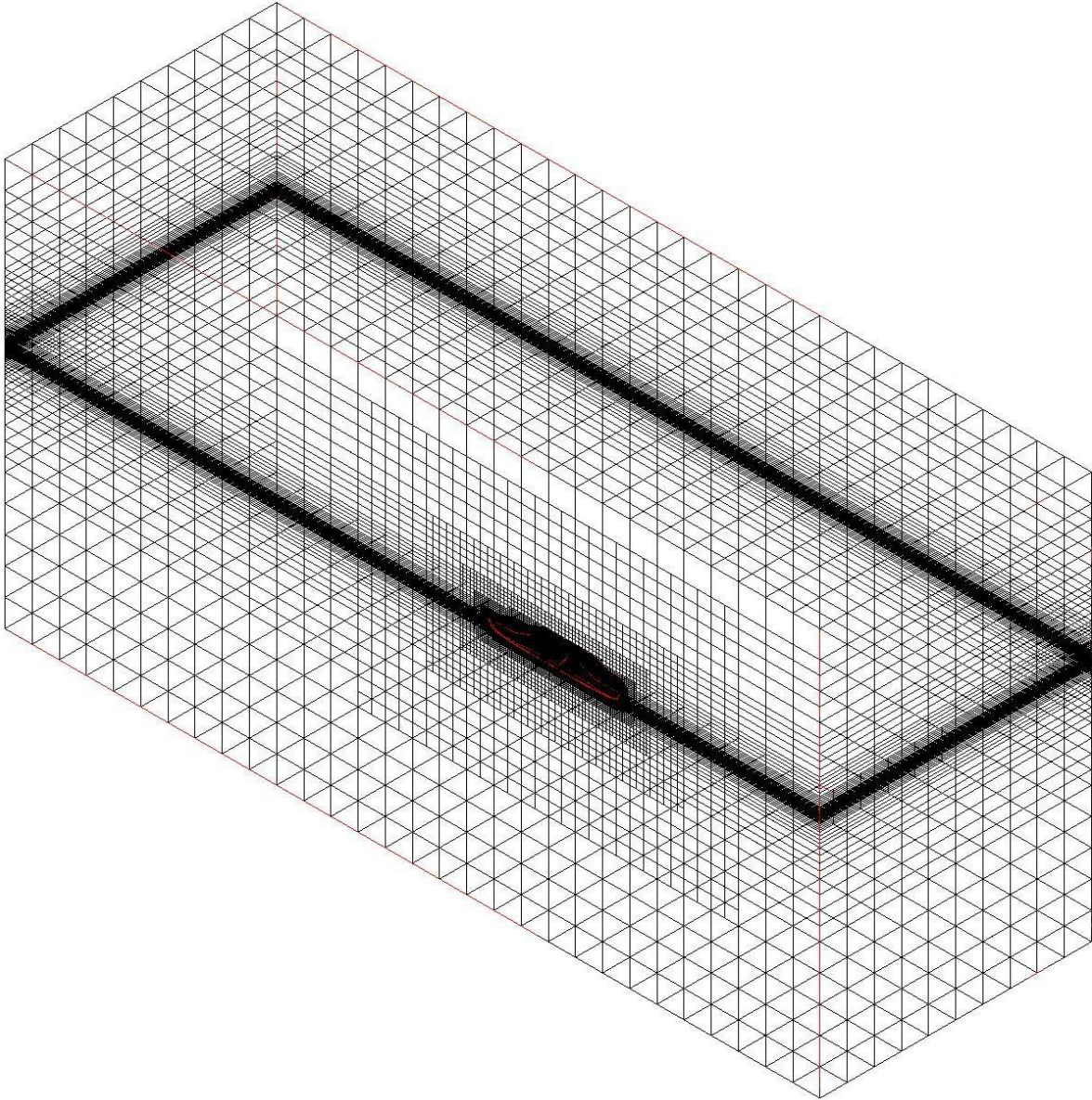


*Figure 32.* Fine mesh for the full scale model,  $T=2.3$  m, Y-plane view

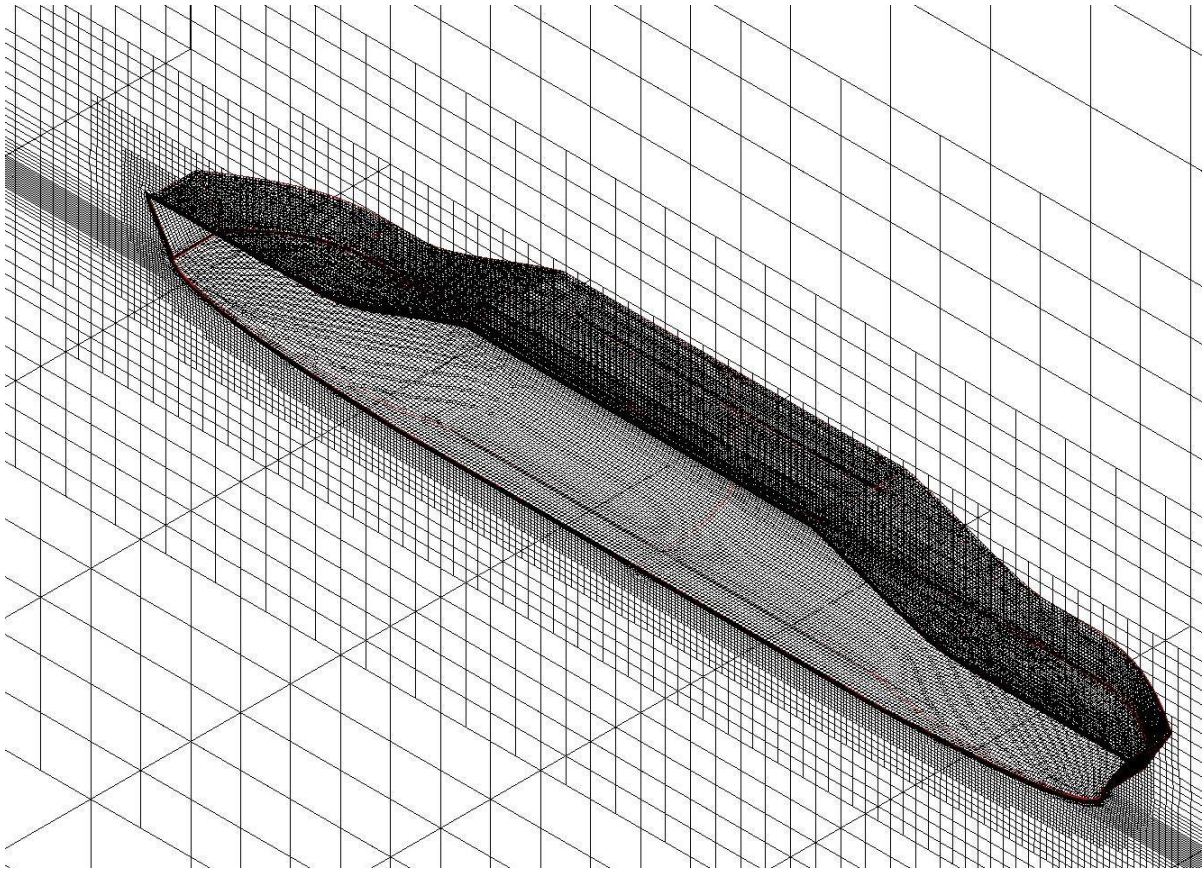


*Figure 33.* Fine mesh for the full scale model,  $T=2.3$  m, isometric view



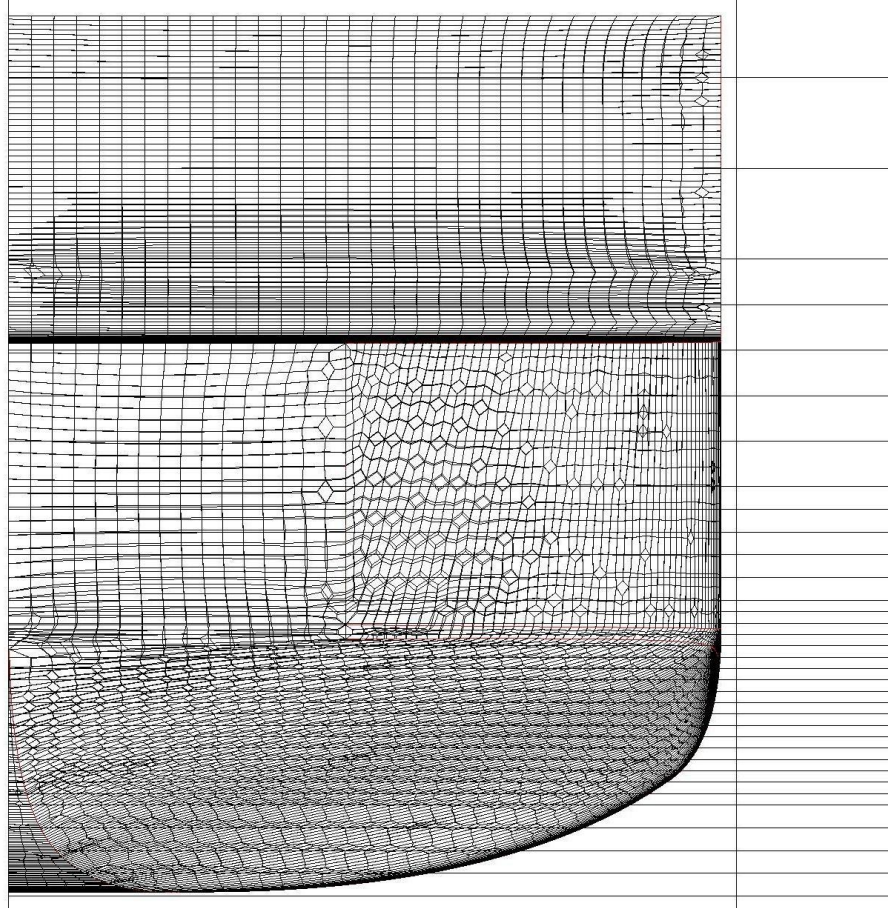


*Figure 34. Fine mesh for the full scale model,  $T=2.5$  m*

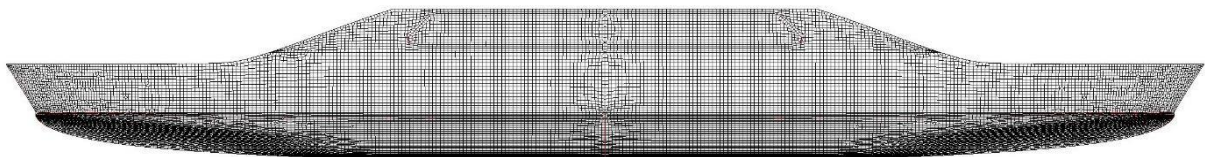


*Figure 35. Fine mesh for the full scale model,  $T=2.5$  m, zoomed view*

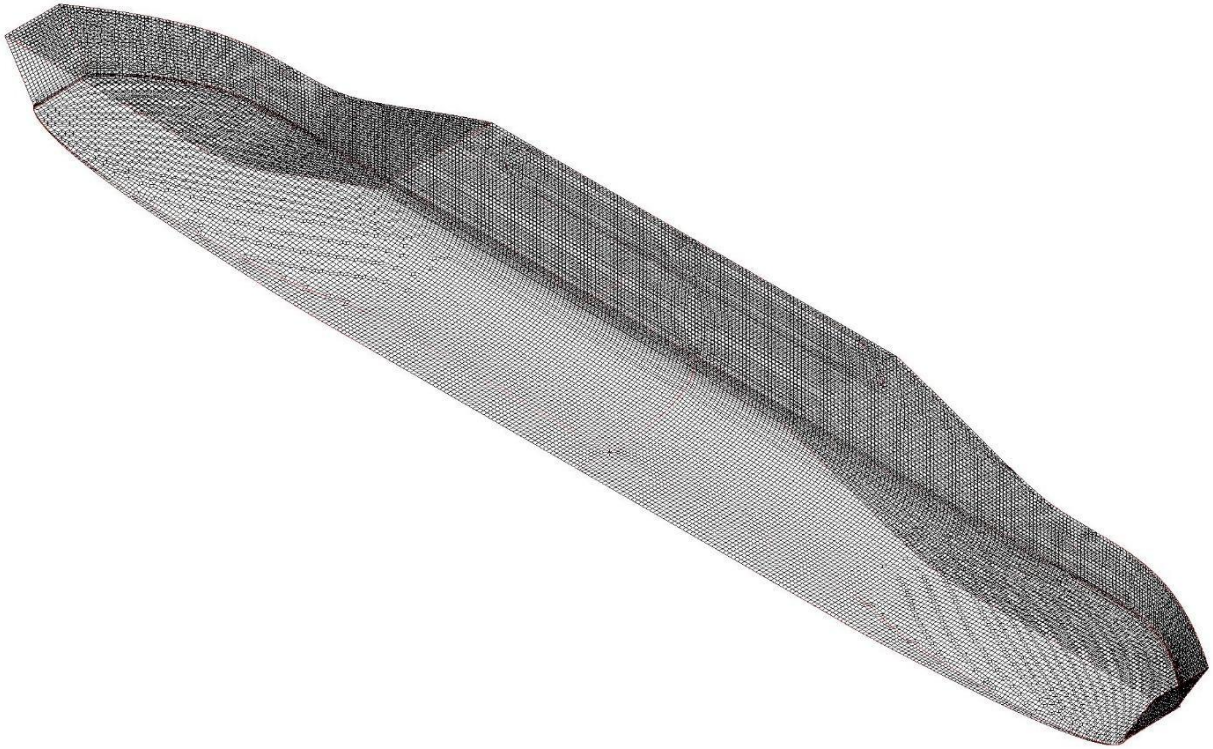




*Figure 36. Fine mesh for the full scale model,  $T=2.5$  m, X-plane view*



*Figure 37. Fine mesh for the full scale model,  $T=2.5$  m, Y-plane view*



*Figure 38.* Fine mesh for the full scale model,  $T=2.5$  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  $1024.8 \text{ kg/m}^3$ , dynamic viscosity  $0.001077 \times 10^{-3} \text{ N s/m}^2$ ,
- air: density  $1.2 \text{ kg/m}^3$ , dynamic viscosity  $1.85 \times 10^{-5} \text{ N s/m}^2$ .

## 5.3. Results

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

### 5.3.1. Design draught $T = 2.3 \text{ m}$

Table 6. shows the numerically obtained resistance for the corresponding Froude numbers at design draught of 2.3 m. The numerical values are graphically shown in the Figure 39.

*Table 6. Resistance vs. Froude number*

$V_{kN}$ , knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0
$V$ , m/s	1.0289	2.0578	3.0867	4.1156	5.1444	6.1733	7.2022
$Fr$	0.0329	0.0658	0.0987	0.1315	0.1644	0.1973	0.2302
$F_x$ , kN	4.071	11.561	17.485	30.134	49.062	79.096	120.931

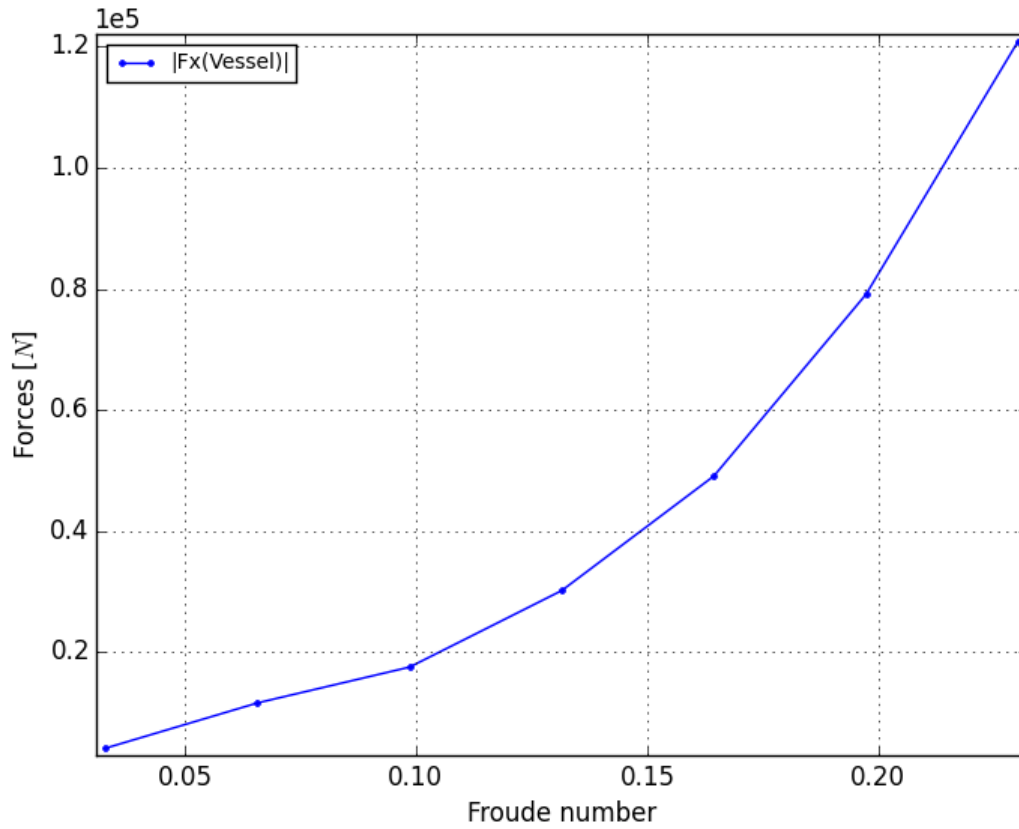
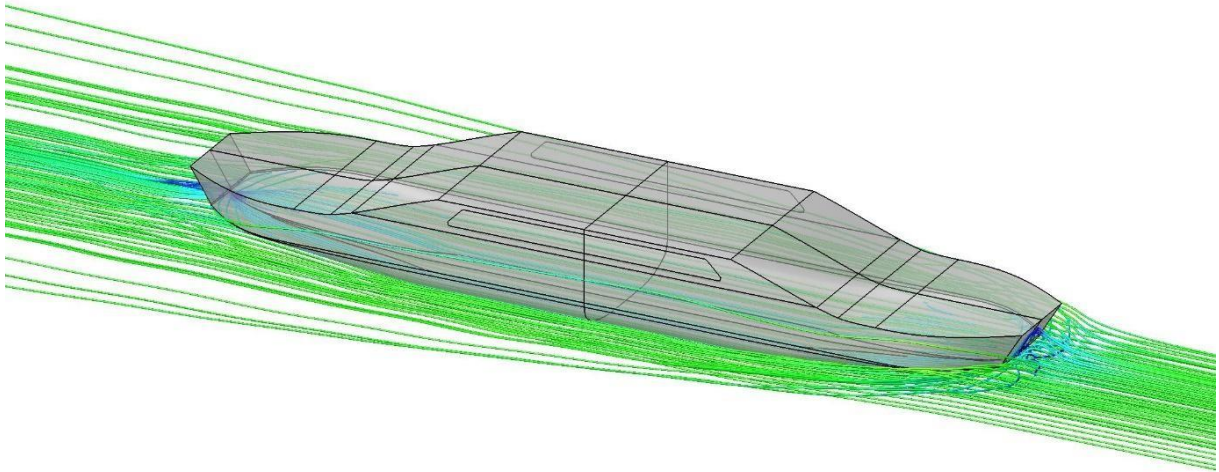


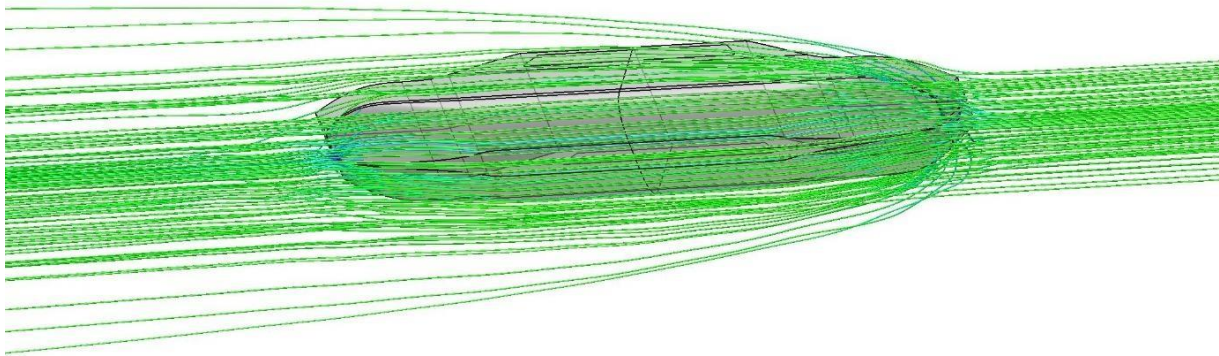
Figure 40. Resistance vs. Froude number

Figures 41. – 50. visually show some results obtained by CFD simulations. First, the results for the design speed of 12.0 knots and then the results for other speeds between 2.0 knots and 14.0 knots were shown.





*Figure 41. Relative velocity streamlines, design speed  $V= 12.0$  knots*



*Figure 42. Relative velocity streamlines, design speed  $V= 12.0$  knots*

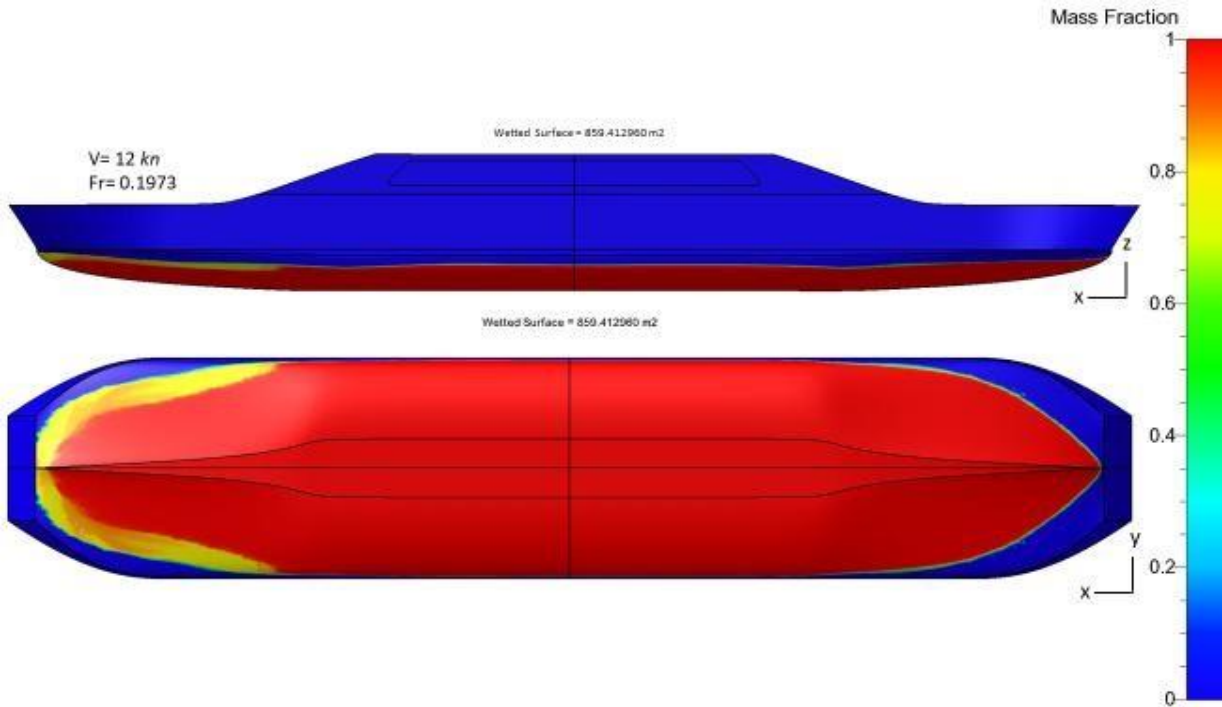


Figure 43. Wetted surface, design speed  $V= 12.0$  knots

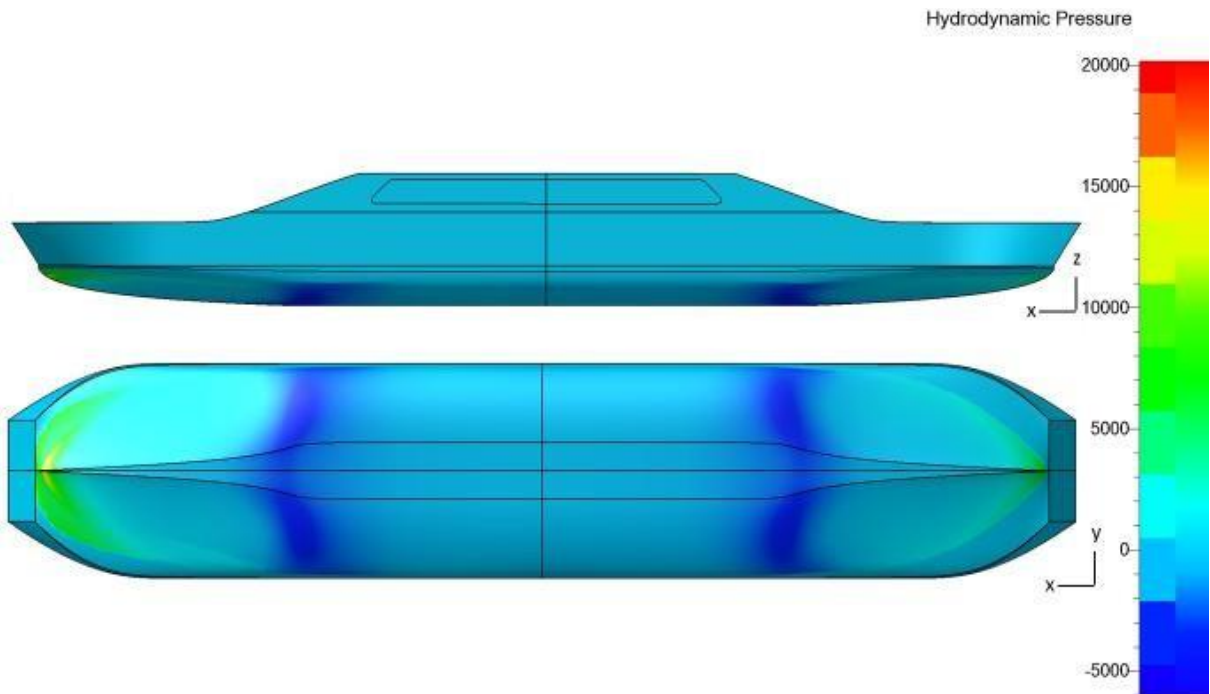


Figure 44. Hydrodynamic pressure, design speed  $V= 12.0$  knots

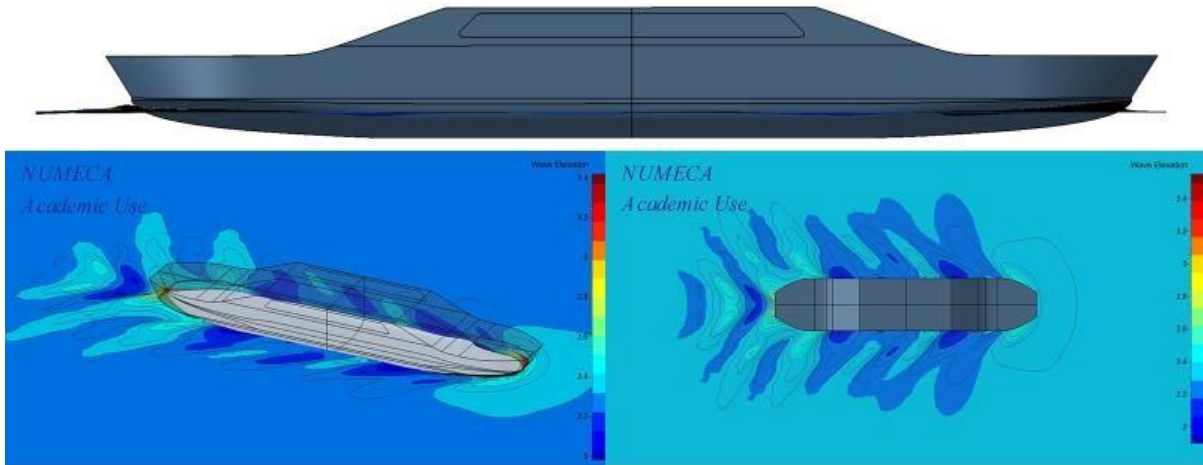


Figure 45. Wave elevations, design speed  $V=12.0$  knots

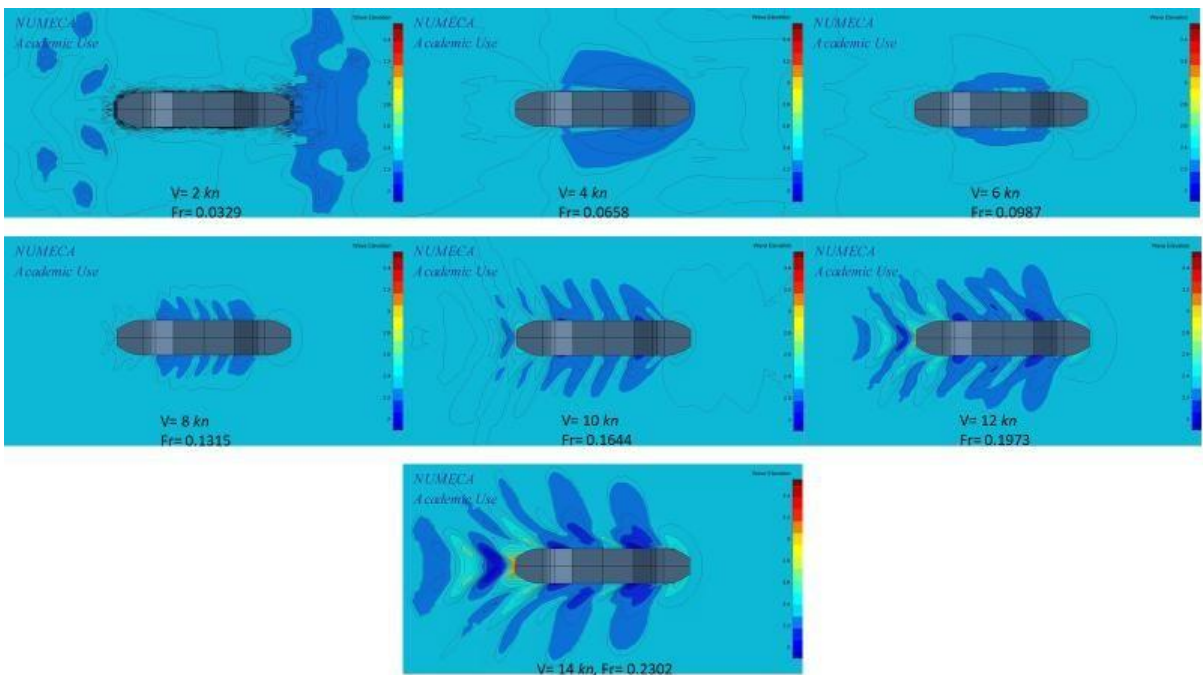


Figure 46. Wave elevations,  $V=2.0$  knots to  $14.0$  knots, Y-plane view



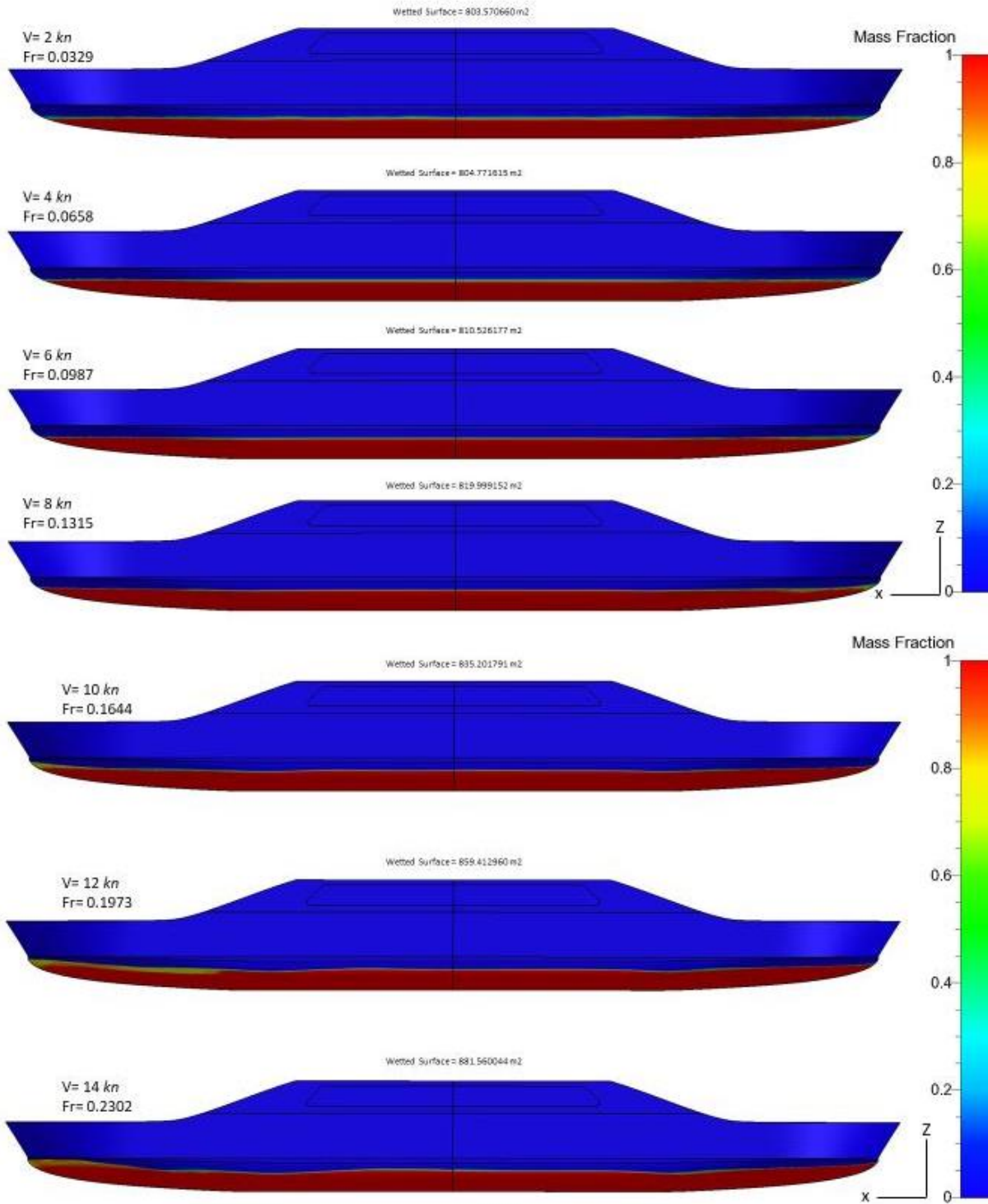
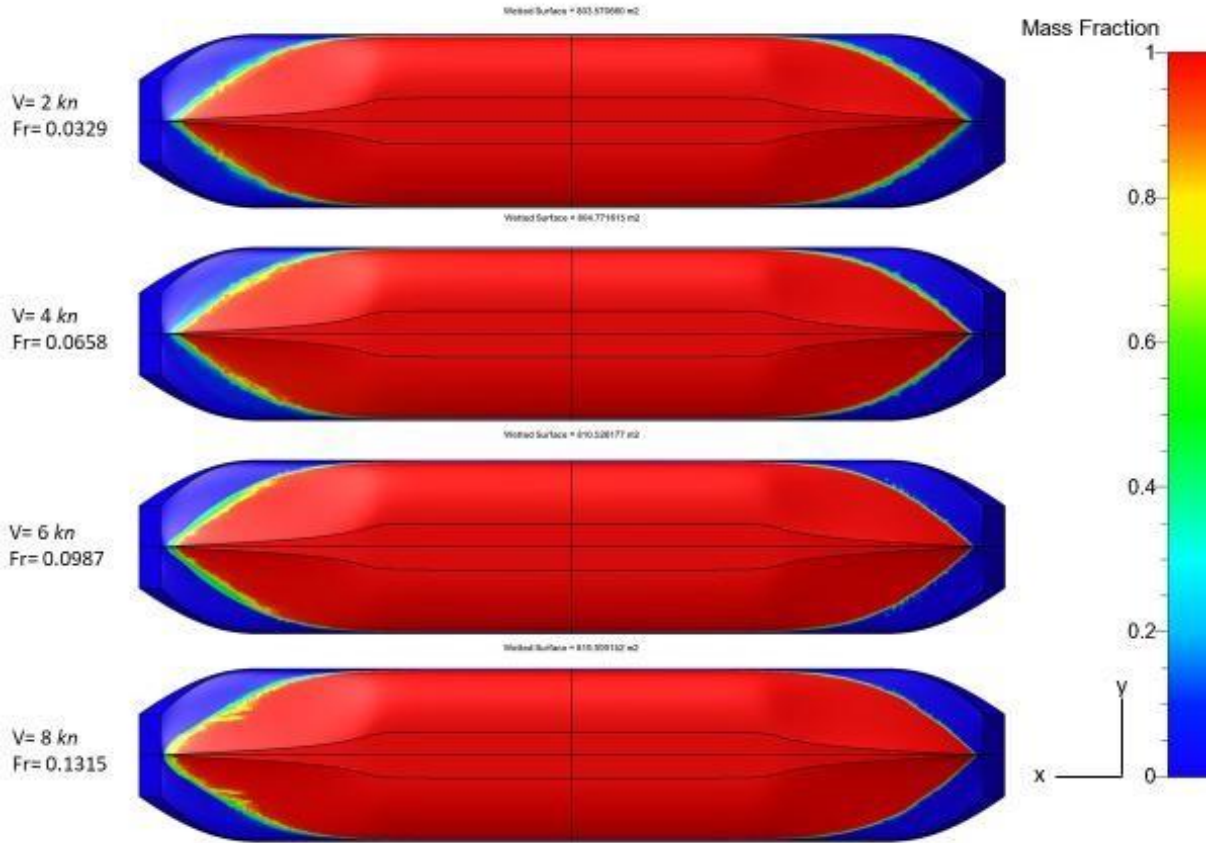


Figure 47. Wetted surface, V= 2.0 knots to 14.0 knots, Y-plane view





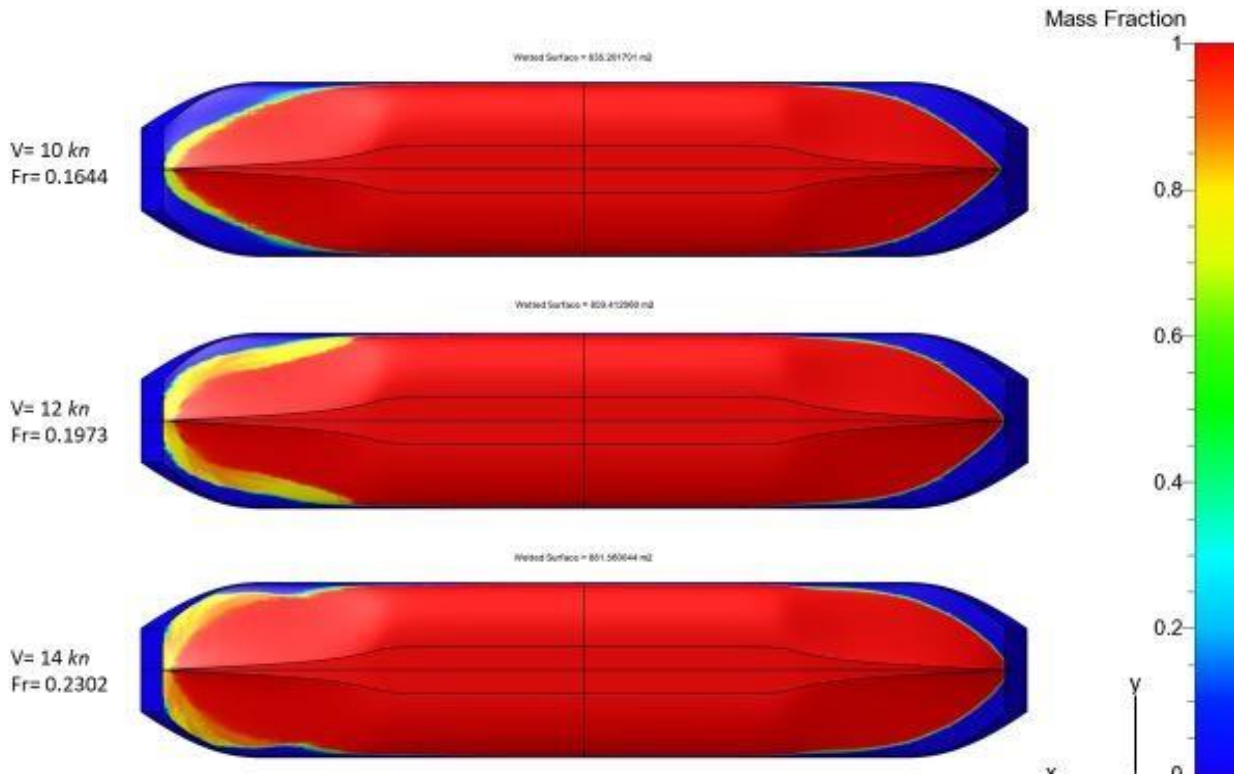


Figure 48. Wetted surface, V= 2.0 knots to 14.0 knots, Z-plane bottom view

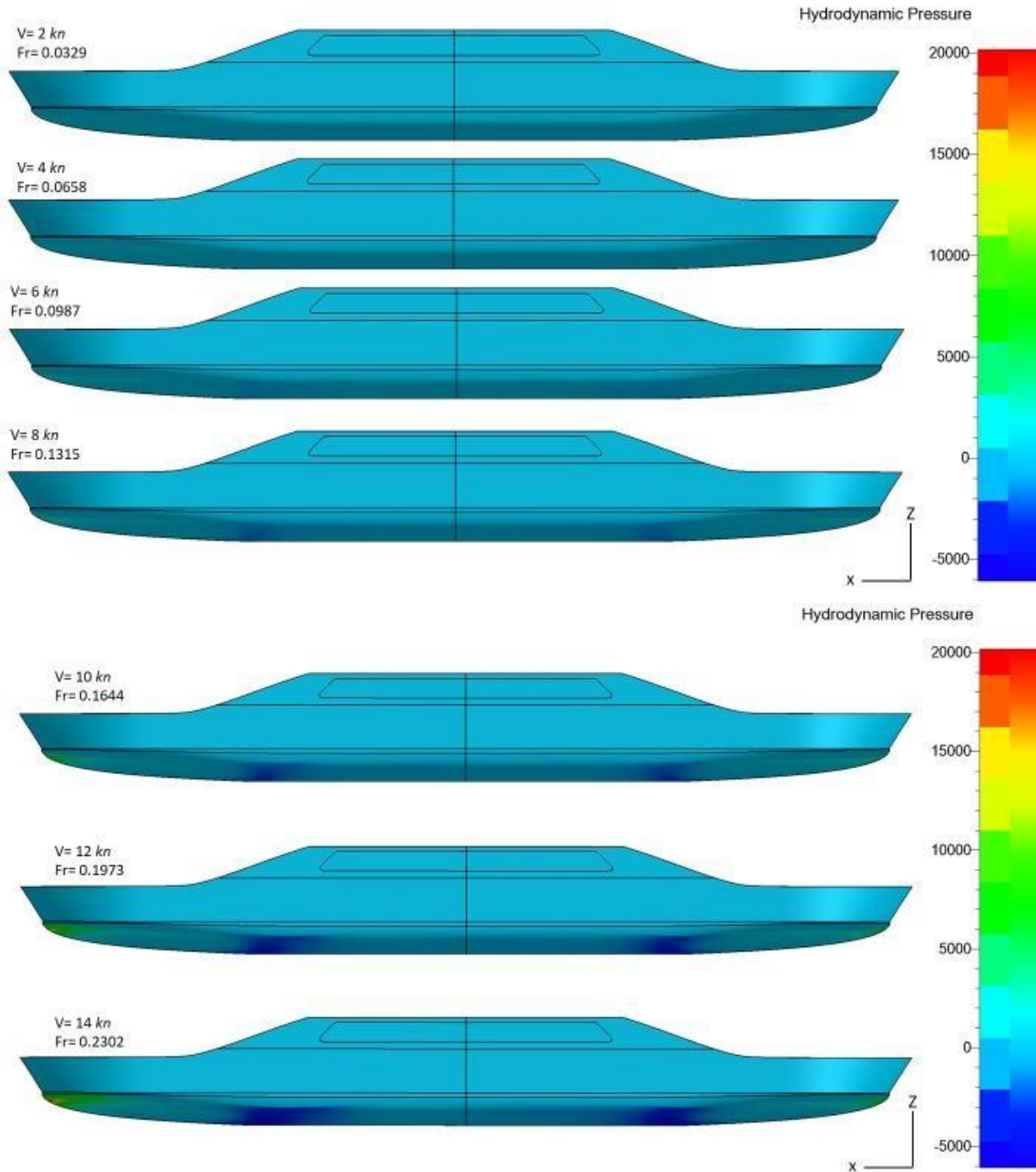
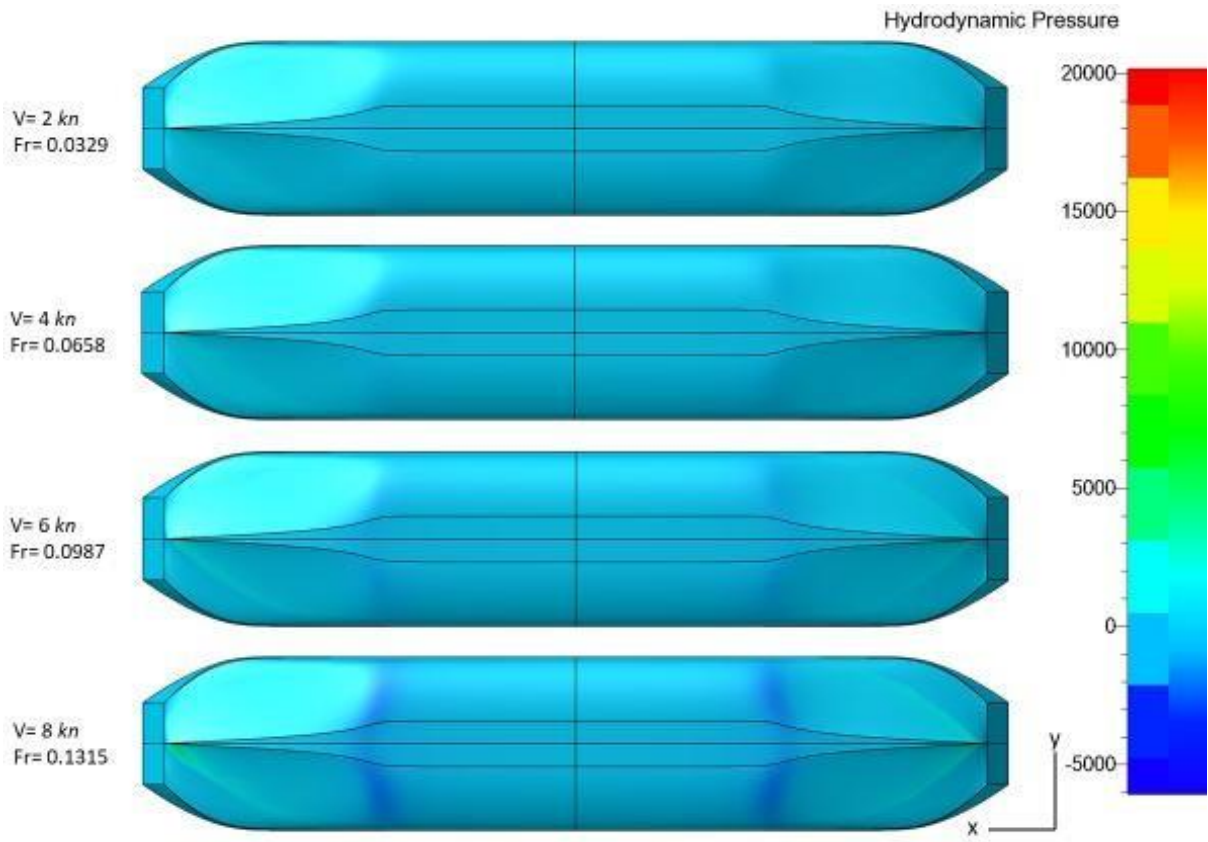


Figure 49. Hydrodynamic pressure,  $V = 2.0$  knots to  $14.0$  knots, Y-plane view





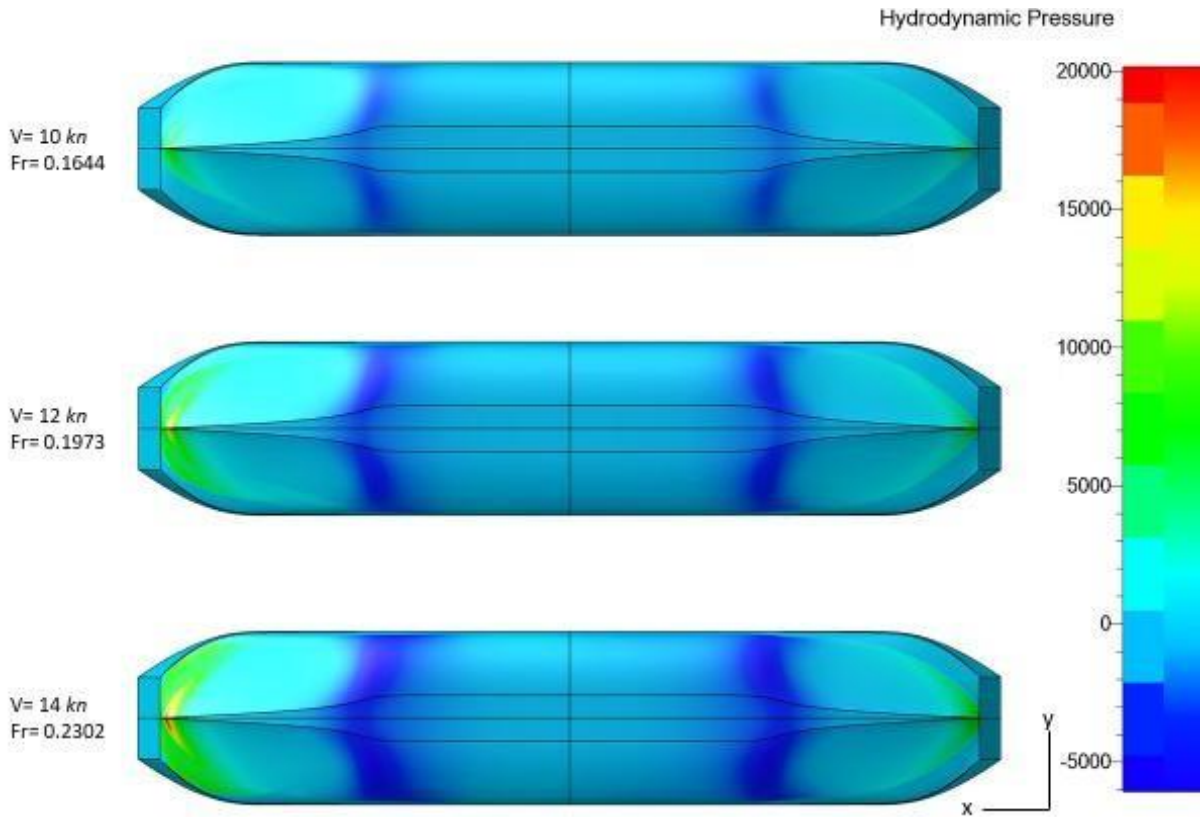


Figure 50. Hydrodynamic pressure,  $V = 2.0$  knots to  $14.0$  knots, Z-plane bottom view

### 5.3.2. Maximum draught $T = 2.5$ m

Table 7. shows the numerically obtained resistance for the corresponding Froude numbers at maximum draught of  $2.5$  m. The numerical values are graphically shown in the Figure 51.

Table 7. Resistance vs. Froude number

$V_{KN}$ , knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0
$V$ , m/s	1.0289	2.0578	3.0867	4.1156	5.1444	6.1733	7.2022
$Fr$	0.0329	0.0658	0.0987	0.1315	0.1644	0.1973	0.2302
$F_x$ , kN	4.242	10.012	18.704	32.724	54.053	84.455	129.819

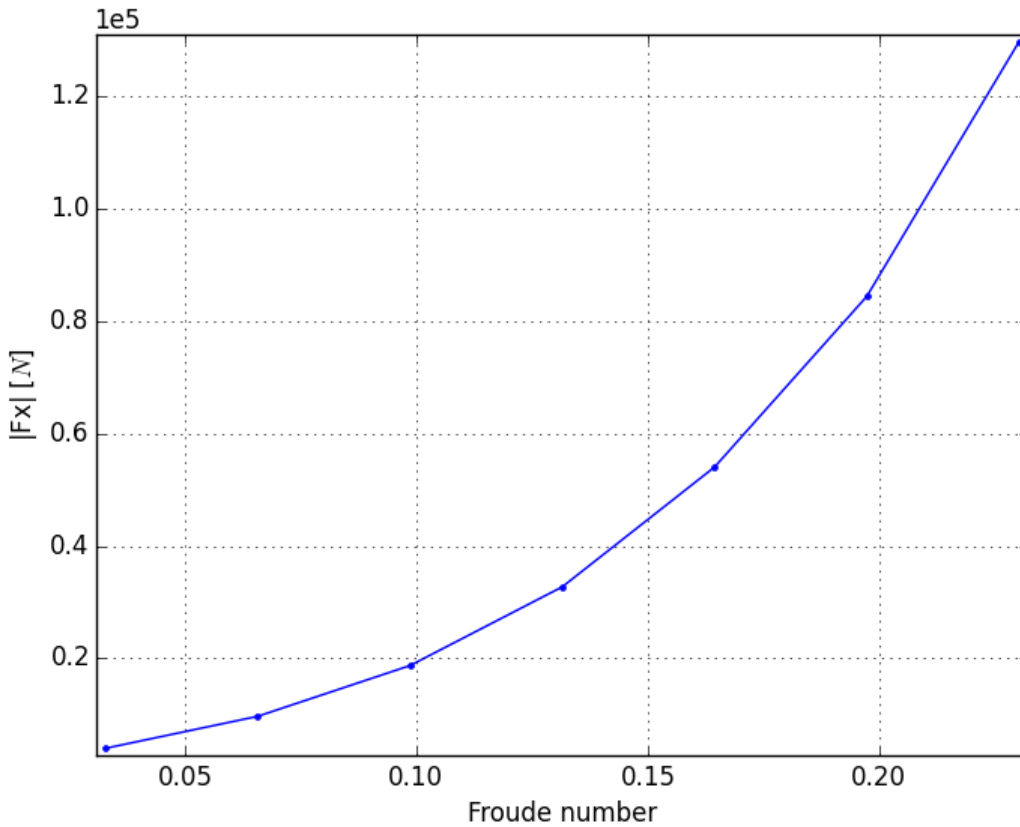
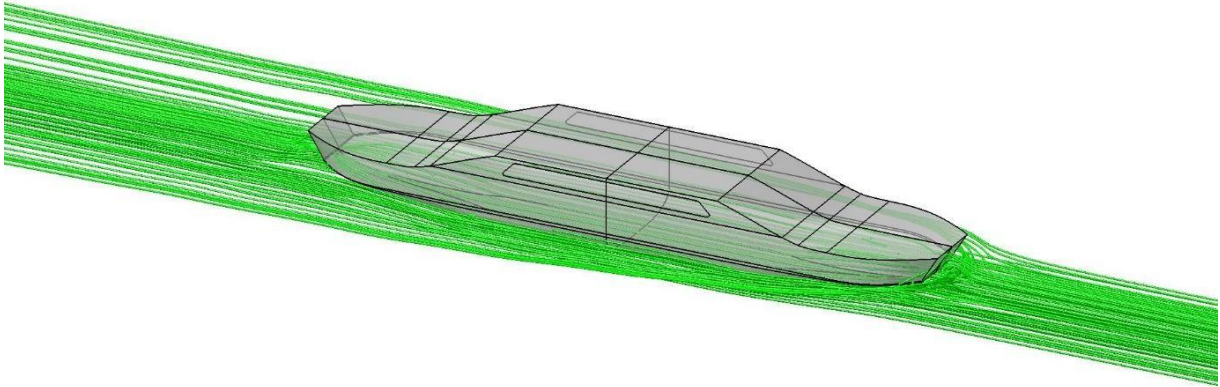
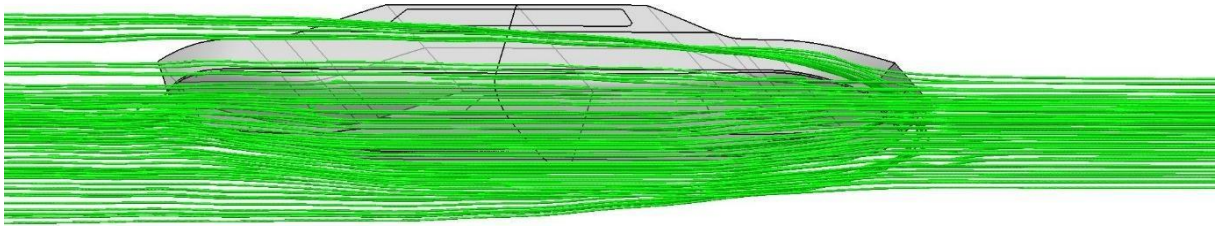


Figure 51. Resistance vs. Froude number

Figures 52. – 61. visually show some results obtained by CFD simulations. First, the results for the design speed of 12.0 knots and then the results for other speeds between 2.0 knots and 14.0 knots were shown.



*Figure 52. Relative velocity streamlines, design speed  $V= 12.0$  knots*



*Figure 53. Relative velocity streamlines, design speed  $V= 12.0$  knots*

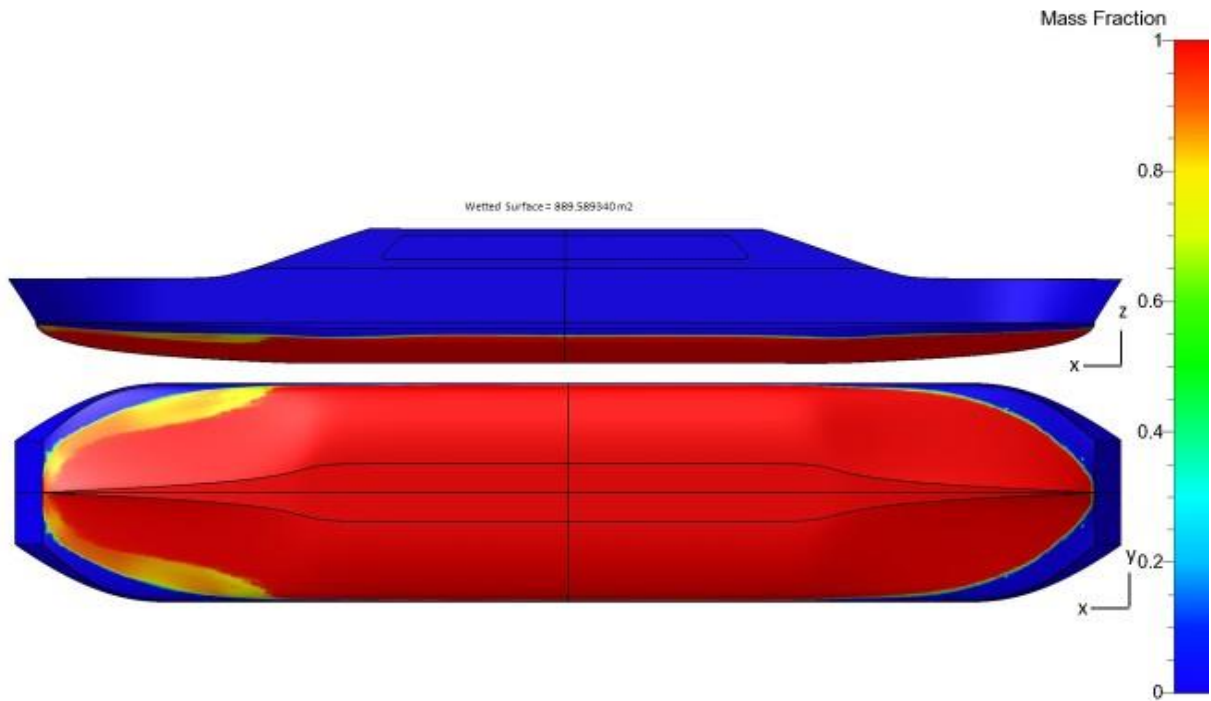


Figure 54. Wetted surface, design speed  $V= 12.0$  knots



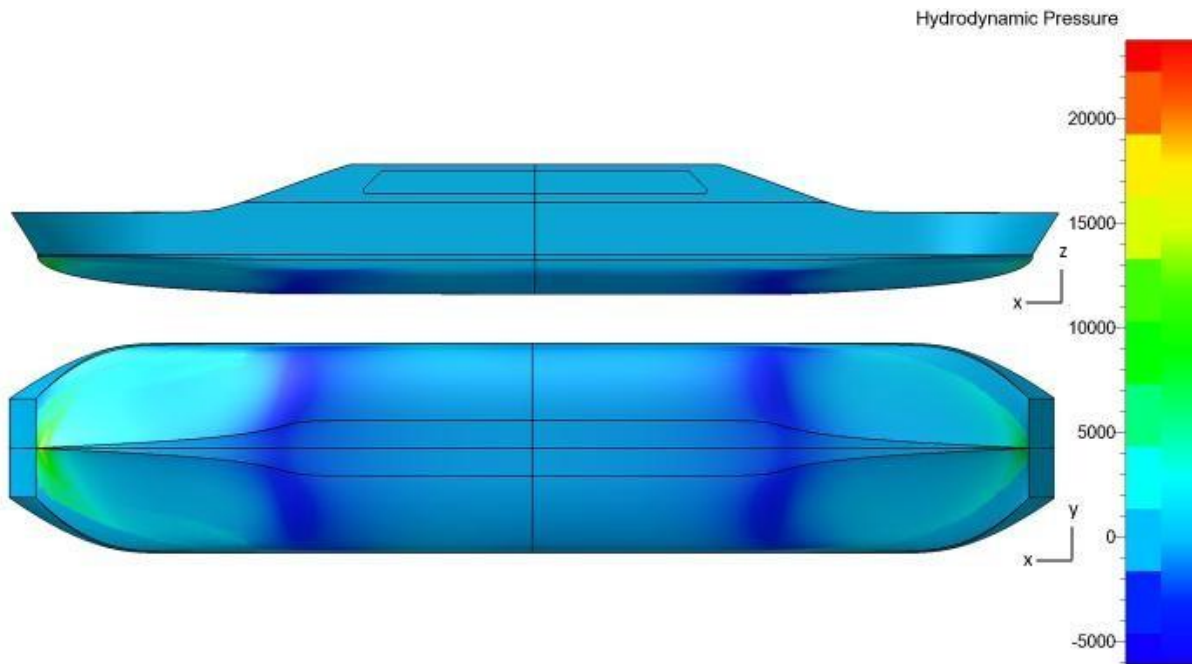


Figure 55. Hydrodynamic pressure, design speed  $V= 12.0$  knots

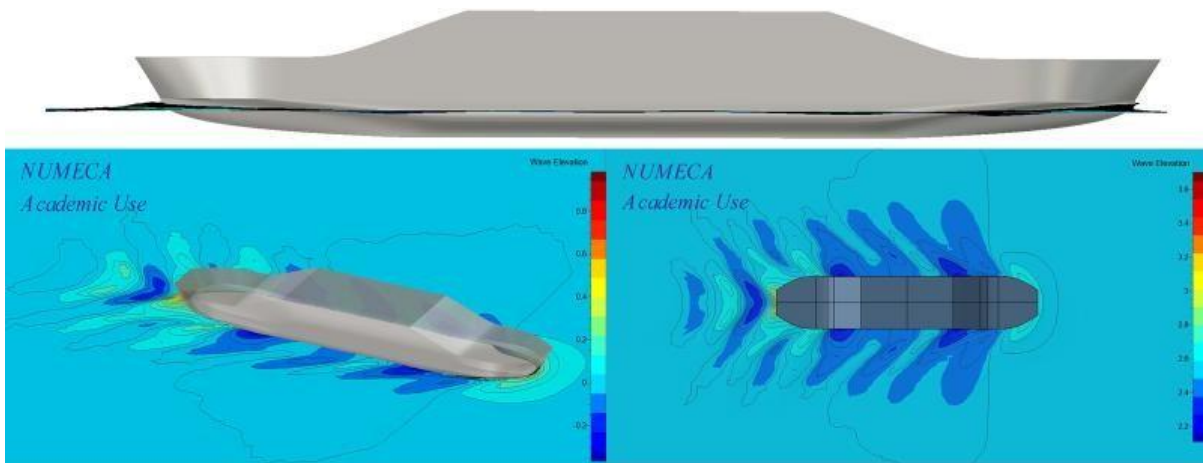


Figure 56. Wave elevations, design speed  $V= 12.0$  knots

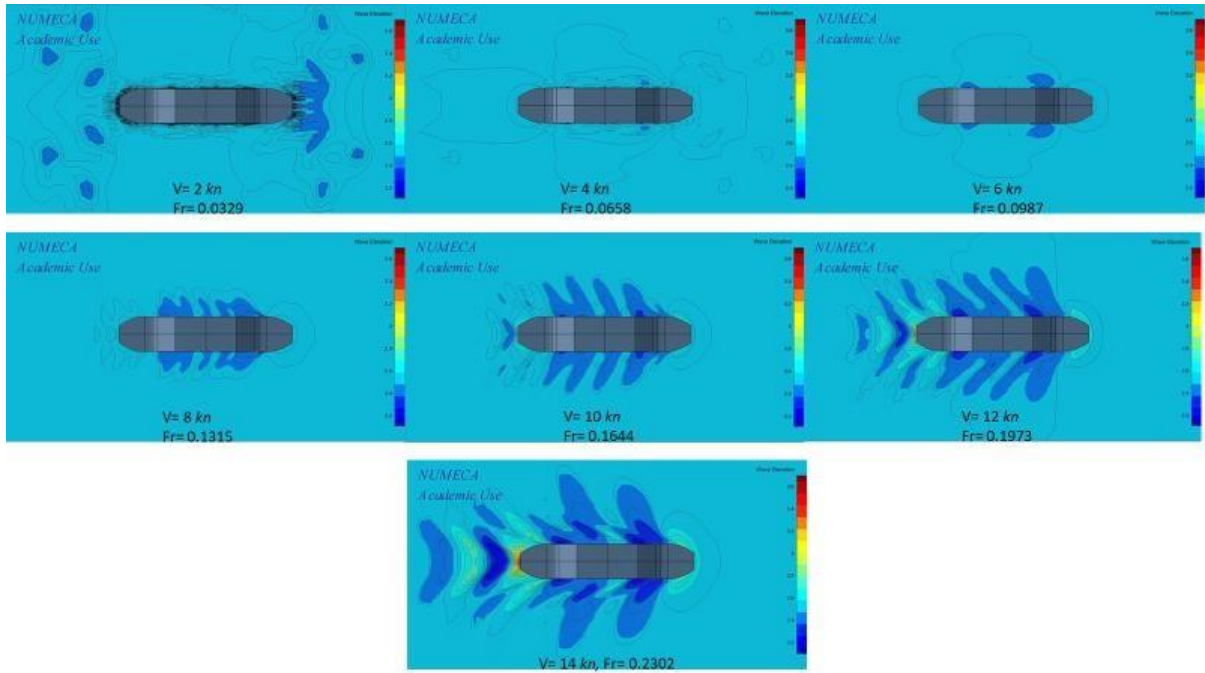


Figure 57. Wave elevations, V= 2.0 knots to 14.0 knots, Y-plane view

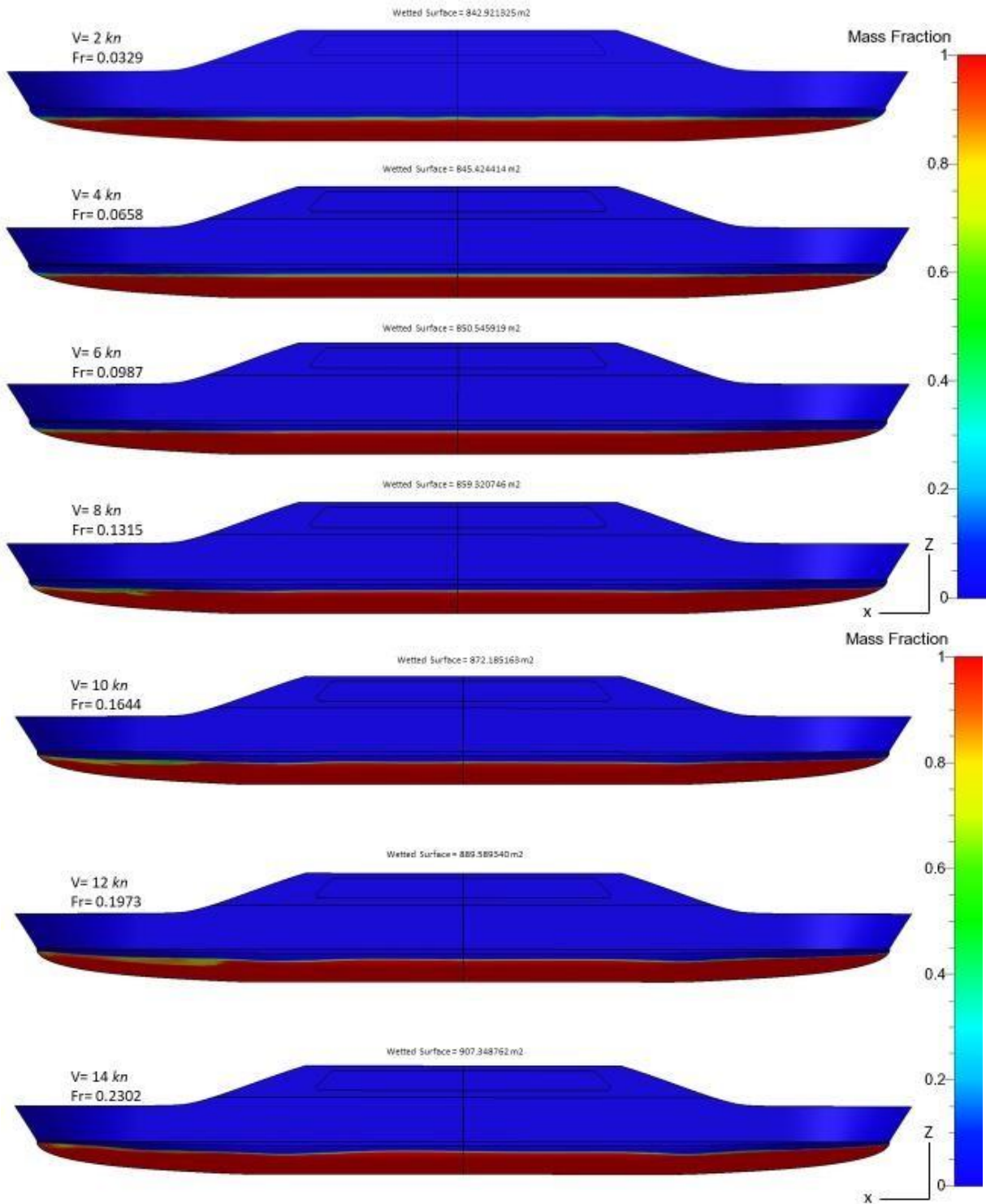
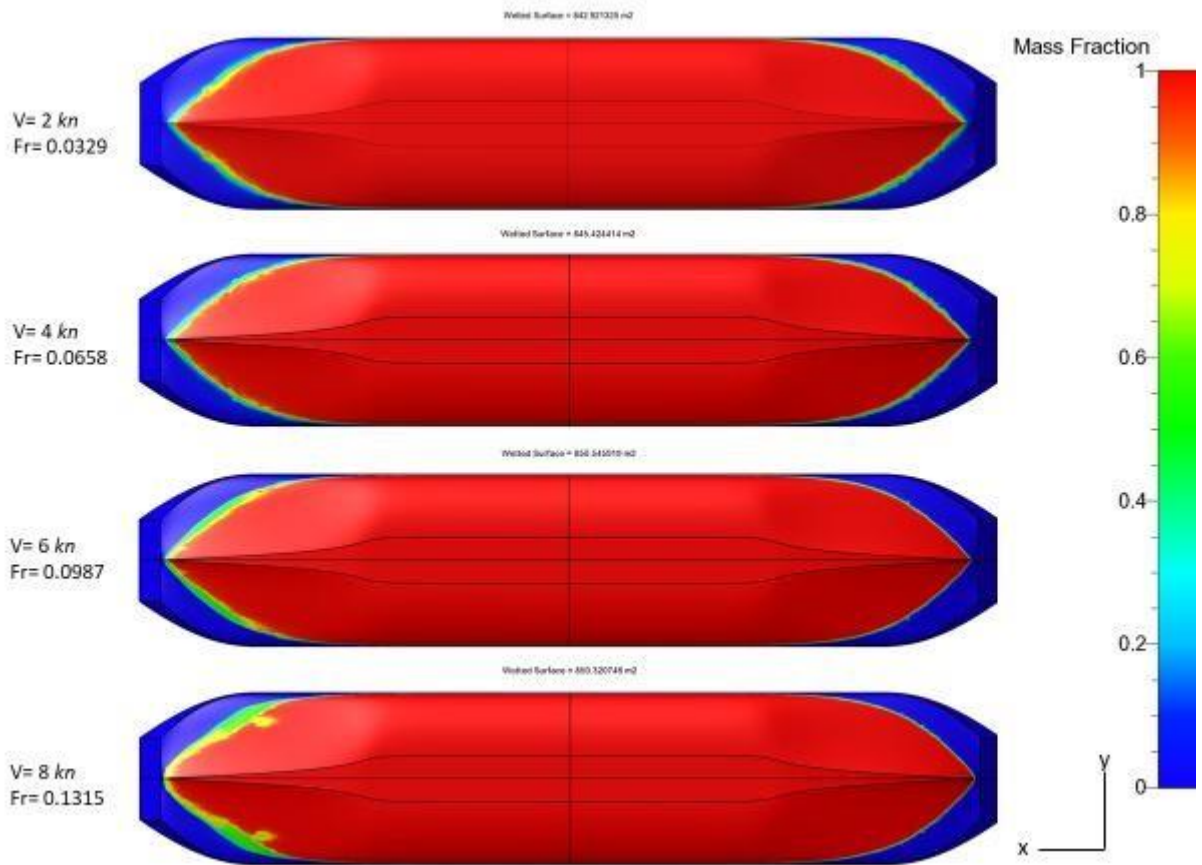


Figure 58. Wetted surface, V= 2.0 knots to 14.0 knots, Y-plane view





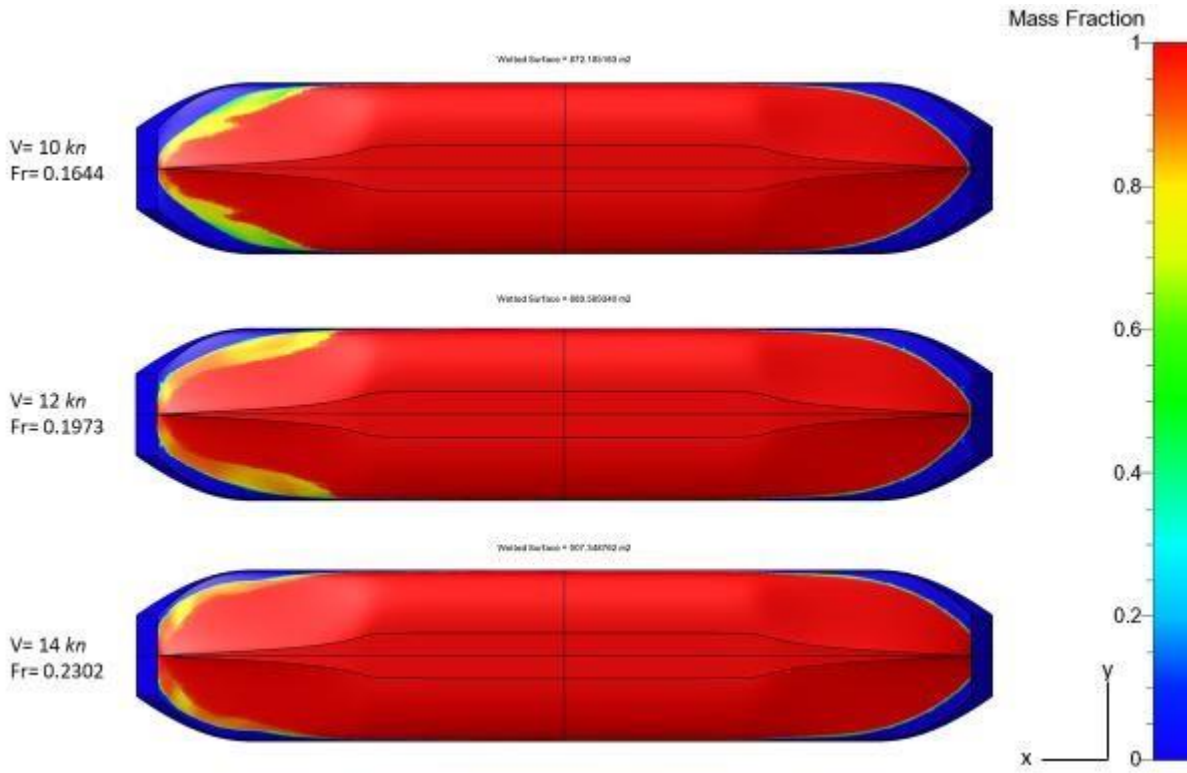


Figure 59. Wetted surface, V= 2.0 knots to 14.0 knots, Z-plane bottom view

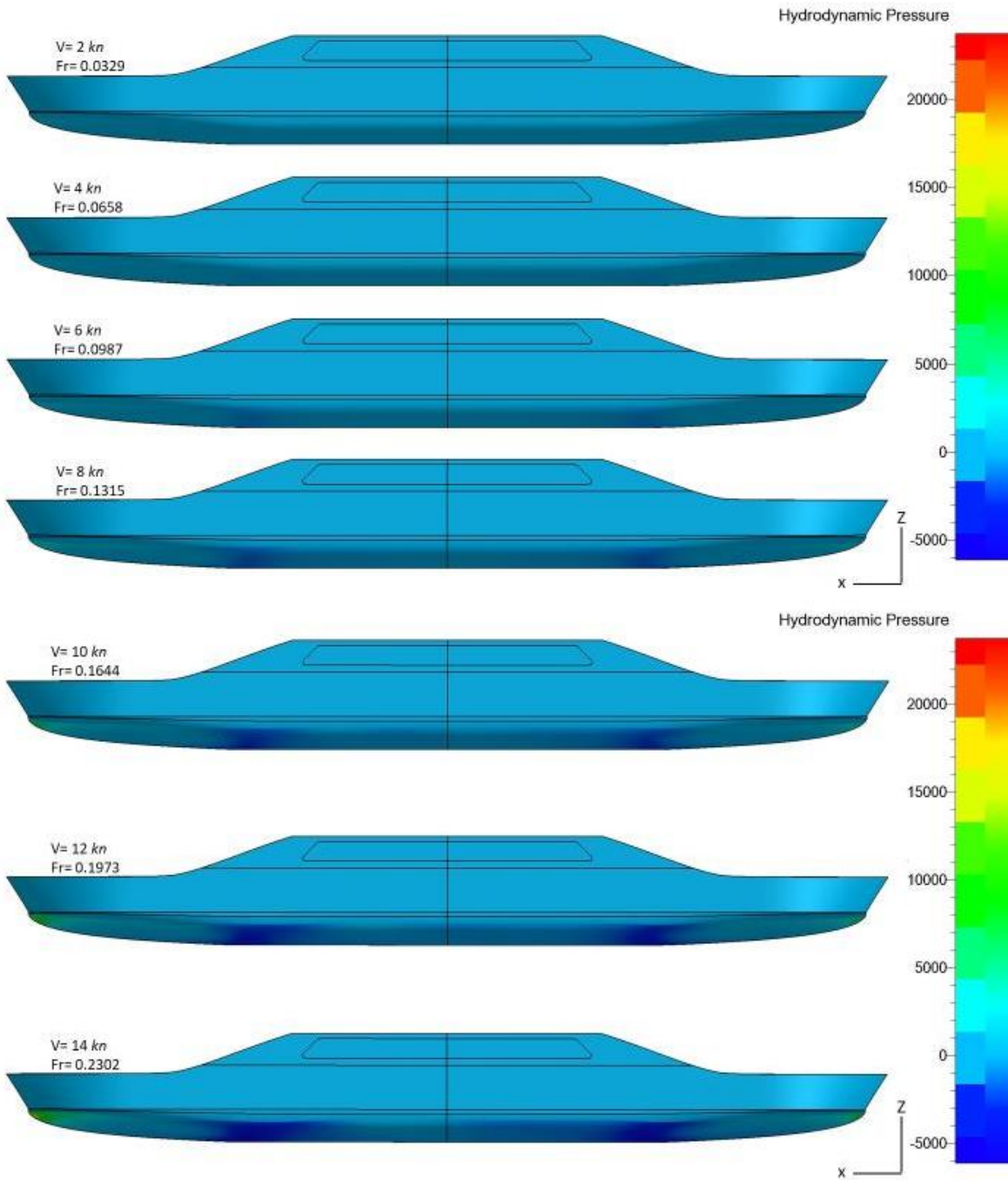
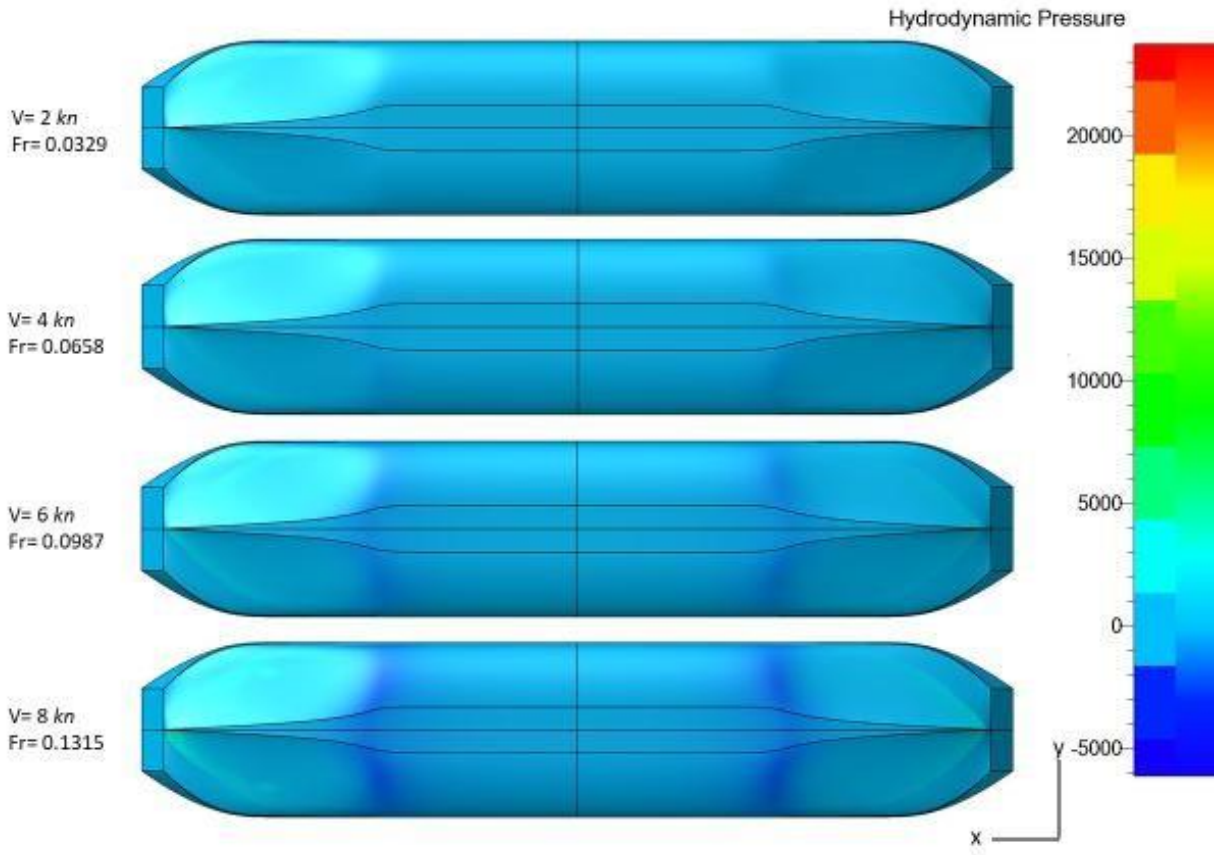


Figure 60. Hydrodynamic pressure,  $V= 2.0$  knots to  $14.0$  knots, Y-plane view



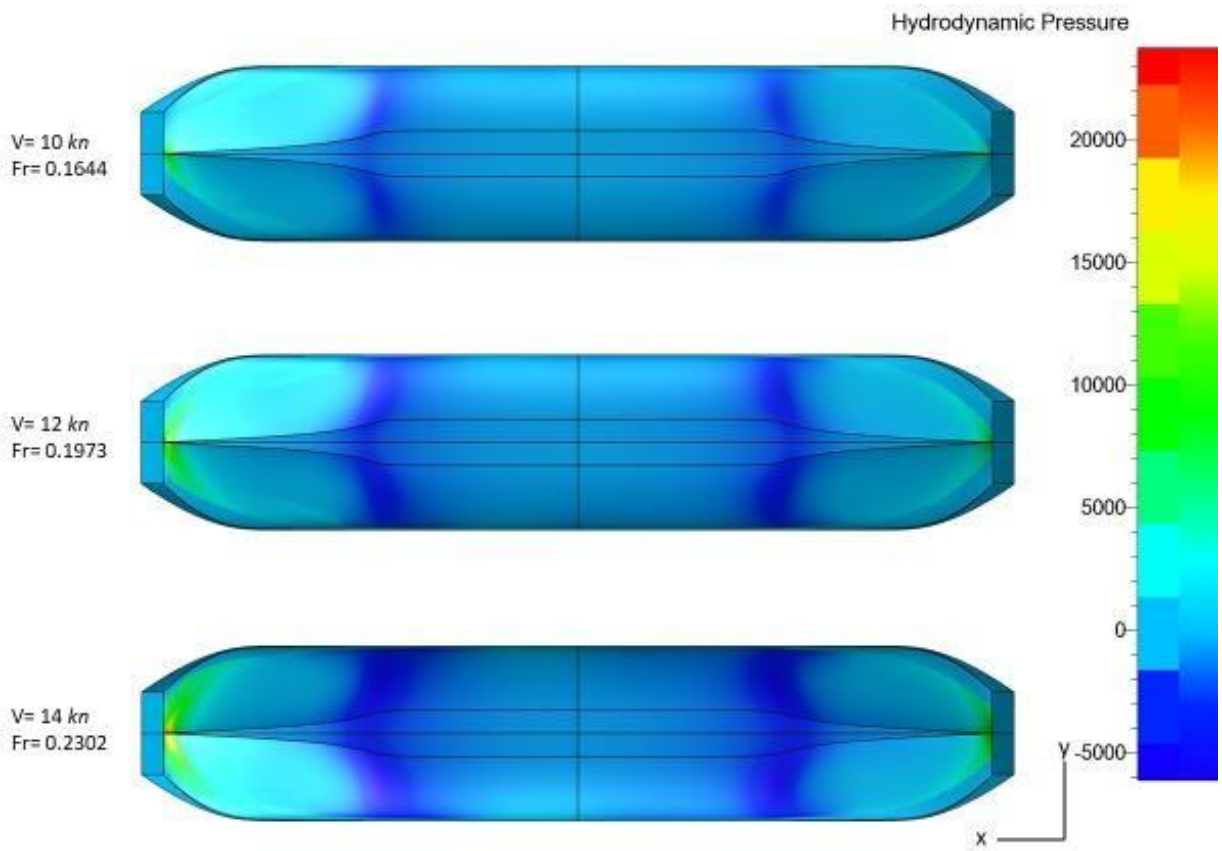


Figure 61. Hydrodynamic pressure, V= 2.0 knots to 14.0 knots, Z-plane bottom view



## 6. THE POWER-SPEED DIAGRAM

This estimation of the power-speed is made after the CFD analysis of the double-ended ferry is accomplished. Table 8. shows the results related to ship resistance for the draught  $T=2.3$  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 only appendages, the ferry has bilge keels (2 x HP300, abt. 35% of length).

Table 8. Resistance and effective power,  $T=2.3$  m

$V_{KN}$ , knot	2.0	4.0	6.0	8.0	10.0	12.0	14.0
$V$ , m/s	1.0289	2.0578	3.0867	4.1156	5.1444	6.1733	7.2022
$Fr$	0.0329	0.0658	0.0987	0.1315	0.1644	0.1973	0.2302
$R_T$ (CFD), kN	4.071	11.561	17.485	30.134	49.062	79.096	120.931
$R_{APP}$ , kN	0.082	0.297	0.631	1.079	1.637	2.301	3.070
$R_T$ , kN	4.153	11.858	18.116	31.213	50.699	81.397	124.001
$P_E$ , kW	4.273	24.400	55.917	128.458	260.816	502.491	893.084

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

Engine:

- MCR: 4 x 370 kW
- Rate of revolution:  $N_M = 750.0 \text{ min}^{-1}$

Propeller:

- $Z = 4$
- $D = 1.685$  m
- $P = 1.290$  m
- $A_E/A_0 = 0.55$
- Rate of revolution  $N_P = 370.0 \text{ min}^{-1}$

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

- $P_D = 362.6$  kW (100% of one thruster)
- $N_P = 370,0$  min<sup>-1</sup> (100% of one thruster)
- $V_{KN} = 14,0$  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
$V$ , m/s	1.0289	2.0578	3.0867	4.1156	5.1444	6.1733	7.2022
$Fr$	0.0329	0.0658	0.0987	0.1315	0.1644	0.1973	0.2302
$w$	0.100	0.100	0.100	0.100	0.100	0.100	0.100
$t$	0.180	0.180	0.180	0.180	0.180	0.180	0.180
$V_A$ , m/s	0.926	1.852	2.778	3.704	4.630	5.556	6.482
$\eta_H$	0.911	0.911	0.911	0.911	0.911	0.911	0.911
$P_T$ , kW	4.690	26.781	61.373	140.991	286.261	551.515	980.214
$T$ , kN (4 thrusters)	5.064	14.460	22.092	38.064	61.827	99.265	151.221
$T$ , kN (1 thruster)	1.266	3.615	5.523	9.516	15.457	24.816	37.805
$N_P$ , min <sup>-1</sup>	61.40	113.90	156.27	207.38	260.45	320.13	382.77
$\eta_O$	0.608	0.638	0.659	0.664	0.668	0.668	0.666
$\eta_R$	0.970	0.970	0.970	0.970	0.970	0.970	0.970
$\eta_S$	0.98	0.98	0.98	0.98	0.98	0.98	0.98
$\eta_P$	0.527	0.553	0.571	0.575	0.578	0.579	0.577
$P_B$ , kW (1 thruster)	2.027	11.039	24.504	55.834	112.752	217.067	386.953
$P_B$ , kW (4 thrusters)	8.109	44.157	98.015	223.336	451.007	868.266	1547.813

Figure 62. shows the corresponding power-speed diagram.

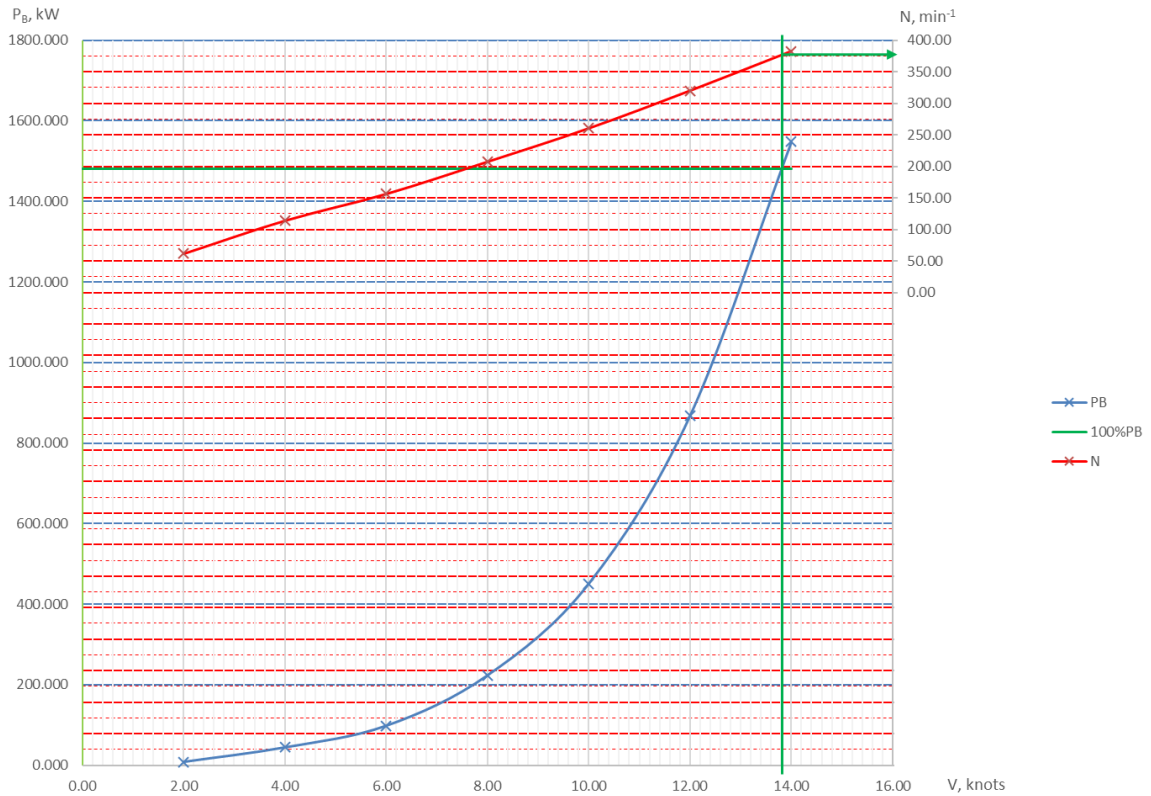


Figure 62. Power-speed diagram

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.

- $N_p = 379 \text{ min}^{-1}$
- $V_{KN} = 13.8 \text{ kn}$

Based on the obtained results, it can be concluded that with the envisaged propulsion engines and thrusters, the ferry would meet all the 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 double-ended 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 double-ended ferry show that the designed symmetrical double-ended hull would ensure 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 thruster propeller at the maximum propeller speed, a speed of almost 14 knots was obtained. On the basis of the obtained results it can be confirmed that the developed hybrid double-ended 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 continuous rating, kW
$N_M$	- engine rate of revolution, $\text{min}^{-1}$
$N_P$	- propeller rate of revolution, $\text{min}^{-1}$
$P$	- 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
- $\eta_H$  - hull efficiency
- $\eta_O$  - open water *propeller* efficiency
- $\eta_P$  - propulsive coefficient
- $\eta_R$  - relative rotative efficiency
- $\eta_S$  - gear and shaft efficiency
- $\lambda$  - 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