



Flanders
State of
the Art

19_012_1
FHR reports

Étude des conditions de navigation pour la traversée de Paris (VNF)

CFD computations of a barge at 90° drift
for 18 combinations of draft and under-keel clearance

DEPARTMENT
**MOBILITY &
PUBLIC
WORKS**

www.flandershydraulicsresearch.be

Étude des conditions de navigation pour la traversée de Paris (VNF)

CFD computations of a barge at 90° drift for 18 combinations of draft and under-keel clearance

Van Hoydonck, W.; Delefortrie, G.; Lemmens, M.; Mostaert, F.

Legal notice

Flanders Hydraulics Research is of the opinion that the information and positions in this report are substantiated by the available data and knowledge at the time of writing.

The positions taken in this report are those of Flanders Hydraulics Research and do not reflect necessarily the opinion of the Government of Flanders or any of its institutions.

Flanders Hydraulics Research nor any person or company acting on behalf of Flanders Hydraulics Research is responsible for any loss or damage arising from the use of the information in this report.

Copyright and citation

© The Government of Flanders, Department of Mobility and Public Works, Flanders Hydraulics Research, 2020
D/2020/3241/47

This publication should be cited as follows:

Van Hoydonck, W.; Delefortrie, G.; Lemmens, M.; Mostaert, F. (2020). Étude des conditions de navigation pour la traversée de Paris (VNF): CFD computations of a barge at 90° drift for 18 combinations of draft and under-keel clearance. Version 2.0. FHR Reports, 19_012_1. Flanders Hydraulics Research: Antwerp

Reproduction of and reference to this publication is authorised provided the source is acknowledged correctly.

Document identification

Customer:	Flanders Hydraulics Research	Ref.:	WL2020R19_012_1
Keywords (3-5):	Barge, shallow water, CFD, drift, UKC		
Fields of knowledge:	Manoeuvring behaviour > Influence under keel clearance > Numerical calculations		
Text (p.):	27	Appendices (p.):	10
Confidentiality:	No	☒ Available online	
Author(s):	Van Hoydonck, W.		

Control

	Name	Signature
Revisor(s):	Delefortrie, G.	Getekend door: Guillaume Delefortrie (Signi Getekend op: 2020-04-29 07:08:29 +01:00 Reden: Ik keur dit document goed 
Project leader:	Lemmens, M.	Getekend door: Mats Lemmens (Signature) Getekend op: 2020-04-29 08:35:28 +01:00 Reden: Ik keur dit document goed 

Approval

Head of division:	Mostaert, F.	Getekend door: Frank Mostaert (Signature) Getekend op: 2020-04-28 10:12:02 +01:00 Reden: Ik keur dit document goed 
-------------------	--------------	--



Abstract

The objective of this report is to compute forces on a barge with a drift angle of 90 degrees at six different drafts between 1 m and 4 m and three different under-keel clearances for a total of 18 conditions. For each of these conditions, a separate grid had to be constructed. The output is used to improve the mathematical model of barges and push convoys for a study for VNF in Paris. This report describes the barge geometry, the setup of the computations and some results.

Contents

Abstract	III
List of Figures	VI
List of Tables	VII
1 Geometry and Conditions.....	1
1.1 Introduction	1
1.2 Geometry.....	1
1.3 Conditions	2
1.3.1 Draft, UKC and waterlevels	2
1.3.2 Dimensionless quantities	3
1.3.3 Fluid properties	5
1.4 Initial domain	5
2 Preliminary computations	6
2.1 Initial setup.....	6
2.1.1 Mesh settings	6
2.1.2 Solver settings.....	7
2.1.3 Results	7
2.2 Improving the mesh.....	13
2.2.1 Improved grids	13
2.2.2 Flow visualisations.....	14
2.3 Grid convergence study.....	14
2.3.1 Flow visualisations: wave elevation	16
2.3.2 Convergence of forces and moments	16
2.3.3 Sinkage and trim	20
2.3.4 Conclusion	21
3 Computations	22
3.1 Introduction	22
3.2 Output conversion.....	22
3.3 Results	24
3.3.1 Forces and moments convergence	24
3.3.2 Motion convergence	24
3.3.3 Residual convergence	25
3.3.4 Comparison of average quantities	25
4 Conclusions	27
A1 Residuals of the grid convergence study	A1
A2 Graphs of final results	A2
A2.1 Time histories of forces and moments	A2
A2.2 Time histories of motions	A3
A2.2.1 Residual time histories	A3

List of Figures

Figure 1	Initial geometry of the barge as available in the towing tank database.....	1
Figure 2	Immersed volume in water (left) and longitudinal position of the centre of gravity (right) as a function of the draft.....	2
Figure 3	Y+ values on the submerged part of the hull.	8
Figure 4	Hydrodynamic pressure coefficient in the water fraction.	9
Figure 5	Water surface elevation w.r.t. the initial water level ($Z = 2.4$ m).	9
Figure 6	Relative velocity at $Z = 1.2$ m, which shows the wake behind the barge.	10
Figure 7	Detail of Fig. 6 near the bow including the cell distribution.	10
Figure 8	Velocity distribution in the water fraction in vertical planes 10 m apart.....	10
Figure 9	Streamlines showing the extend of the wake aft of the barge.	11
Figure 10	Convergence of the residuals.	11
Figure 11	Convergence of the forces and moments on the barge as a function of time: time signals (blue) and cumulative moving averages (orange).	12
Figure 12	Comparison of the water surface elevation for grids C14_8 (left) and C14_7 (right).	15
Figure 13	Comparison of the mass fraction for grids C14_8 (left) and C14_7 (right) in a vertical plane through midship showing the fluid interface directly upstream of the hull.	15
Figure 14	Wave elevation as a function of grid resolution.	17
Figure 15	Convergence of the hull forces and moments as a function of the grid size.	18
Figure 16	Time traces of forces and moments including the CMA for the <i>coarse 1</i> grid.....	18
Figure 17	Detail of Fig. 16.	19
Figure 18	Steady computation followed by an unsteady (second-order time integration) restart.	19
Figure 19	Frequency content of the steady and unsteady time traces of X from Fig. 18.	20
Figure 20	Solver settings related to the body motion with two solved motions (pitch and heave).	21
Figure 21	Time history of motions for coarse 1.....	21
Figure 22	Domain and global axis system convention used in FINE/Marine.	22
Figure 23	Time traces of the forces and moments for cases C15, C25 and C35.....	24
Figure 24	Time traces of the forces and moments for cases C15, C25 and C35.....	25
Figure 25	Averaged values of the forces, moments, sinkages and trim as a function of draft.	26
Figure 26	Dependence of the convergence of the hull forces and moments on the averaging interval length.	A1
Figure 27	Time traces of the forces and moments for cases C11, C12, C13, C14, C15 and C16.....	A2
Figure 28	Time traces of the forces and moments for cases C21, C22, C23, C24, C25 and C26.....	A3
Figure 29	Time traces of the forces and moments for cases C31, C32, C33, C34, C35 and C36.....	A4
Figure 30	Time traces of the motions for cases C11, C12, C13, C14, C15 and C16.	A5
Figure 31	Time traces of the motions for cases C21, C22, C23, C24, C25 and C26.	A6
Figure 32	Time traces of the motions for cases C31, C32, C33, C34, C35 and C36.	A7
Figure 33	Time traces of the residuals for cases C11, C12, C13, C14, C15 and C16.....	A8
Figure 34	Time traces of the residuals for cases C21, C22, C23, C24, C25 and C26.....	A9
Figure 35	Time traces of the residuals for cases C31, C32, C33, C34, C35 and C36.....	A10

List of Tables

Table 1	Immersed volume in water Vol_w , longitudinal position of the centre of gravity x_{cg} with respect to the stern and midship location, displacement Δ and mass m as a function of the draft.	2
Table 2	Free surface water levels (m) for the two lowest UKC values as a function of draft (left) and UKC values for the highest waterlevel as a function of draft (right).	3
Table 3	Vertical translation of the hull as a function of the under-keel clearance and the draft.	3
Table 4	Computation identification as a function of draft and water depth.	3
Table 5	Depth-based Froude number as a function of draft and under-keel clearance.	4
Table 6	Tuck number as a function of draft and under-keel clearance.	4
Table 7	Fluid properties at 15 °C.	5
Table 8	Cell sizes as a function of refinement level	6
Table 9	Absolute values (mean and standard deviation) of resultant force and moment components on the barge.	12
Table 10	Relative contribution of friction (due to viscosity) to resultant forces and moments on the barge.	13
Table 11	Mesh sizes and average $Y+$ values on submerged part of hull bottom for the grid convergence study.	16
Table 12	Grid sizes (number of cells) for the 18 cases.	22

1 Geometry and Conditions

1.1 Introduction

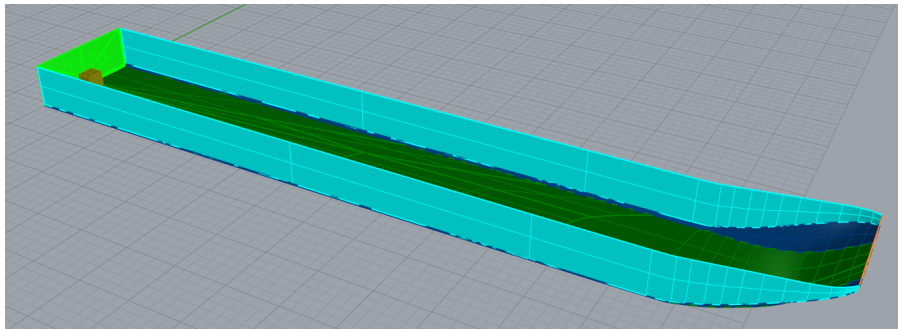
This chapter discusses the geometry of the barge and the conditions (draft, water depth) for which the simulations will be executed.

1.2 Geometry

The geometry of the barge is stored in the file format of Rhino in the database of the towing tank¹. Inspection of the CAD geometry (see Fig. 1) shows that this model was likely created in Delftship because it contains topological discretisations that are typical for that software package (due to the use of subdivision surfaces instead of NURBS).

The computations will be executed in FINE/Marine, which expects either a Parasolid or an STL as input. For the current project, a parasolid will be created by exporting the geometry to IGES format, importing that in CADFix for cleaning and exporting the resulting geometry as a parasolid file.

Figure 1 – Initial geometry of the barge as available in the towing tank database.



Inspection of the hull reveals that the rounding between the sides and the bottom of the hull (dark blue surfaces in Fig. 1) may give significant issues with the cleaning step in CADFix². For this reason, the geometry was modified in Rhino by creating new discretisations for some of the hull surfaces. In addition to fixing problematic surfaces to improve and speed up the cleaning process in CADFix, a deck was added to make the hull watertight. After cleaning the improved hull, it was found that the fillets of the small cubic recess at the lower back of the hull gave very low quality cells in HEXPRESS. To improve the grid quality, an alternative hull geometry was created with the cubic recess removed. Both models were added to the CFD CAD model repository in IGES, Rhino and Parasolid format³.

For each of the different draft values, the submerged hull volume and the longitudinal position of the centre of gravity (or centre of buoyancy) are computed. There is a linear relationship between the immersed volume and the draft (left graph of Fig. 2). The forward shift of the longitudinal position of the centre of gravity (shown in the right graph of Fig. 2) between the minimum draft of 1 m and the maximum draft of 4 m is approximately 1.7 m. For all draft values, the centre of gravity is located aft of midship. The numerical values are gathered in Table 1 including the displacement and mass values. The latter values were computed with domhydro⁴ and

¹https://wlsow.vlaanderen.be/shpgenerator/Lists/Lijnenplannen/bak_76m.3dm

²So much that it may not be possible to create a parasolid that can be loaded successfully in FINE/Marine.

³The model description page is located here: <https://wlwiki.vlaanderen.be/wiki/display/wlwiki/DOM>.

⁴domhydro is part of the FINE/Marine software suite.

are the mass values used in the body motion setup in the FINE/Marine computations. The mass is computed from the complete hull geometry, not only from the part that is submerged in water. For the largest draft (with the smallest volume immersed in air), this means that 441 kg of the total mass is due to displacement of air while for the smallest draft, 3505 kg of the total mass is due to air displacement. In relative numbers, these values add 0.05 % and 0.1 % to the mass due to submergence in water.

Figure 2 – Immersed volume in water (left) and longitudinal position of the centre of gravity (right) as a function of the draft.

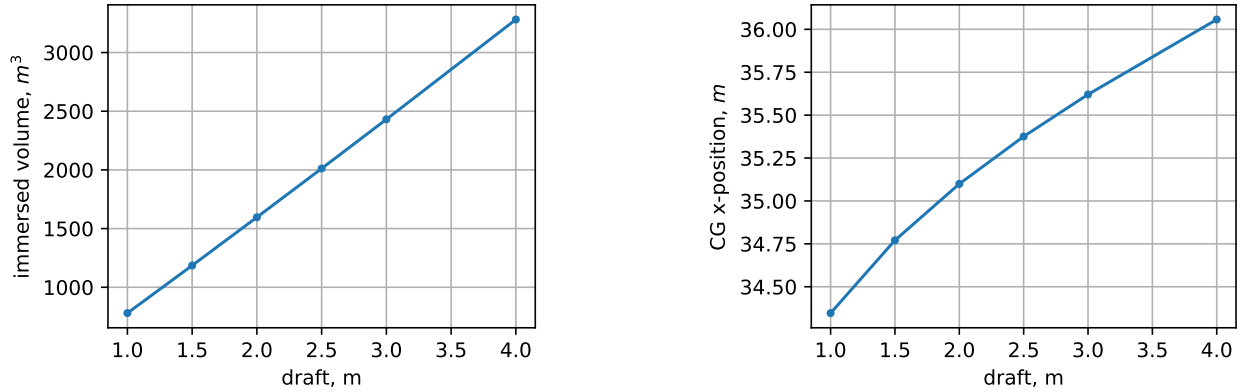


Table 1 – Immersed volume in water Vol_w , longitudinal position of the centre of gravity x_{cg} with respect to the stern and midship location, displacement Δ and mass m as a function of the draft.

T	Vol_w (m ³)	x_{cg} (m)	$x_{mid} - x_{cg}$ (m)	Δ (N)	m (kg)
1	$7.798\,880 \times 10^2$	34.346 53	−3.953 47	7 643 835	3 279 000
1.5	$1.185\,460 \times 10^3$	34.770 82	−3.529 18	11 618 926	2 430 324
2	$1.596\,299 \times 10^3$	35.099 27	−3.200 73	15 645 640	2 011 806
2.5	$2.011\,614 \times 10^3$	35.375 56	−2.924 44	19 716 224	1 597 372
3	$2.431\,022 \times 10^3$	35.620 42	−2.679 58	23 826 924	1 187 405
4	$3.281\,504 \times 10^3$	36.057 17	−2.242 83	32 162 665	782 693.5

1.3 Conditions

1.3.1 Draft, UKC and waterlevels

The mathematical model of the barge needs to be enhanced for low draft values. At the laboratory, a similar barge has only been tested in the towing tank with drafts of 3 m and 4 m. For the study of VNF in Paris, drafts will be lower at 2.5 m or less.

Computations will be executed with a drift angle β equal to 90° and a velocity V_m equal to 0.12 m/s at model scale 1/25, which corresponds to a velocity at prototype scale $V_p = 0.6$ m/s.

Computations are executed for six draft values (4 m, 3 m, 2.5 m, 2 m, 1.5 m and 1 m) and three water levels expressed in terms of the under-keel clearance for shallow cases (20 % and 50 %) and a fixed water level of 8 m for deep water.

The combination of six draft values and three under keel clearances gives a total of 18 conditions. For the shallow water cases, the free surface water levels are shown on the left hand side of Table 2 as a function of the draft and UKC values. For the deep water cases (shown on the right in Table 2, the water level is fixed and the UKC values vary as a function of the draft values.

Table 2 – Free surface water levels (m) for the two lowest UKC values as a function of draft (left) and UKC values for the highest waterlevel as a function of draft (right).

Draft, m \ UKC, %	UKC, %		Draft, m \ UKC, %	UKC, %
	20	50		
4.0	4.8	6.0	4.0	100
3.0	3.6	4.5	3.0	$\frac{500}{3} \approx 166.7$
2.5	3.0	3.75	2.5	220
2.0	2.4	3.0	2.0	300
1.5	1.8	2.25	1.5	$\frac{13}{3} \approx 433.3$
1.0	1.2	1.5	1.0	700

Using these values of water levels and drafts, the vertical translation of the hull to position it properly in the domain can be computed by subtracting the draft from the water level for each computation.

Table 3 – Vertical translation of the hull as a function of the under-keel clearance and the draft.

Draft, m \ UKC, %	UKC, %		
	20	50	deep
4.0	0.8	2	4
3.0	0.6	1.5	5
2.5	0.5	1.25	5.5
2.0	0.4	1	6
1.5	0.3	0.75	6.5
1.0	0.2	0.5	7.

A naming system is used to identify the different computations using the position of the computations in Table 3. These are shown in Table 4.

Table 4 – Computation identification as a function of draft and water depth.

Draft, m \ UKC, %	UKC, %		
	20	50	deep
4.0	C11	C21	C31
3.0	C12	C22	C32
2.5	C13	C23	C33
2.0	C14	C24	C34
1.5	C15	C25	C35
1.0	C16	C26	C36

1.3.2 Dimensionless quantities

At prototype scale, the length of the barge equals 76.59 m, which corresponds to a length of 3.0636 m at model scale. The database of the towing tank gives a length of 3.0640 m for both the L_{PP} and L_{OA} , a negligible difference of 0.4 mm. Midship is located halfway between the bow and the stern, 1.5320 m from the stern at model scale. At prototype scale, midship is located 38.3 m from the stern. The total height of the CAD model at full scale equals 5.854 m and its depth is 4.3 m. The highest point of the bow protrudes 1.549 m above the deck. The largest draft value in Table 2 is thus only 30 cm less than the depth of the hull.

The full scale and model scale Reynolds numbers based on barge length are 0.323×10^6 and 40.35×10^6 , respectively. The Froude number based on barge length equals 0.022. Due to the drift angle of 90° at which the computations will be executed, the breadth of the vessel may be a more appropriate reference length for these dimensionless quantities. At model scale, it is 0.456 m which results in a prototype scale value of 11.4 m. For the Froude number, a larger value is obtained: $Fr_B = 0.057$, while for the Reynolds numbers, a significant reduction is found: 0.0481×10^6 and 6.0×10^6 for model and prototype scale, respectively. The model scale Reynolds number is close to the lower boundary of the Reynolds number range in which boundary layers naturally transition from laminar to turbulent flow. Due to the blunt shape of the barge at 90° drift, laminar flow separation might occur in combination with transition to turbulent flow at model scale. These physical phenomena may make model scale CFD computations more difficult because it requires a good transition model. For this reason, computations will be executed at prototype scale, where the flow in the boundary layer is expected to be fully turbulent by the time separation occurs.

In shallow water research, additional parameters are used to define the ship condition: these are the depth-based Froude number and the Tuck parameter. The former is defined as

$$F_{nh} = \frac{u}{\sqrt{gh}}, \quad (1)$$

where g is the acceleration due to gravity and h is the water depth. The Tuck-parameter in turn is defined as

$$Tu = \frac{F_{nh}^2}{\sqrt{1 - F_{nh}^2}}. \quad (2)$$

The values for F_{nh} and Tu for the 18 conditions are given in Tables 5 and 6, respectively.

Table 5 – Depth-based Froude number as a function of draft and under-keel clearance.

Draft, m \ UKC, %	20	50	deep
4.0	0.08743718	0.07820619	0.06772855
3.0	0.10096376	0.09030473	0.06772855
2.5	0.11060025	0.09892387	0.06772855
2.0	0.12365484	0.11060025	0.06772855
1.5	0.14278431	0.12771017	0.06772855
1.0	0.17487435	0.15641238	0.06772855

Table 6 – Tuck number as a function of draft and under-keel clearance.

Draft, m \ UKC, %	20	50	deep
4.0	0.00767465	0.006135	0.00459771
3.0	0.01024604	0.0081884	0.00459771
2.5	0.01230793	0.00983417	0.00459771
2.0	0.01540878	0.01230793	0.00459771
1.5	0.02059841	0.01644454	0.00459771
1.0	0.03105965	0.0247697	0.00459771

1.3.3 Fluid properties

Two-phase flow is used with a free surface. The fluid properties of fresh water and air at 15 °C are used (see Table 7).

Table 7 – Fluid properties at 15 °C.

	density, kg/m ³	dynamic viscosity, Pa s	kinematic viscosity, m ² /s
air	1.225	1.7894×10^{-5}	1.46073×10^{-5}
water	999.1026	0.001 138	1.1386×10^{-6}

1.4 Initial domain

A single domain box will be used for all computations. This box must be large enough to accommodate the barge for the different combinations of water level and draft as documented in Table 2. The extreme case occurs with the highest water level and the smallest draft: computation C36 in Table 4. The width of the domain will be taken equal to the ratio of the model length to towing tank width, which means that the width at prototype scale must be 175 m. The length and height of the domain are chosen such that the initial grid can be constructed using (approximately) cubic cells. The inlet will be placed one L_{pp} upstream of the barge, this should be enough to ensure that the water surface rise ahead of the barge is not felt at the inlet. If not, the domain will be extended. Due to the bluff shape of the barge, the wake aft of the barge will extend aft of the body multiple times its length, especially for very low UKC values. Initially, the domain length aft of the barge will be set to approximately 160 m, such that the total domain length equals 250 m. If this length is found to be too short (either upstream or downstream of the barge) during the initial computations, the domain will be extended.

The height of the domain is set to 50 m. This gives the air that passes over the barge plenty of space. In summary, the initial domain size is 250 m × 175 m × 50 m.

2 Preliminary computations

In this chapter initial computations are executed to arrive at a suitable grid for case *C14*. Initially, focus is on the domain size, grid settings, solver settings and results. Once a suitable grid is constructed, a grid convergence study will be executed to verify that quantities converge as the grid is refined.

2.1 Initial setup

2.1.1 Mesh settings

The initial grid is kept as simple as possible, but should contain all features to capture the flow physics. These include a refinement surface to capture the air-water interface with enough detail, a refinement box to capture the separated wake aft of the barge and viscous layers on the hull and the bottom surface.

The initial Cartesian mesh contains 25, 18 and 10 cells in the three directions. In both horizontal directions, the initial cells have sides of approximately 10 m. Due to the relatively low water levels, the vertical cell size is halved (5 m).

The table with refinements as a function of the initial cell size is given in Table 8 relative to the initial cell size. The maximum number of refinements is set to 7, which will give cells with linear dimensions of 7.8, 7.8 and 4cm. All surfaces of the barge are activated for refinement. Those that make up the deck get three levels of refinement, while the large side surfaces are refined 5 times. The barge bottom is refined six times and the hull chamfers are refined maximally. A refinement surface is added at $Z = 2.4$ m that is maximally refined with non-zero target cells sizes of 11, 10 and 0.01m, a maximum aspect ratio of 100 and a refinement diffusion of 2. Of the domain surfaces, only the bottom is refined. Its settings are similar to the water surface refinement settings, only the target cell size in the Z-axis is set to 0.1 m.

A box refinement region is added in the wake of the barge in the water fraction with five (volumetric) refinements. It is as long as the barge and its width is twice the barge width.

Table 8 – Cell sizes as a function of refinement level

refinement level	cell size	refinement level	cell size
0	10.0	5	0.3125
1	5.0	6	0.156 25
2	2.5	7	0.078 125
3	1.25	8	0.039 062 5
4	0.625	9	0.019 531 25

For the initial computations, wall functions will be used on the barge and domain bottom surfaces. Initially, no extra viscous layer refinements will be added to the domain bottom surface. Based on the results of the Y^+ values on the bottom, viscous layers will be added to (part of) the domain bottom. For the barge, viscous layers are only added to surfaces that come in contact with water: the deck surfaces are excluded. With the reference velocity of 0.6 m/s, a reference length of 11.4 m and a target y^+ value of 50, the first layer thickness is 3.564×10^{-3} m. The appropriate number of layers for the hull surfaces vary between 10 and 17. After inserting viscous layers, the mesh contains 4.507×10^6 cells. Regarding mesh quality, the minimum orthogonality equals 16.12°, which is higher than the recommended minimum of 10°.

2.1.2 Solver settings

The solver is run in steady mode (first-order integration in time) with two fluids. Fully turbulent flow is assumed and the *k ω* -SST turbulence model is used. The gravitational acceleration is set to 9.81 m/s². The reference length and reference velocity are 11.4 m and 0.6 m/s, respectively. FINE/Marine reports that with these numbers, the Reynolds number in water equals 6×10^6 while the Froude number equals 0.056 74. These are the same values as those reported in section 1.3.2.

The interface position is set to 2.4 m and the initial velocity field is set to 0 m/s for the three velocity components.

Boundary conditions

All solid surfaces of the hull except for the two deck surfaces are treated as no-slip solid surfaces with wall functions. The deck surfaces are treated as slip walls. The domain bottom is also a solid wall with wall functions while for the lateral sides of the tank, a no-slip boundary condition is selected.

On the top and outlet patches, the prescribed pressure (updated hydrostatic pressure) boundary condition is used. At the inlet, a far field boundary condition is used with zero velocity values for all three components.

Body motion

The surfaces of the barge are grouped in an object (hull) to which extra parameters are assigned. For future computations, the yaw (cardan) angle is set to 90°. The motion in the X-direction is imposed with a 1/2 sinusoidal ramp that attains a velocity of 0.6 m/s in 5 s. For the first computations, the other degrees of freedom are all fixed. The reference point around which the forces and moments are computed is set on the waterline: Z = 2.4 m.

Mesh management

The domain rigid motion copies the rigid motion of the hull in the X-direction. The boundary conditions for grid deformation are set to *No-slip* for all domain surfaces except the bottom, where a *Bottom (shallow water)* boundary condition is used.

Control variables

A maximum of 8 non-linear iterations are executed for each time step and the convergence criterium is increased to 5 orders. The solution is saved every 100 time steps. In total, 5000 time steps are executed with a uniform time step value of 0.19 s.

2.1.3 Results

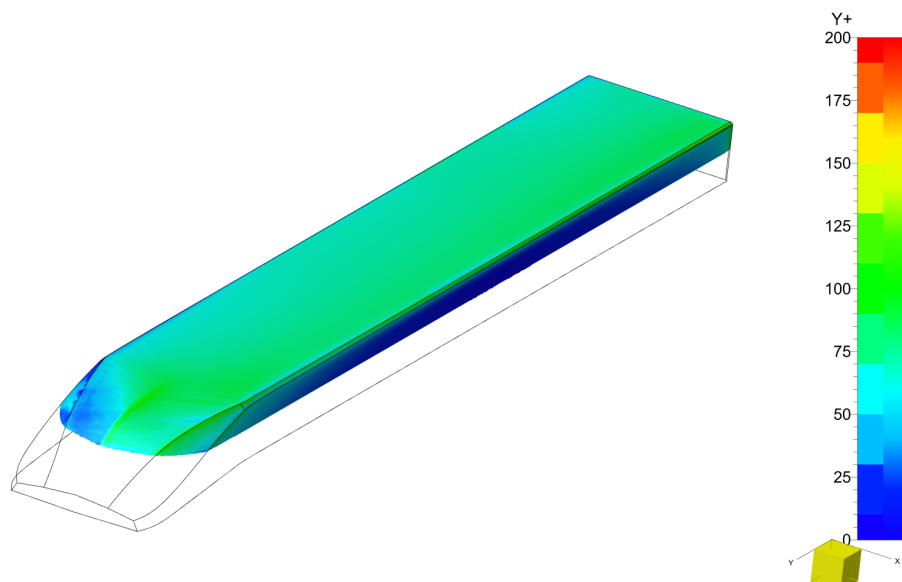
Various parameters and variables have to be checked to judge the grid. Amongst others, there are

- the Y+ values on the hull and domain bottom;
- the water surface elevation in the domain;
- the hydrodynamic pressure in the water fraction;
- unsteady phenomena such as flow separation and vortices in the wake aft of the barge;
- convergence of the total forces and moments on the hull, both the pressure and viscous components;
- relative magnitude of the pressure and viscous contributions to the forces;
- convergence of global residuals;
- convergence of local, cell-based residual values.

Y+ on hull and domain bottom

Due to oversight, the stern surface was not active when wall functions were added in the grid generation process. For the submerged part of this surface, the Y+ values are almost one order larger than the recommended upper boundary for wall functions (300). On the bottom, the average Y+ is close to 100, which is twice as high as the target Y+ value used to generate the viscous layers. This is caused by the increased velocity of the fluid below the hull as compared to the reference velocity used to estimate the height of the first cell in the boundary layer. The port side of the vessel (pointing away from the oncoming flow) has Y+ values that are on average between 10 and 20. By tripling the first layer height on this surface, the Y+ values could be increased to at least 30 although wall functions normally only work for attached flow conditions. The side surfaces above the water level have Y+ values close to 1. The deck has values in the correct range for wall functions.

Figure 3 – Y+ values on the submerged part of the hull.



Hydrodynamic pressure and water surface elevation

The hydrodynamic pressure coefficient in the water fraction is visualised in Fig. 4 and the water surface elevation with respect to the initial water level at $Z = 2.4$ m is shown in Fig. 5. With only one ship length between the barge and the inlet, the pressure field extends all the way to the inlet: the inlet is located too close to the barge. Inspection of the water level plot shows that the water level rise extends all the way to the inlet. Aft of the barge, there is a short region with significantly reduced water surface height. This region contains highly turbulent and unsteady flow. What is also clear for the water surface plot is that unlike OpenFOAM, FINE/Marine cannot modify the water level at the inlet during the course of the simulation due to a water level change upstream of the inlet.

Figure 4 – Hydrodynamic pressure coefficient in the water fraction.

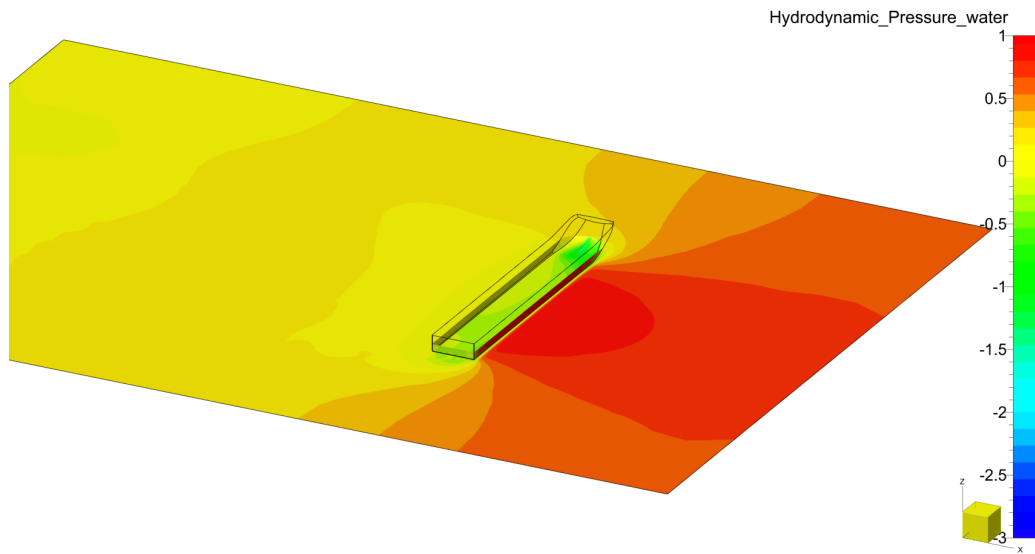
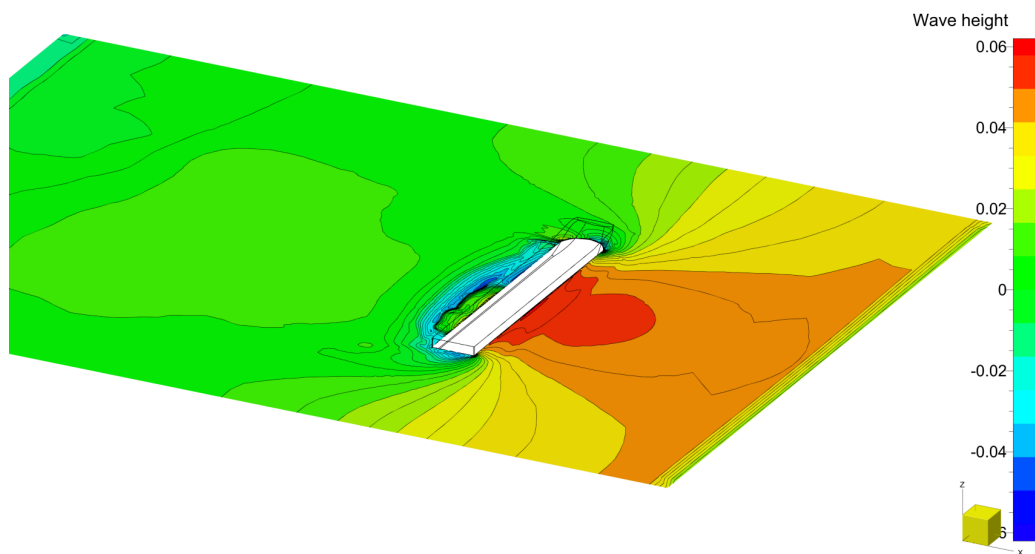


Figure 5 – Water surface elevation w.r.t. the initial water level ($Z = 2.4$ m).



Separating flow and barge wake

The wake region behind the barge is visualised in Fig. 6. This figure shows that the wake extends all the way to the outlet. In seven vertical cross sections spaced 10 m apart, the relative velocity before, below and aft of the hull is visualised (Fig. 8). The water that moves below the hull forms a jet that sticks to the bottom for two ship widths. The refinement box aft of the hull extends far enough downstream of the hull to capture a large part of this jet and the recirculating water on top of it (Fig. 9), but inspection of the grid at the stern of the hull in the water (Fig. 7) does show that it should be made wider: the water that separates from the edge between the stern and the starboard side loses information due to the grid coarsening (yellow region in Fig. 7).

Figure 6 – Relative velocity at $Z = 1.2$ m, which shows the wake behind the barge.

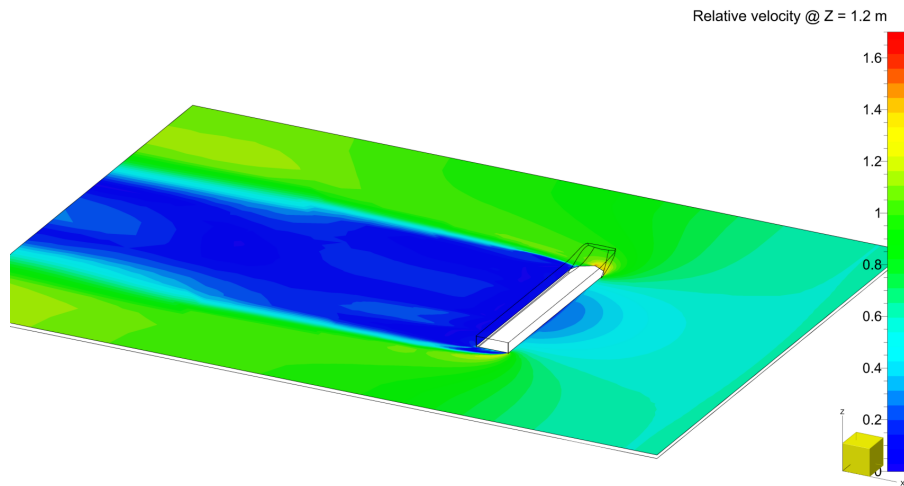


Figure 7 – Detail of Fig. 6 near the bow including the cell distribution.

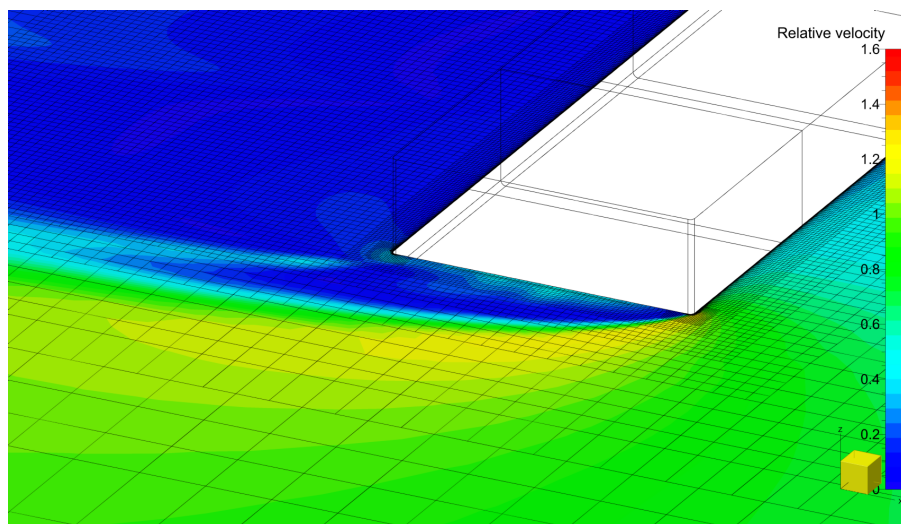


Figure 8 – Velocity distribution in the water fraction in vertical planes 10 m apart.

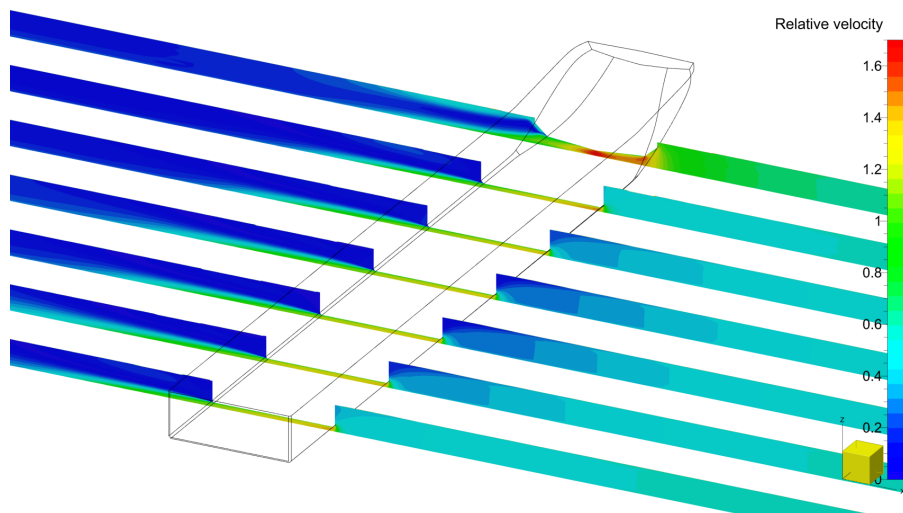
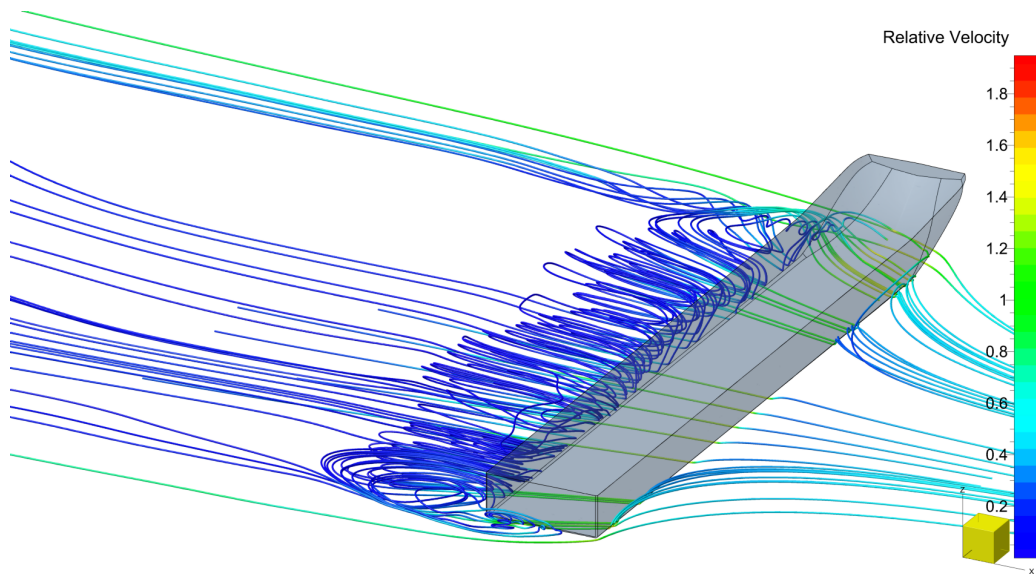


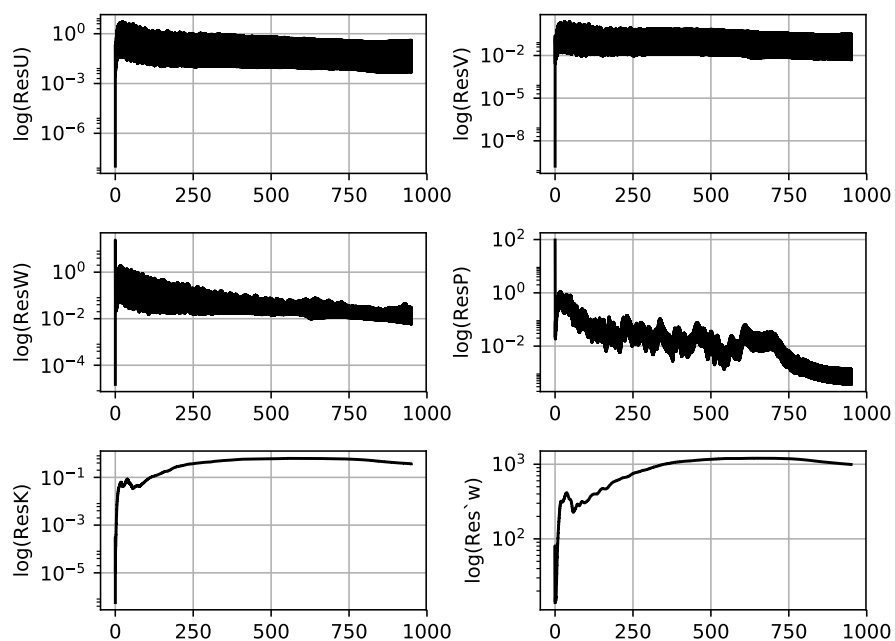
Figure 9 – Streamlines showing the extend of the wake aft of the barge.



Convergence of residuals and forces

The residuals of the computation are shown in Fig. 10. These graphs show that for the velocity components and the pressure, the convergence is not very good: residuals do not decrease much as the number of iterations is increased. These residuals also contains significant fluctuations that are related to the unsteady flow phenomena associated with separating flow in the wake of the barge. The residuals related to the turbulence quantities (K and ω - bottom two graphs) do not decrease at all.

Figure 10 – Convergence of the residuals.



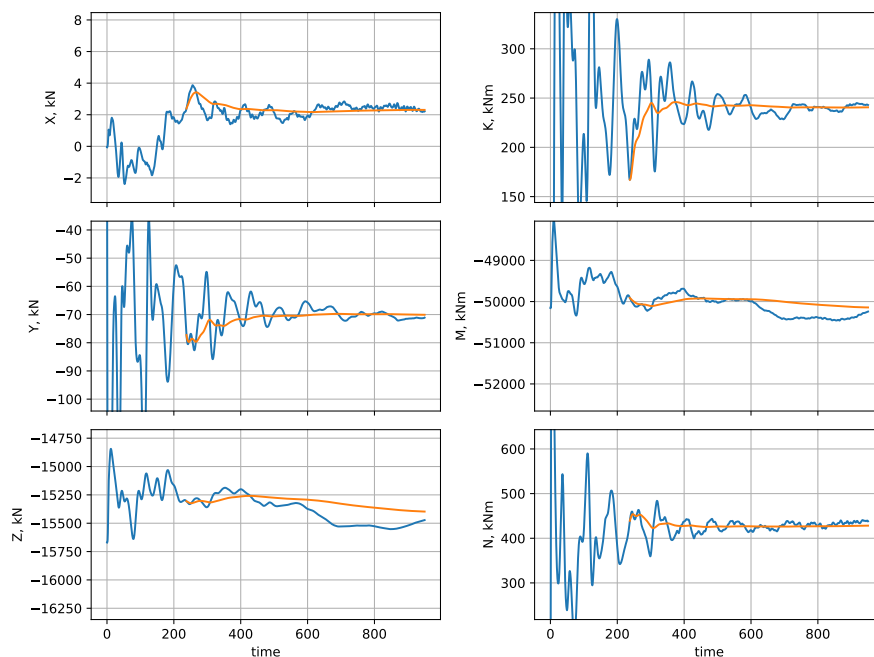
The forces on the barge due to pressure and friction are computed at the midship location on the waterline. Although convergence is slow, the fluctuations do subside as the plots in Fig. 11 show. The orange line shows the cumulative moving average (CMA) starting at 25% of the data or at 200 s, whichever is largest. Future

Table 9 – Absolute values (mean and standard deviation) of resultant force and moment components on the barge.

component	total		pressure		viscous	
	mean	std. dev.	mean	std. dev.	mean	std. dev.
X, kN	2.306	0.4161	2.172	0.4153	0.1344	0.005465
Y, kN	-70.08	4.194	-68.36	4.168	-1.723	0.05029
Z, kN	-15400.0	117.3	-15400.0	117.3	0.07581	0.002537
K, kNm	240.6	15.99	236.8	15.94	3.812	0.1094
M, kNm	-50140.0	231.8	-50140.0	231.8	0.785	0.02492
N, kNm	428.2	15.47	423.0	15.37	5.272	0.1749

computations that are executed in steady mode may have to be run for longer periods to minimise the influence of the low-frequency oscillations on the computation of the average values. Absolute values of the averages and standard deviations of all force components (in the ship-fixed axes system) are shown in Table 9 while the relative contribution of the viscous part to the total force is presented in Table 10. The vertical component of the force does not include the contribution of weight. The latter table shows that the maximum contribution of friction to the total force balance is less than 6 %. Friction contributes less than 3 % to the drag of the barge. The negative sign for the relative contribution of two of the force components is caused by the opposite sign of the viscous and pressure contributions. The contribution of friction to the total forces is very small which means that the use of wall functions is justified to keep the cell count as low as possible.

Figure 11 – Convergence of the forces and moments on the barge as a function of time: time signals (blue) and cumulative moving averages (orange).



Conclusions

Results of this exploratory computation of a barge in shallow and confined water at 90° drift shows massive regions of separated flow. Due to the unsteady nature of the flow, residuals of the steady computation do not show large reductions as the simulation progresses. The forces on the barge also show significant fluctuations that subside to an acceptable level. The mesh has to be improved and the simulation period must be increased.

Table 10 – Relative contribution of friction (due to viscosity) to resultant forces and moments on the barge.

component	viscous contribution, %
X	5.83
Y	2.46
Z	-0.000492
K	1.58
M	-0.00156
N	1.23

2.2 Improving the mesh

New computations have been configured to improve the results. After each improvement, a steady computation was launched and its results were analysed. Based on the insights gained, an improved grid is constructed. This iterative process will not be documented here in full detail, only an overview of the changes in the meshing settings will be given with possibly some visualisations of field variables and time histories of scalar quantities.

2.2.1 Improved grids

For the second grid (C14_4), the domain upstream of the barge is enlarged by $2 L_{pp}$ and some refinement boxes are added:

- Box # 0 (non-volumic, 6 refinements) around the hull to reduce cell sizes of submerged parts of the hull surfaces;
- Box # 1 (volumic, 5 refinements) below and aft of the hull to refine the water volume in the separation region;
- Sector # 2 (volumic, 5 refinements) at the barge stern where flow separates from the hull;
- Box # 3 (volumic, 3 refinements) in the water fraction.

Both the stern surface and the domain bottom were activated for insertion of viscous layers. For the domain bottom, the target $Y+$ size is set to 100. After insertion of viscous layers, the cell count is increased to 6.9×10^6 cells (from 4.5×10^6 cells in the initial grid).

For grid number three (C14_5), the domain size upstream of the barge was reduced by $1 L_{pp}$ with respect to the second grid. For this case, the domain length upstream and downstream of the barge is approximately the same. The global maximum number of refinements is increased to eight. For this case, the refinement sector was removed and the non-volumetric refinement box around the hull (Box # 0) was extended in all directions around the hull such that the domain bottom near the hull and in the wake of the barge is refined more. The volumetric refinement box (Box # 1) is extended upstream. The settings of the last box (Box # 3) are left unchanged. For this case, the cell count is approximately 9.1×10^6 . The domain size and extents of this case are used for all subsequent grids.

For the fourth grid (C14_6), an extra internal surface is added around the hull to resolve the water surface better. The maximum number of refinements is set to 10000, while the target cell sizes for this surface are 11 m, 11 m and 0.001 m. Refinement box # 1 is extended upwards to ensure that more uniform cells are created on the hull sides. This grid contains 9.3×10^6 cells.

For grid number five (C14_7), the maximum number of refinements for box # 3 is reduced to two (from three). The target cell sizes for the two internal surfaces at the water interface were sanitized as well: the target cell size (TCS) was set to 11 m, 11 m and 0.08 m, or approximately $L_{pp}/1000$ in the Z-direction. The maximum number of refinements was decreased to six. For the small internal surface around the hull, the TCS values

were set to 11 m, 11 m and 0.02 m and the maximum number of refinements for this surface was set to 10000. Its refinement diffusion is four, which is one higher than the refinement diffusion of the large water refinement surface. Viscous layer insertion parameters were not changed. This mesh contains 8.6×10^6 cells.

Grid number six (C14_8) is used for the grid convergence study. The small internal surface to refine the water surface was deactivated and the TCS Z-value for the large surface was halved (0.02 m). This gives a better resolution of the free surface in the complete domain, but less refinement near the hull. It removes unphysical water surface elevations at the edges of the small refinement surface. The cell count is further reduced to 7.8×10^6 .

2.2.2 Flow visualisations

For the last two grids, a side-by-side comparison of the water surface elevation is shown in Fig. 12. This shows the effect of the extra refinement in the Z-direction for the small surface around the hull: unphysical changes in the water surface elevation are visible near the edges of the small refinement surface. Despite the higher vertical resolution for C14_7, the interface is not as sharp upstream of the hull as it is for C14_8. This is likely caused by the lower vertical resolution at the water surface further upstream. For this reason, the extra refinement is disabled in all further computations. As an added bonus, the cell count is reduced with 0.8×10^6 cells.

Both figures show that the water surface elevation ahead of the vessel is increased over the full length of the domain. Aft of the hull, the wake extends until the outlet, the circular depressions at the edges of the wake downstream from the bow and stern are caused by low frequency oscillations of the wake where vertically-oriented vortices with low pressure cores move downstream.

2.3 Grid convergence study

Based on the settings for the last grid (C14_8), a grid convergence study is executed to determine the characteristics of various variables (both integral and field) as the grid is refined. Derived grids are constructed in such a way that the distribution of cells in the domain is similar to the base grid. This is achieved by resizing the cells in the initial Cartesian grid. For example, for the grid `coarse1`, the number of cells in the initial Cartesian grid are 26, 14 and 8. For comparison, the cell count in each direction for the original grid was 33, 18 and 10. The cell ratios are close to or exactly 1.25. This value was used to scale all absolute settings in the grid generation process such as the TCS for the water surface refinement. To obtain a similar mesh with viscous layers, special care must be taken with the viscous layer insertion process: the number of layers that are inserted depends on the target Y^+ value. If this value is not changed, the number of layers is modified and the grids are not similar any more. In the limit of a very fine grid, no viscous layers would be inserted at all. Hence, for a coarser grid, higher values for the target Y^+ value are used: multiplied with the above ratio for a coarser grid and divided by the scaling factor for a finer grid. This ensures that the number of viscous layers remains (approximately) the same.

For the current study, six additional grids are constructed, four coarser grids and two finer. The number of cells in each grid are shown in Table 11, together with the scaling factor used to construct the derived meshes, average Y^+ value on the submerged part of the bottom hull surface and the grid size g_i , defined as

$$g_i = \sqrt[3]{n_1/n_i}, \quad (3)$$

where n_1 is the number of cells in the finest mesh and n_i the number of cells in mesh i . In this table, the grid with id `fine1` is missing due to an issue encountered during grid optimization: a negative cell could not be removed from the grid which meant that with a scaling factor of 1.25, no valid grid could be generated. By lowering the scaling factor to 1.2, the negative cell was not encountered. This factor was then also used to construct the finest mesh. For the coarse grids, the scaling factor is $1/1.25 = 0.8$.

Figure 12 – Comparison of the water surface elevation for grids C14_8 (left) and C14_7 (right).

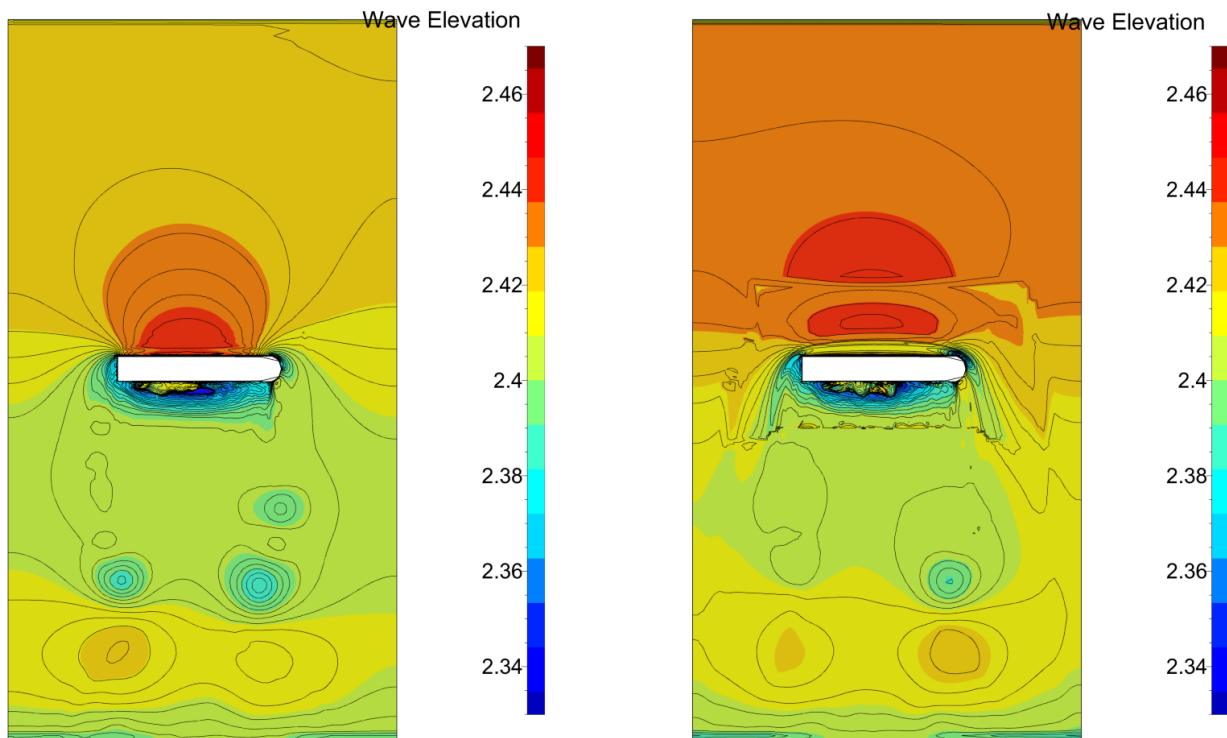


Figure 13 – Comparison of the mass fraction for grids C14_8 (left) and C14_7 (right) in a vertical plane through midship showing the fluid interface directly upstream of the hull.

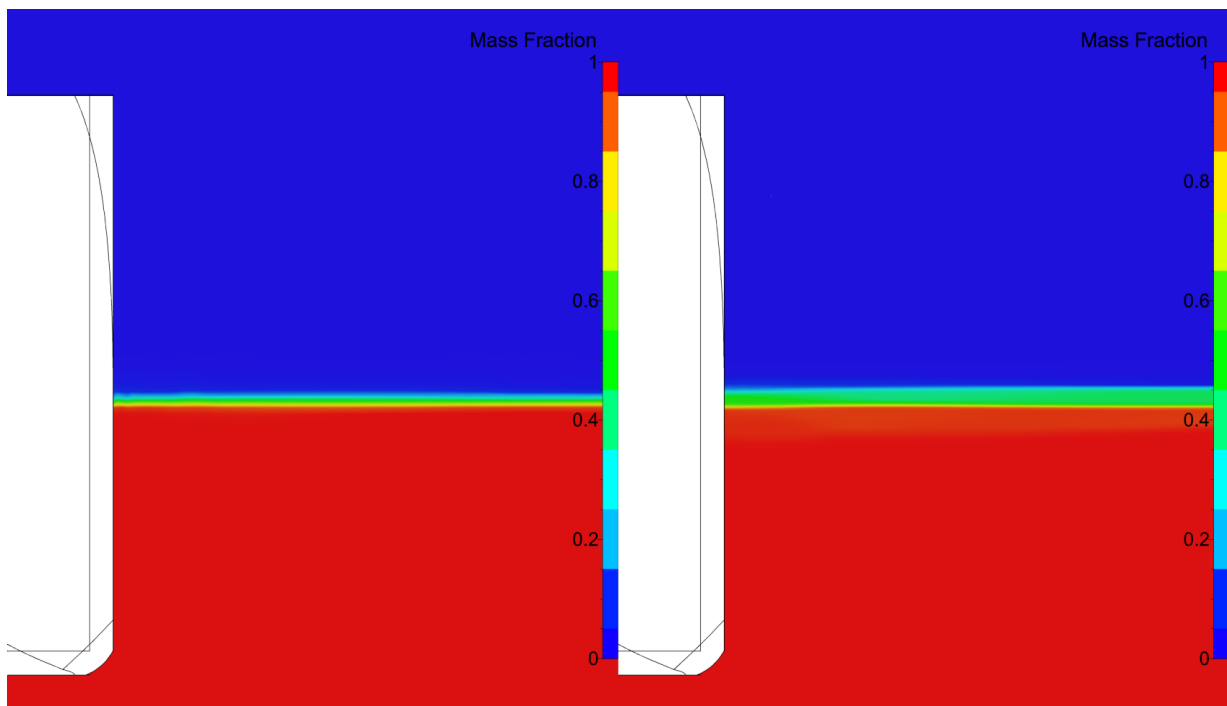


Table 11 – Mesh sizes and average Y^+ values on submerged part of hull bottom for the grid convergence study.

Grid Id	scaling factor	Number of cells n_c	grid size g_i	$\bar{Y}^+$
coarse4	0.8	1 739 581	2.04	167.3
coarse3	0.8	2 461 587	1.82	136.2
coarse2	0.8	3 355 235	1.64	107.9
coarse1	0.8	4 406 141	1.50	90
base	1	7 784 335	1.24	62.7
fine2	1.2	9 488 018	1.16	57.3
fine3	1.2	14 818 244	1.00	44.7

2.3.1 Flow visualisations: wave elevation

The wave elevation for the 7 grids are shown in Fig. 14. The water surface elevation on the coarsest grid (coarse 4) shows significant issues near the edges of the water volume refinement box. These issues are less visible for the second-coarsest grid, but still present. Starting from grid coarse 2, the isolines of water surface elevation are smooth. The differences between the results in the patterns and colours are due to the specific limits of the wave elevation field variable of each case. Including the solutions on the two coarsest grids in the grid convergence study may have a negative affect on the convergence characteristics. This will be checked in the next section, where the forces and moments on the hull as a function of the grid resolution are analysed.

2.3.2 Convergence of forces and moments

Forces and moments on the hull (resolved at the reference point in the ship-fixed axes system) are displayed as a function of the grid size g_i in Fig. 15. The graphs in this figure also show for each data point the standard deviation of the time signal that was used to compute the average value. A quadratic fit (orange) is added to show global data trends. Averages were computed using the last 75 % of the complete time signal. Except for the yawing moment N , the magnitude of the standard deviation is larger than the difference between the maximum and minimum value of the computed average. Hence, the fluctuations caused by the unsteady flow separation are larger than the changes in the forces and moments due to different grid sizes. Also, the averages vary with the length of the averaging interval. Similar graphs are shown in the APPENDIX using different lengths for the averaging interval. This averaging length should be as long as possible but it should not include the initial transients due to the start-up of the computation. Inspecting the time signals makes it clear that as time progresses, the amplitude of low-frequency oscillations increases (see Fig. 16). As a consequence, the cumulative moving averages show oscillations that do not subside with time (as visible in Fig. 17). Despite the already long computation interval (10 000 time steps of length 0.19 s, one could conclude that the computations need to be run longer to get a stable average. On the other hand, the physics of the problem is mainly non-stationary: the oscillations in the time traces of the forces and moments are likely caused by flow separating from the barge hull. One can conclude that it is more important to ensure that simulations are executed for a long time period than that simulations on a very fine grid are executed.

A restart using second-order time integration shows that the low-frequency fluctuations have a fixed period and the resultant forces and moments show stable fluctuations around a mean value (Fig. 18). A comparison of the signals for the longitudinal force component X (Figs. 18 and 19) shows that the time trace of the unsteady computation converges better towards a stable mean (lower amplitude for the lowest frequency). The dominant component for the unsteady computation has a frequency of 0.005 Hz, while for the steady computation, it is slightly higher than 0.01 Hz.

Figure 14 – Wave elevation as a function of grid resolution.

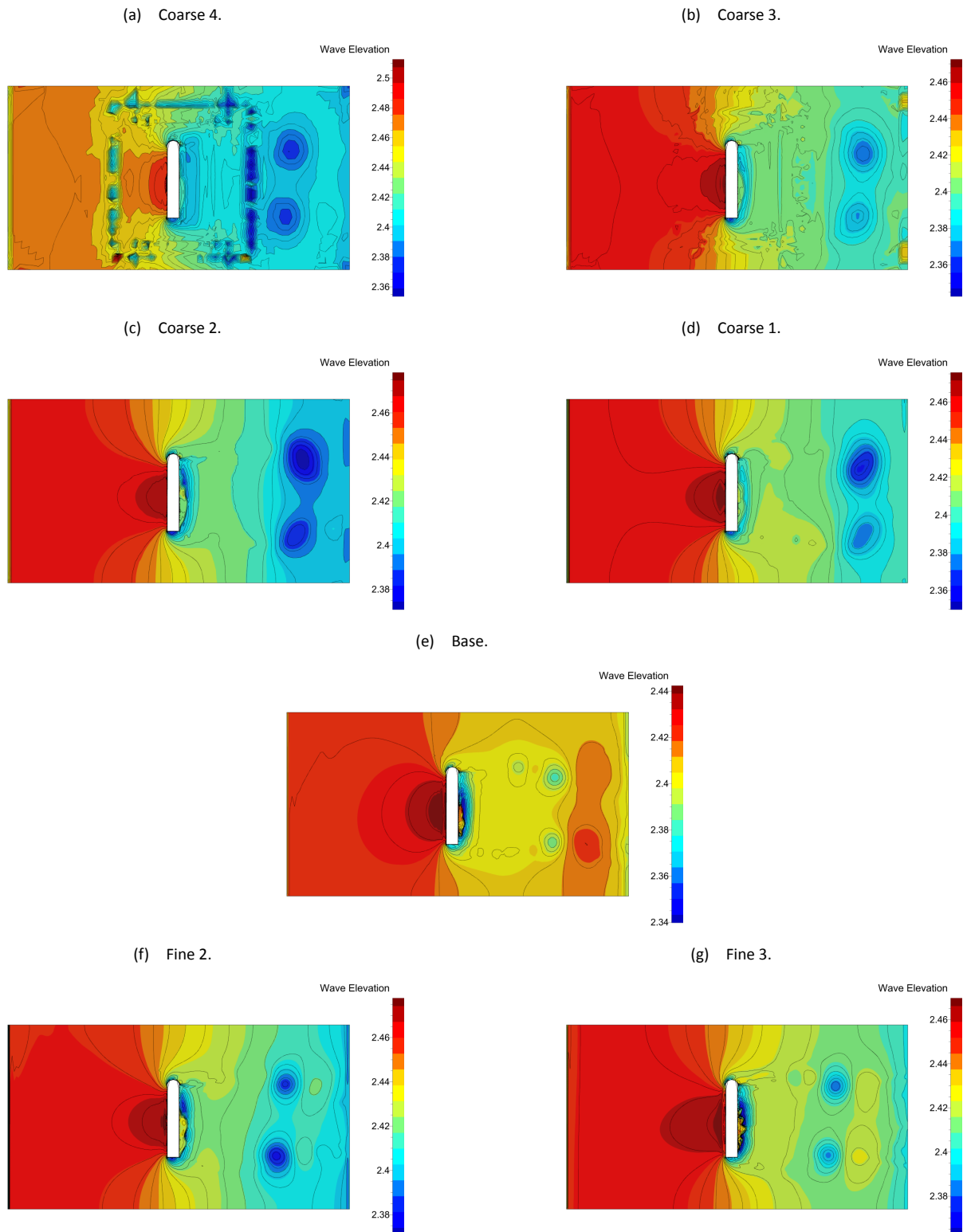


Figure 15 – Convergence of the hull forces and moments as a function of the grid size. Quadratic fits of the data (orange) show global trends.

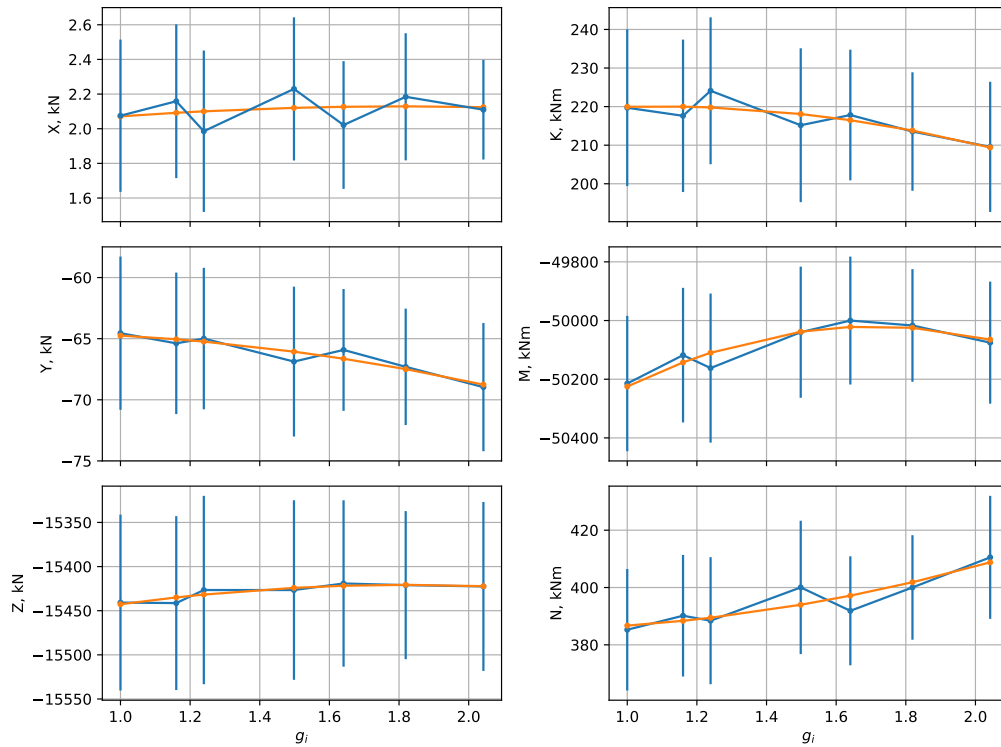


Figure 16 – Time traces of forces and moments including the CMA for the *coarse 1* grid.

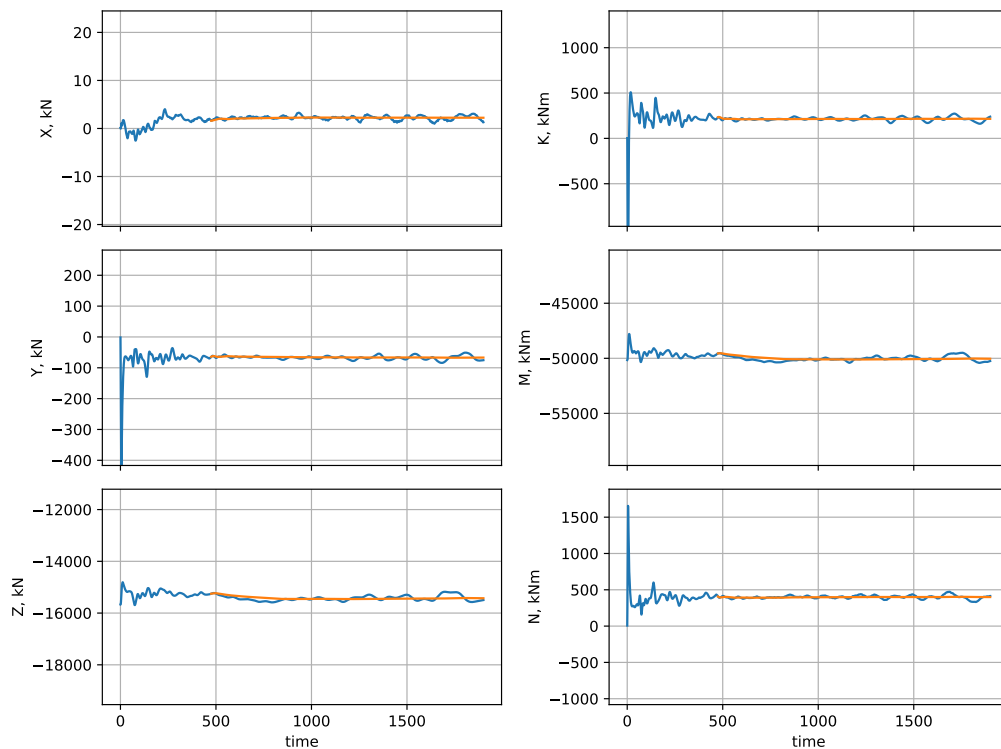


Figure 17 – Detail of Fig. 16.

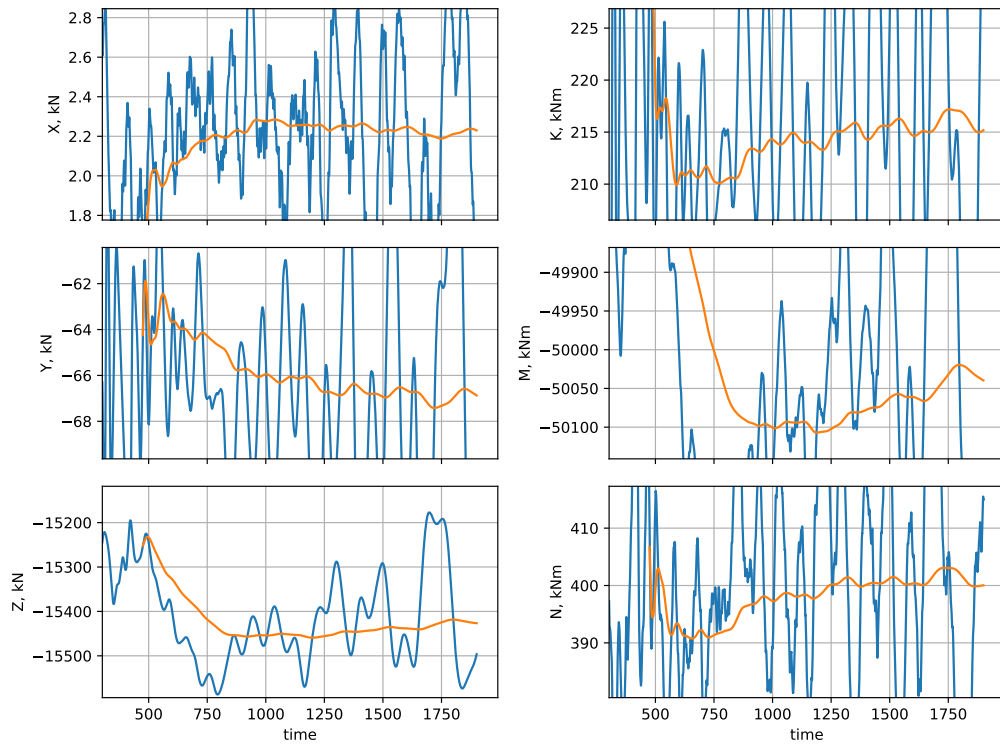


Figure 18 – Steady computation followed by an unsteady (second-order time integration) restart. The unsteady computation converges towards a stable fluctuation around a steady mean.

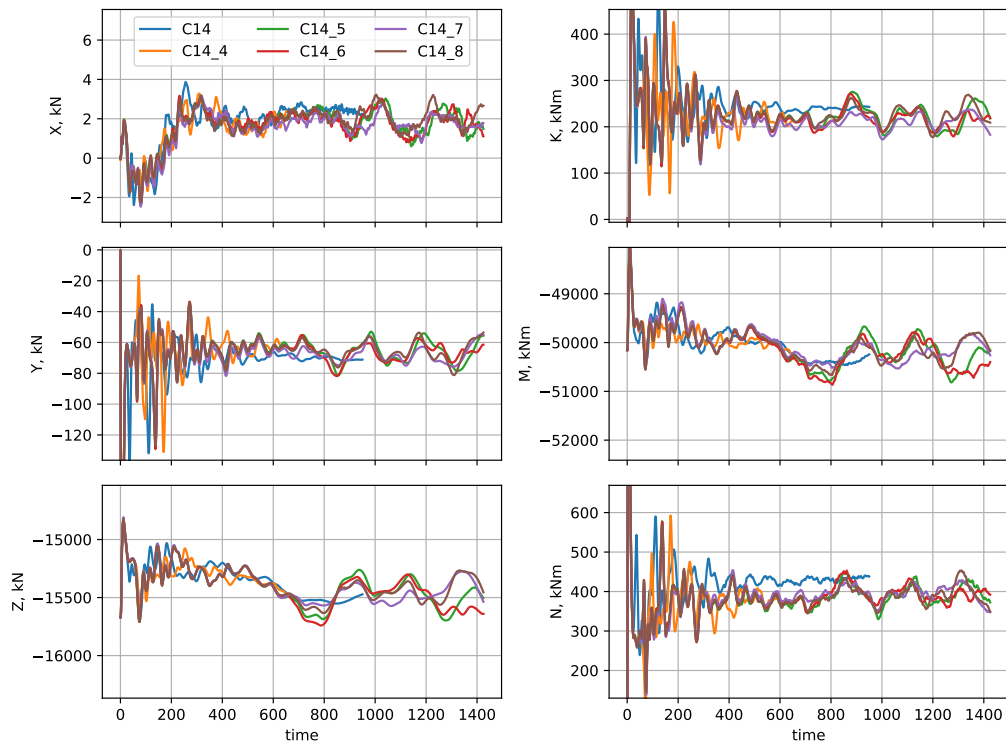
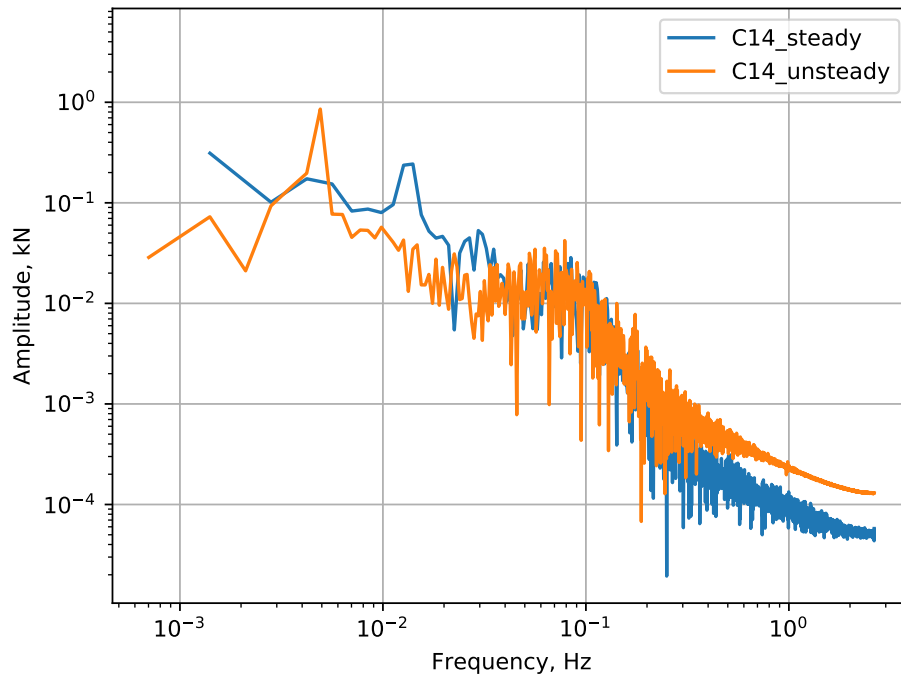


Figure 19 – Frequency content of the steady and unsteady time traces of X from Fig. 18.



2.3.3 Sinkage and trim

One of the goals of the computations is to extend the existing mathematical model of a barge for use in the simulator. In towing tank tests, the sinkage and trim motions are normally left free. For the computations, these are also released and solved for (although only a steady state is searched, using the *Quasi-steady* approach method available in FINE/Marine), see Fig. 20. Due to the drift angle, the *cardan angle* option must be activated (heading/yaw angle set to $\pi/2$), this ensures that the X-axis of the body-fixed frame points towards the bow (green dashed axes system in Fig. 20). The linear degrees of freedom are defined in the (global) axes system attached to the fluid domain, while the angular degrees of freedom are defined in the local axes system. Hence, the $Tx0$ (Surge) is imposed, while $Tz0$ (Heave) and $Ry1$ (Pitch) are solved for.

For the mesh management, the mesh displacement copies the rigid motion of the hull for $Tx0$, while for $Tz0$ and $Ry0$, a weighted deformation is used. The boundary conditions for grid deformation are all set to no-slip except for the domain bottom, where the *bottom (shallow water)* grid deformation boundary condition is selected.

Starting from the final result of a steady computation without motion, the forces, moments and motions converge towards fairly stable fluctuating motions with reasonably constant amplitude and frequency, see Fig. 21.

For the current case, the sinkage is approximately 0.03 m while the pitch (bow up positive) converges to 0.005°. At model scale, the computed sinkage corresponds to 1.2 mm. This pitch angle means that the sinkage at the stern is approximately 0.033 m. With 0.4 m space below the hull, the reduction at the stern is less than 10 %.

Figure 20 – Solver settings related to the body motion with two solved motions (pitch and heave).

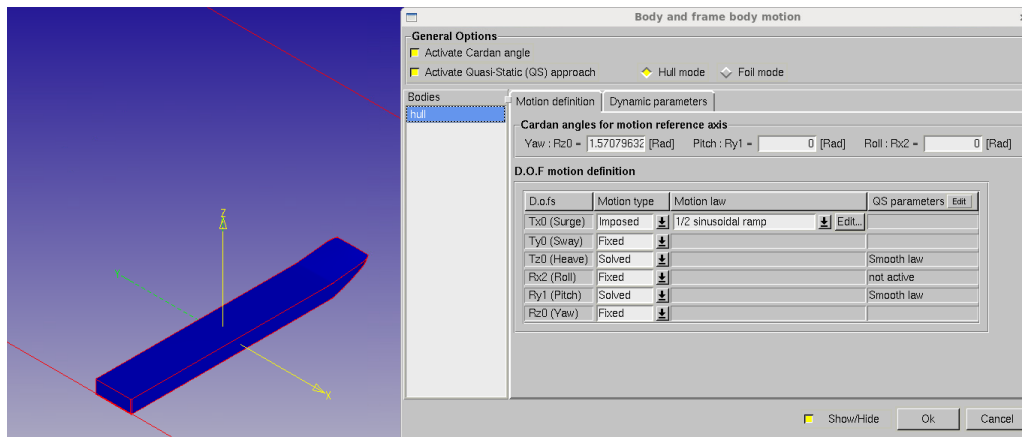
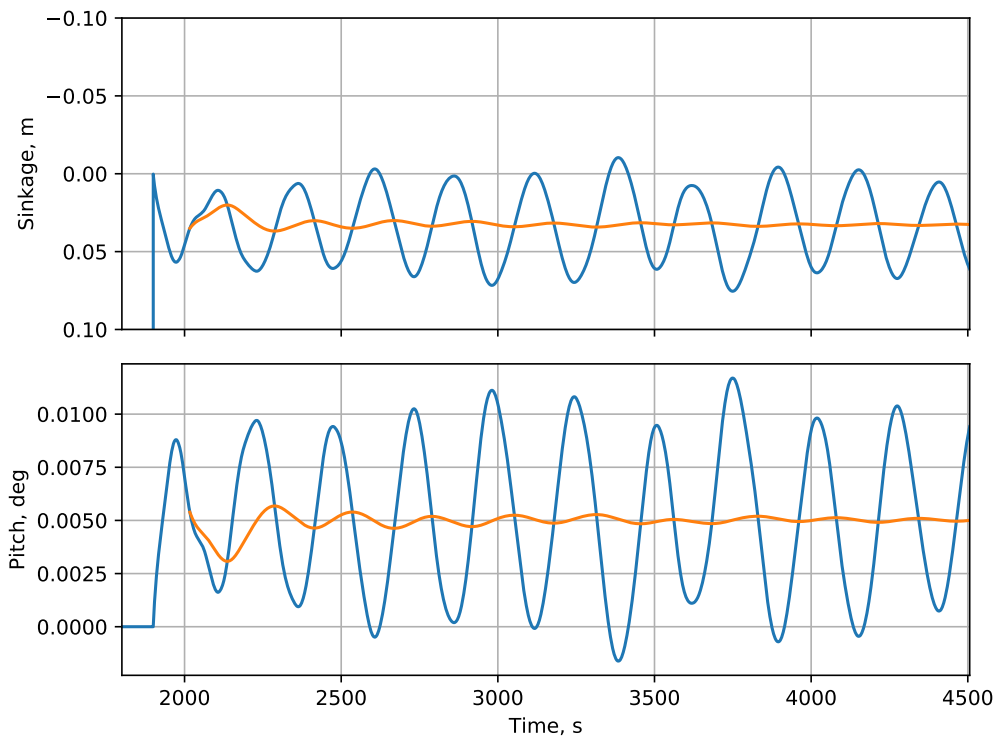


Figure 21 – Time history of motions for coarse 1.



2.3.4 Conclusion

It is decided to use meshes for all cases based on the coarse1 mesh. This means that computations can be run for a longer time (say 20 000 iterations instead of 10 000 iterations) without increasing the wall clock time significantly. It should be noted that due to the mesh deformation as a consequence of the free heave and pitch motions, the time step is halved by default by the GUI of FINE/Marine. This means that the actual time period that is simulated is halved if the total number of iterations is kept the same. Hence, 20 000 iterations with mesh deformation corresponds to 10 000 iterations without mesh deformation. Thus, to simulate a longer time period with mesh deformation, more than 20 000 time steps should be simulated.

3 Computations

3.1 Introduction

The initial computations have been used to fine-tune the mesh and solver settings to arrive at a state from which an average can be extracted (mainly the minimum number of iterations). Table 12 contains an overview of the grid sizes for each of the 18 cases.

On average, the computing time for 40 000 iterations is 3306 min, which corresponds to two days and seven hours. The majority of the computations were run on 128 cores of the stokes queue of the FHR linux cluster.

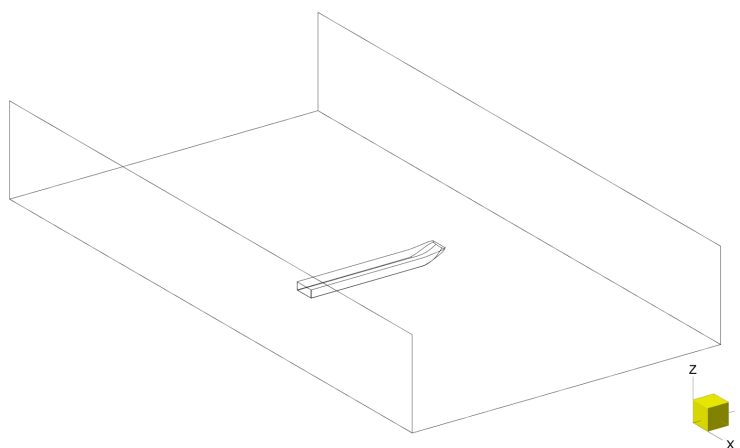
Table 12 – Grid sizes (number of cells) for the 18 cases.

	C1?	C2?	C3?
C?1	4 671 669	5 037 103	5 317 880
C?2	4 623 848	5 044 027	5 642 108
C?3	4 546 358	4 893 116	5 493 442
C?4	4 406 141	4 705 962	5 423 526
C?5	4 331 179	4 563 994	5 723 296
C?6	5 498 549	4 306 670	5 760 045

3.2 Output conversion

In FINE/Marine, whenever there is a solved motion active, the forces on a body are computed at the centre of gravity. Furthermore, the global axis system is such that the positive x-axis points forward (in the direction of motion of the ship) and the y-axis points to port. As a consequence, the z-axis points upwards (Fig. 22). The standard body-fixed axis system used in naval architecture (and aerospace engineering) is such that the z-axis points down and the y-axis points to starboard. This means that except for the longitudinal force and roll moment, all forces and moments must change sign.

Figure 22 – Domain and global axis system convention used in FINE/Marine.



For the mathematical modelling at FHR, the forces and moments must be known at the midship location. Also,

the hydrostatic force on the hull (that balances the weight when the ship is at rest) should be subtracted from the total force in the Z-direction. The hydrostatic force is computed separately from the mass and displaced volume.

Due to the 90° drift angle of the barge, the domain X-axis points towards the starboard side of the barge. The domain Y-axis points towards the bow and the Z-axis points upwards. In a first step, the forces are converted to a body-fixed axes system:

$$X_{bf} = F_y, \quad (4)$$

$$Y_{bf} = F_x, \quad (5)$$

$$Z_{bf} = -F_z, \quad (6)$$

$$K_{bf} = M_y, \quad (7)$$

$$M_{bf} = M_x, \quad (8)$$

$$N_{bf} = -M_z. \quad (9)$$

The hydrodynamic force on the hull is computed using the body mass m and the acceleration of gravity g (9.81 m/s^2). The former one was computed for each draft using the hydrostatic tool of FINE/Marine domhydro (last column in Table 1). These forces apply at the centre of gravity of the barge which is located aft of the midship location. The forces at the midship location are the same as the intermediate forces:

$$X_m = X_{bf}, \quad (10)$$

$$Y_m = Y_{bf}, \quad (11)$$

$$Z_m = Z_{bf}. \quad (12)$$

For the moments, the offset in the x-direction means that both the pitch moment and the yaw moment have a contribution due the vertical force and the lateral force, respectively. Hence,

$$K_m = K_{bf} \quad (13)$$

$$M_m = M_{bf} - Z_m \cdot x_{cg}, \quad (14)$$

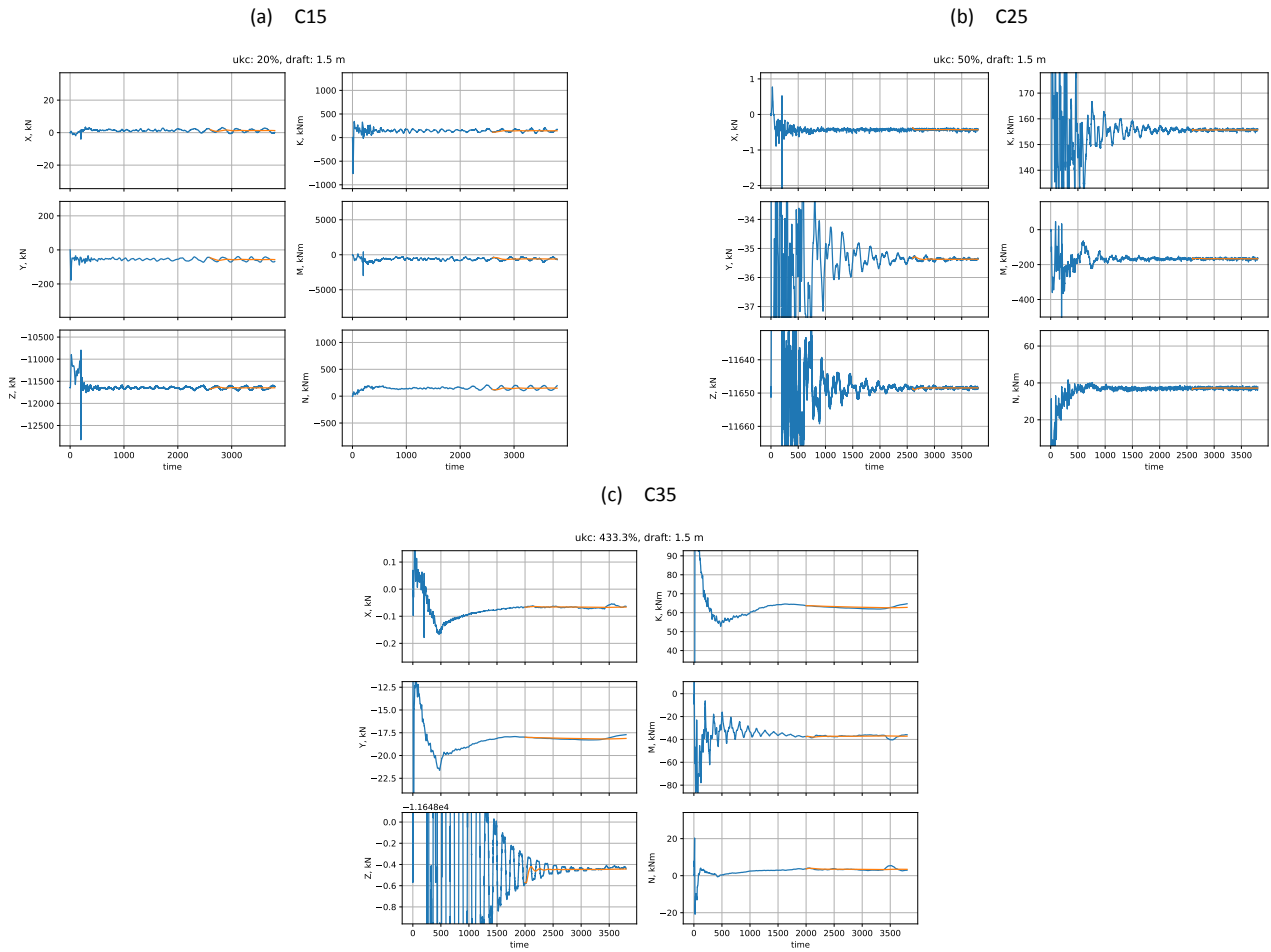
$$N_m = N_{bf} - Y_m \cdot x_{cg}, \quad (15)$$

where x_{cg} is the longitudinal offset of the centre of gravity with respect to the midship location. For the barge, it is a negative value roughly between -4 m and -2 m .

After conversion to the midship location, the average of the forces and moments is taken using a sufficient number of samples. Standard deviations (as a measure for the fluctuations in the time trace) will be shown as well.

For the motions, the pitch motion is negated to get a value that is positive bow-up. The vertical motion is negated as well, such that sinkage is positive when the UKC is reduced. The averages of the sinkage and pitch are computed with the same method as used for the forces and moments.

Figure 23 – Time traces of the forces and moments for cases C15, C25 and C35.



3.3 Results

3.3.1 Forces and moments convergence

For a draft of 1.5 m, the time histories of the forces and moments at the centre of gravity in the body-fixed frame of reference are shown in Fig. 23. Apart from the full time trace (in blue), these figures show in orange the cumulative moving average (CMA) to judge convergence behaviour of the time trace. The last value of the CMA time trace is the average value of the time period over which the CMA was computed. All three cases converge towards a steady mean, although for C35, convergence might improve if the simulation period would be extended somewhat. The fluctuations that are present in the converged results of C15 indicated that time-dependent phenomena occur (such as flow separating periodically from the barge) for this case. With reduced UKC values, the amplitude of the fluctuations increases. For the other cases, the time traces of the forces and moments are shown in Appendix A2.1.

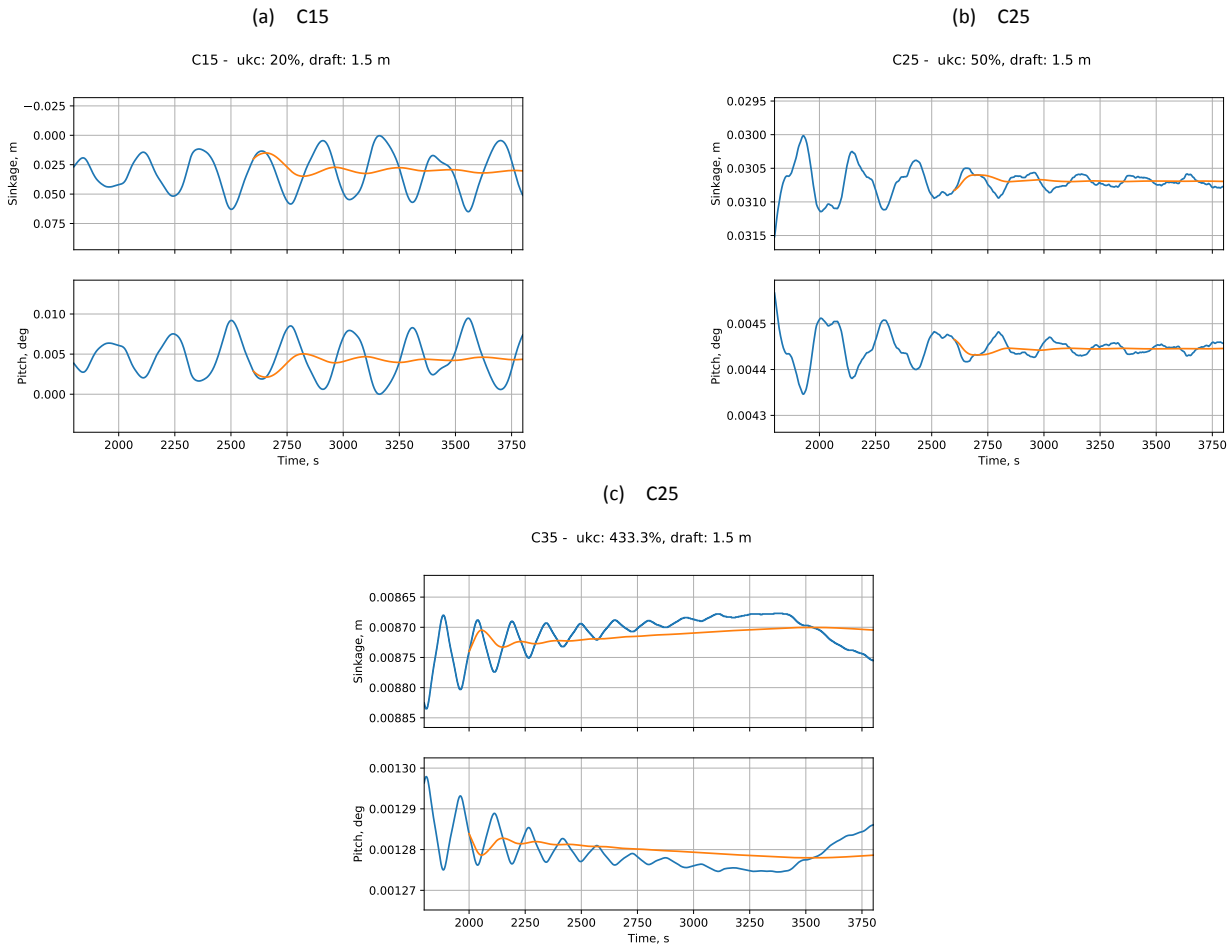
3.3.2 Motion convergence

For the same three computations, the time histories of the motions (sinkage and trim) are shown in Fig. 24. For computation C15 (Fig. 24a) the period of the low-frequency oscillations is close to 250 s, for the other two computations, the period is similar. The shape of the graphs of the pitch and sinkage is exactly the same. This appears to be the case for each computation. Since Newton's second law is not solved to change the pitch and sinkage between iterations the time histories themselves are not relevant. The similarity between the graphs could be explained by assuming that the algorithm to update the degrees of freedom works similar for each

degree of freedom.

All graphs are shown again in Appendix A2.2. Overall, either the graphs converge to a fixed value, or they keep oscillating around a fairly steady mean. Some of the graphs (such as C34 and C35) show divergent behaviour for the sinkage and trim, but it should be noted that the magnitude of the divergence is very small. These computations might settle to a (slightly) different sinkage and or trim if more iterations were executed.

Figure 24 – Time traces of the forces and moments for cases C15, C25 and C35.



3.3.3 Residual convergence

The residuals of all computations are shown in Appendix A2.2.1. From these graphs, it seems that for cases with low UKC, the curves flatten faster. The residuals for cases with higher UKC values seem to take longer to converge, but they do reach lower values. Based on the residuals of ω , one could argue that the computations with intermediate and high UKC values should be run for more time to allow the residuals of all variables to completely flatten out. Overall, convergence is judged to be sufficient to extract averages from the time traces.

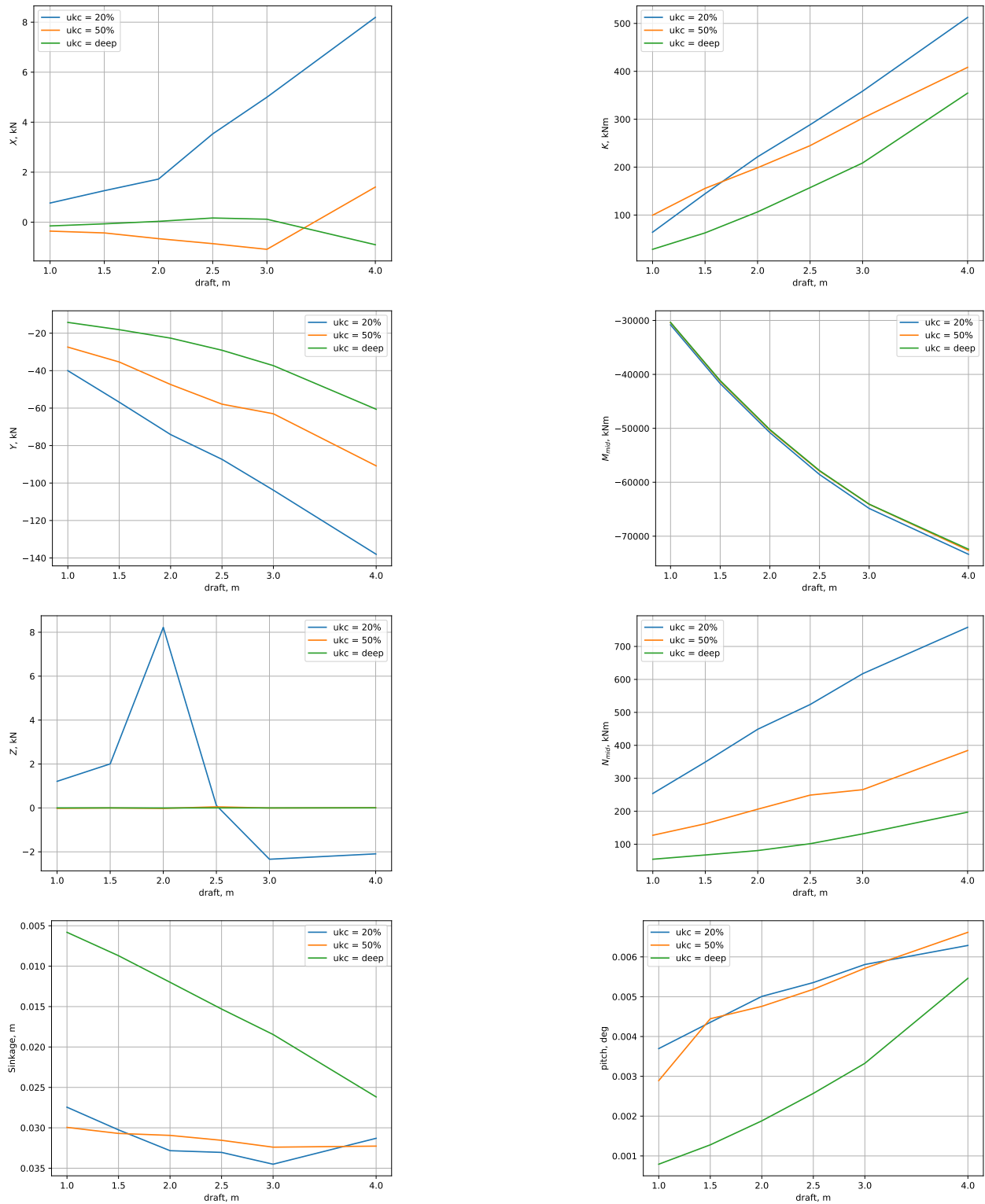
3.3.4 Comparison of average quantities

For all cases, the average forces, moments, sinkage and trim were computed from the time histories of the computations. Graphs will be plotted for each of the three ukc values (20 %, 50 % and deep water). This makes trends as a function of the draft visible.

The longitudinal force on the hull X reduces in magnitude as the draft is decreased. For the shallow water

case, it points forward independent of the draft, whereas for the other UKC values, the graphs do cross the zero line once or twice. The UKC has a significant influence on the lateral (drag) force experienced by the barge. The lateral force for 1 m draft and 20 % UKC is approximately equal to the lateral force at 3 m draft in deep water.

Figure 25 – Averaged values of the forces, moments, sinkages and trim as a function of draft.



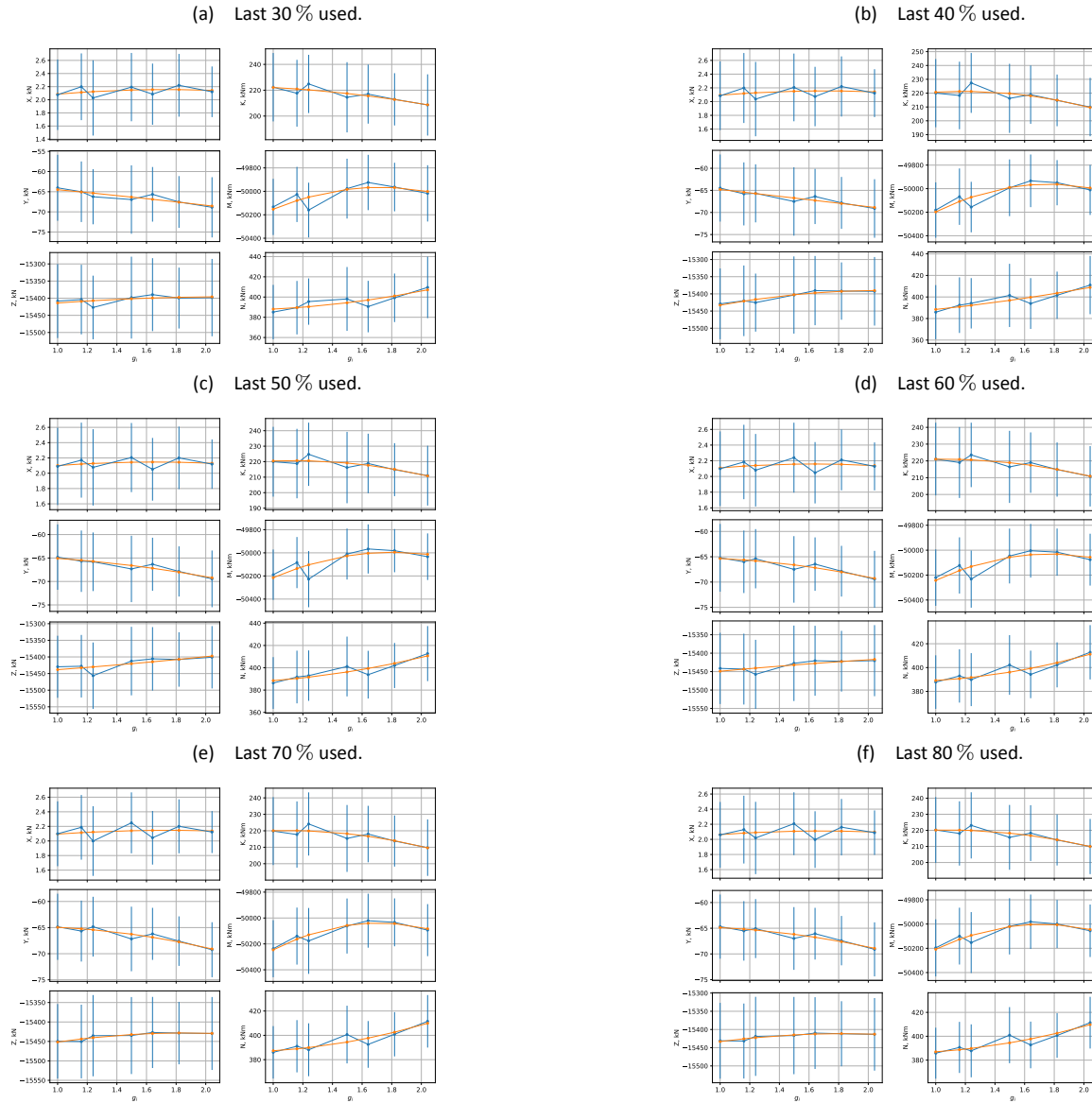
4 Conclusions

For project 19_012, the mathematical models of push barges available at FHR had to be extended with lower draft values than those tested in the towing tank in the past (3 m and 4 m). A CAD model of a 76 m barge was prepared for CFD computations where the draft was varied from 1 m to 4 m (6 conditions) and the under keel clearance was simultaneously varied between 20 %, 50 % and a deep water condition with a fixed water level of 8 m. In total, 18 conditions were tested and for each of these 18 conditions, a separate grid had to be constructed. A two-number identification was assigned to each computation. Computation C14 (with a draft of 2 m and a UKC of 20 %) was used to configure the computational environment. First, the focus was on the domain size, grid settings and solver settings. Once a suitable grid was constructed, a grid convergence study using a total of 7 grids was executed. A visualisation of the water surface showed that for the two coarsest grids, issues are visible near the edges of the water volume refinement box. These are not present for the finer grids. Convergence of resultant forces and moments on the barge showed that for all quantities except the yawing moment, standard deviations of the time signals (the fluctuations) are larger than the difference between the minimum and maximum value of the computed averages. These high fluctuations are caused by the massive flow separation that occurs on the downstream side of the barge. It was concluded that it is more important to execute simulations for a long time period than to execute simulations on a fine grid. The mesh settings for the coarse1 grid were used to construct the grids for the other 17 cases.

Plots of averaged quantities show that both the under-keel clearance and the draft have a significant influence on the hull forces and moments.

A1 Residuals of the grid convergence study

Figure 26 – Dependence of the convergence of the hull forces and moments on the averaging interval length. Quadratic fits of the data (orange) show global trends.



A2 Graphs of final results

A2.1 Time histories of forces and moments

Figure 27 – Time traces of the forces and moments for cases C11, C12, C13, C14, C15 and C16.

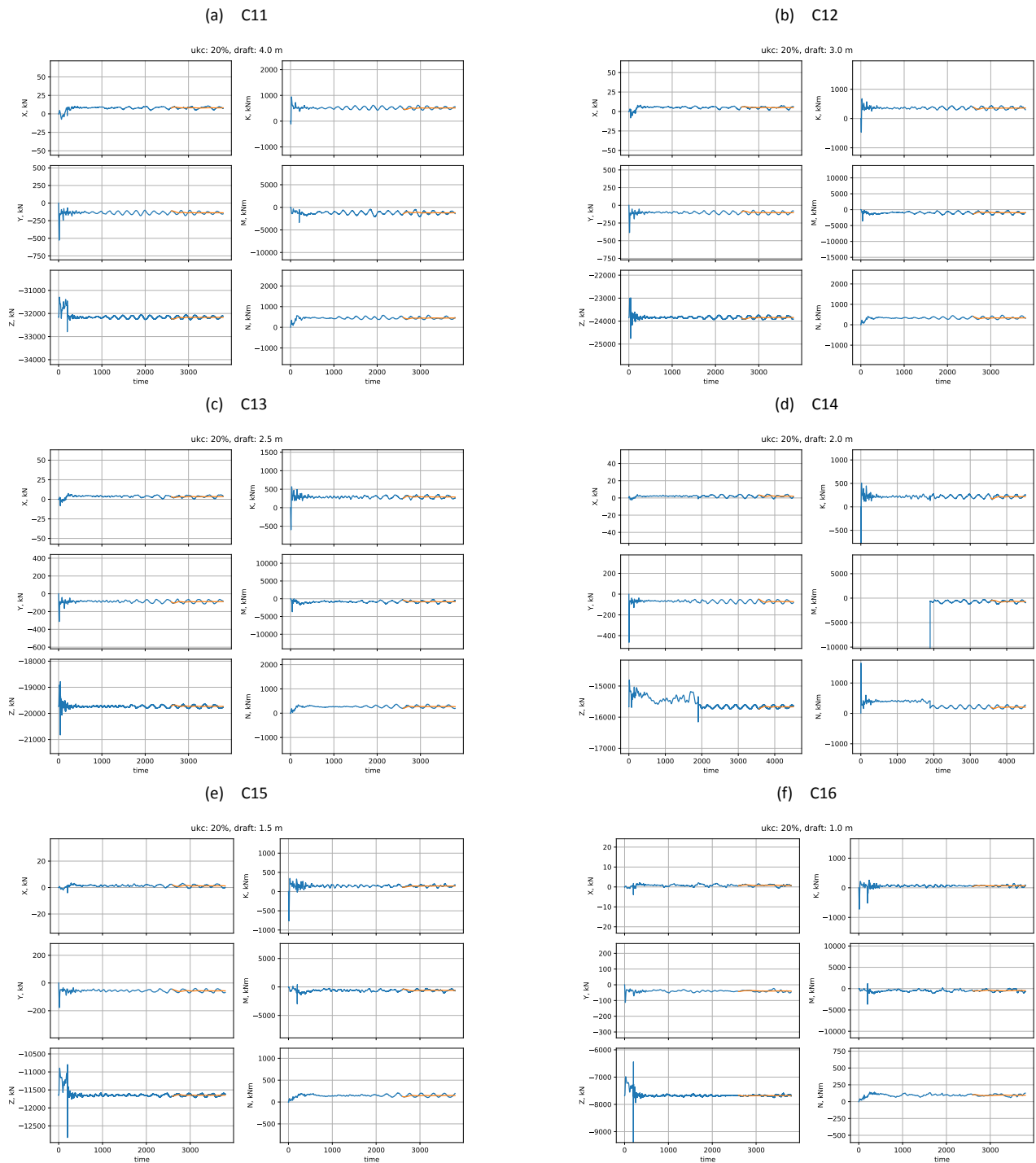
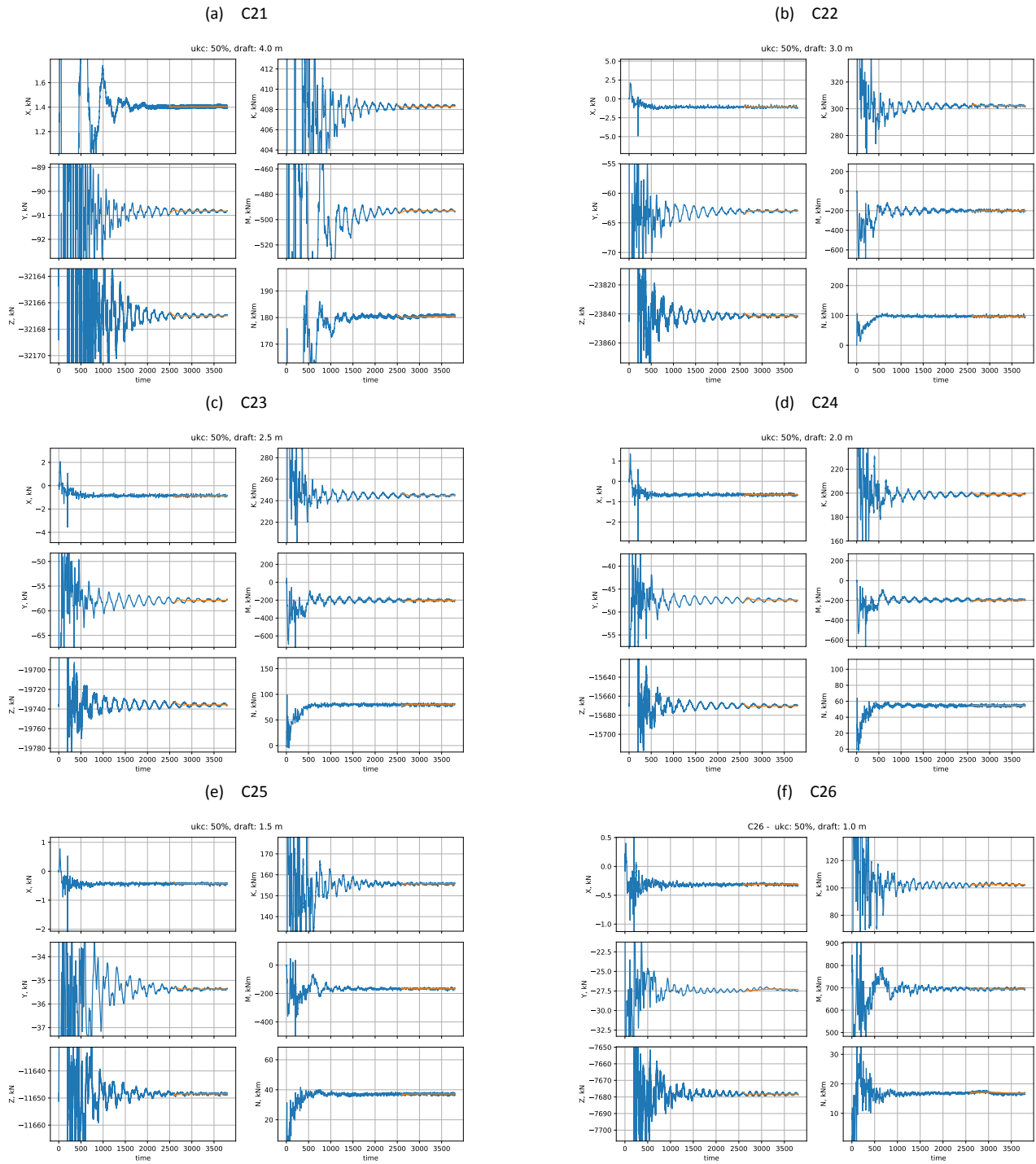


Figure 28 – Time traces of the forces and moments for cases C21, C22, C23, C24, C25 and C26.



A2.2 Time histories of motions

A2.2.1 Residual time histories

Figure 29 – Time traces of the forces and moments for cases C31, C32, C33, C34, C35 and C36.

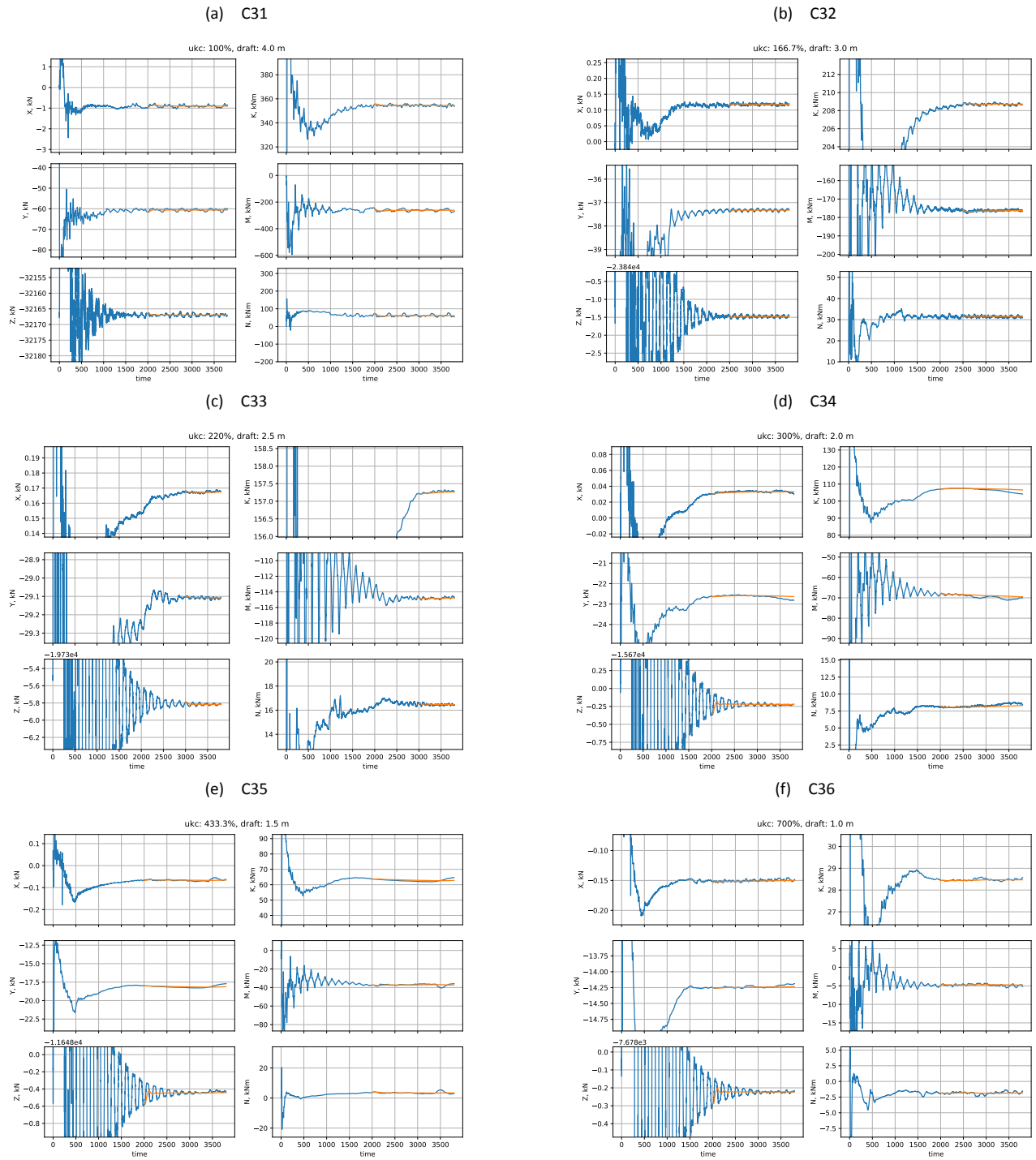


Figure 30 – Time traces of the motions for cases C11, C12, C13, C14, C15 and C16.

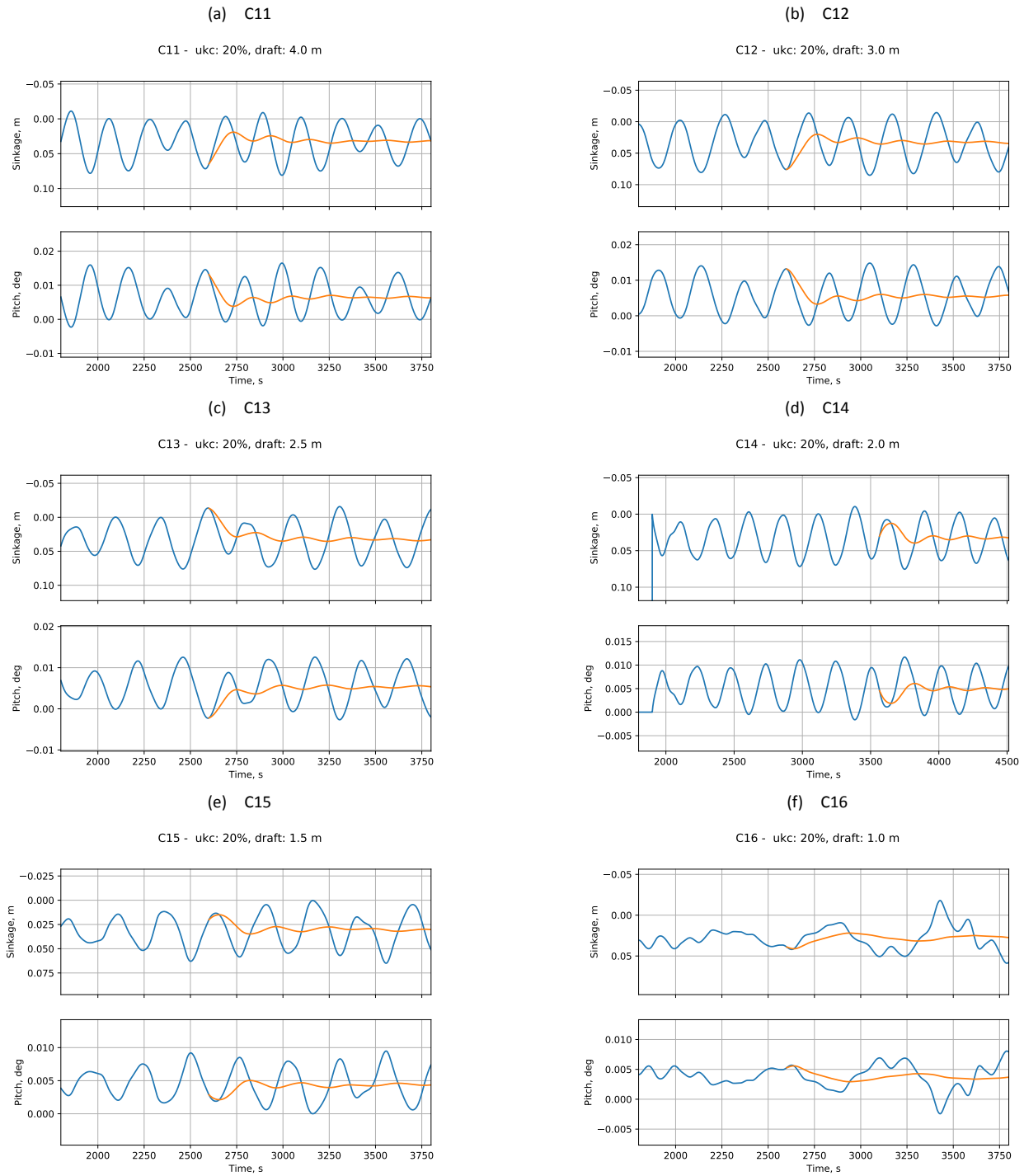


Figure 31 – Time traces of the motions for cases C21, C22, C23, C24, C25 and C26.

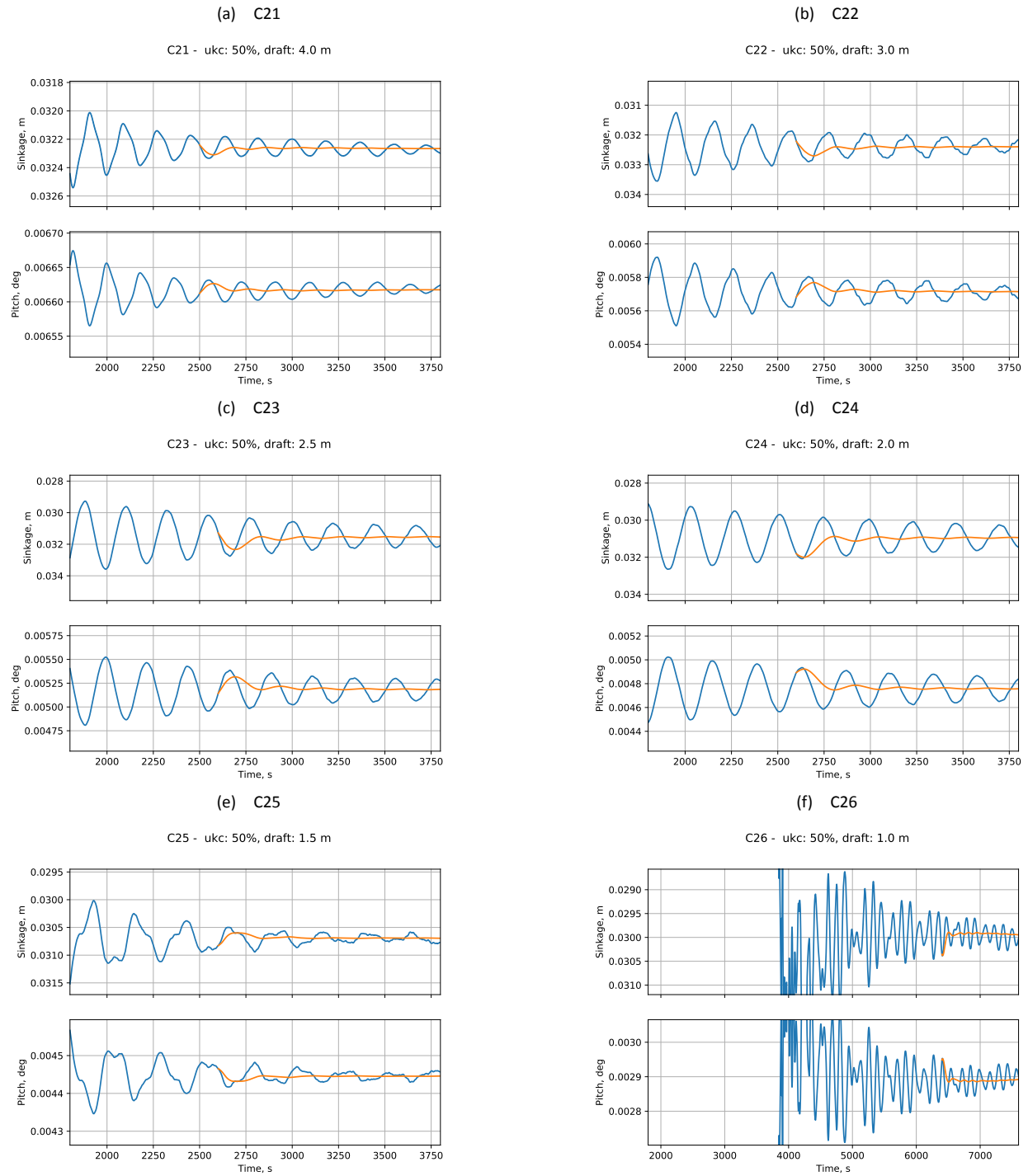


Figure 32 – Time traces of the motions for cases C31, C32, C33, C34, C35 and C36.

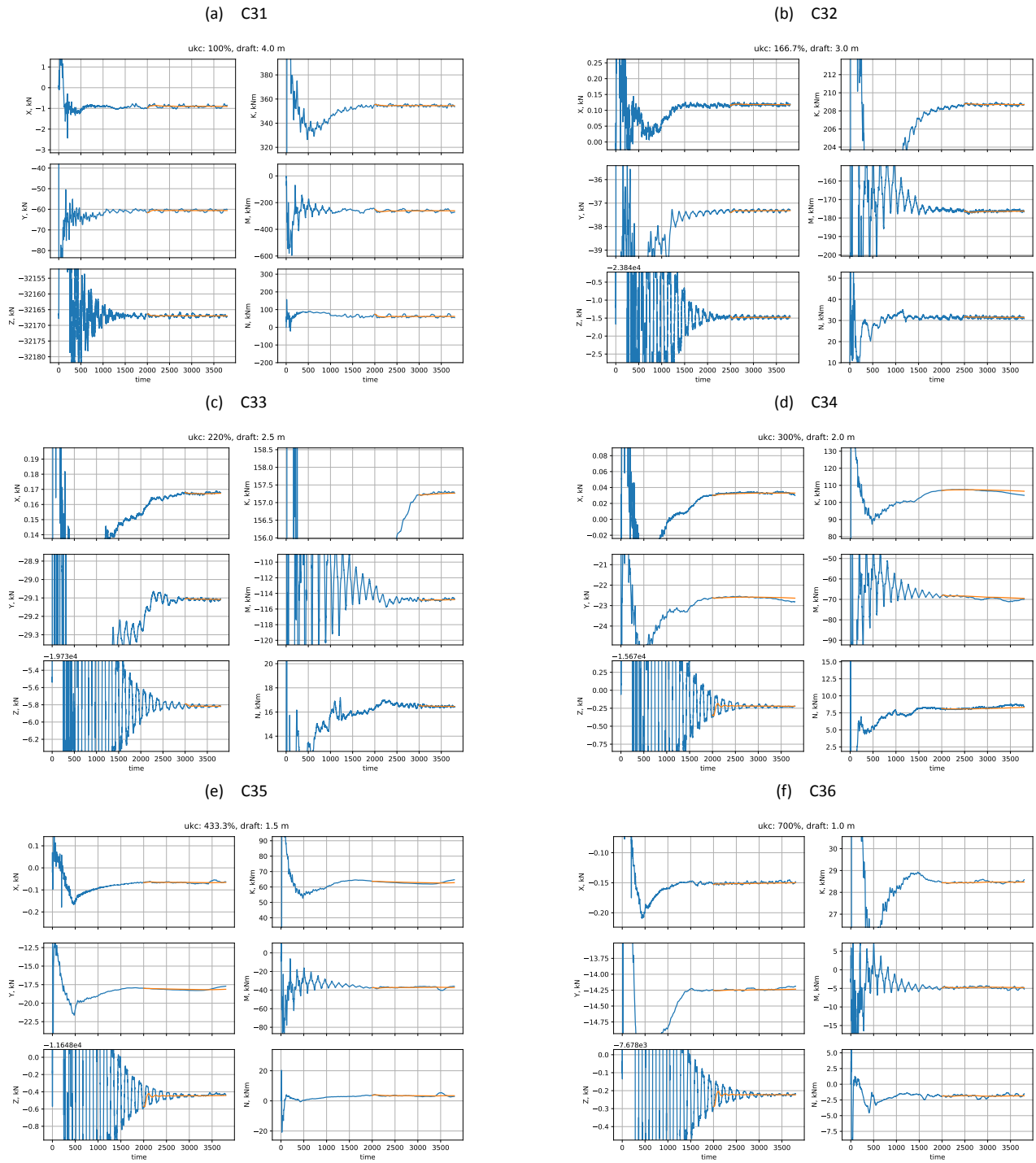


Figure 33 – Time traces of the residuals for cases C11, C12, C13, C14, C15 and C16.

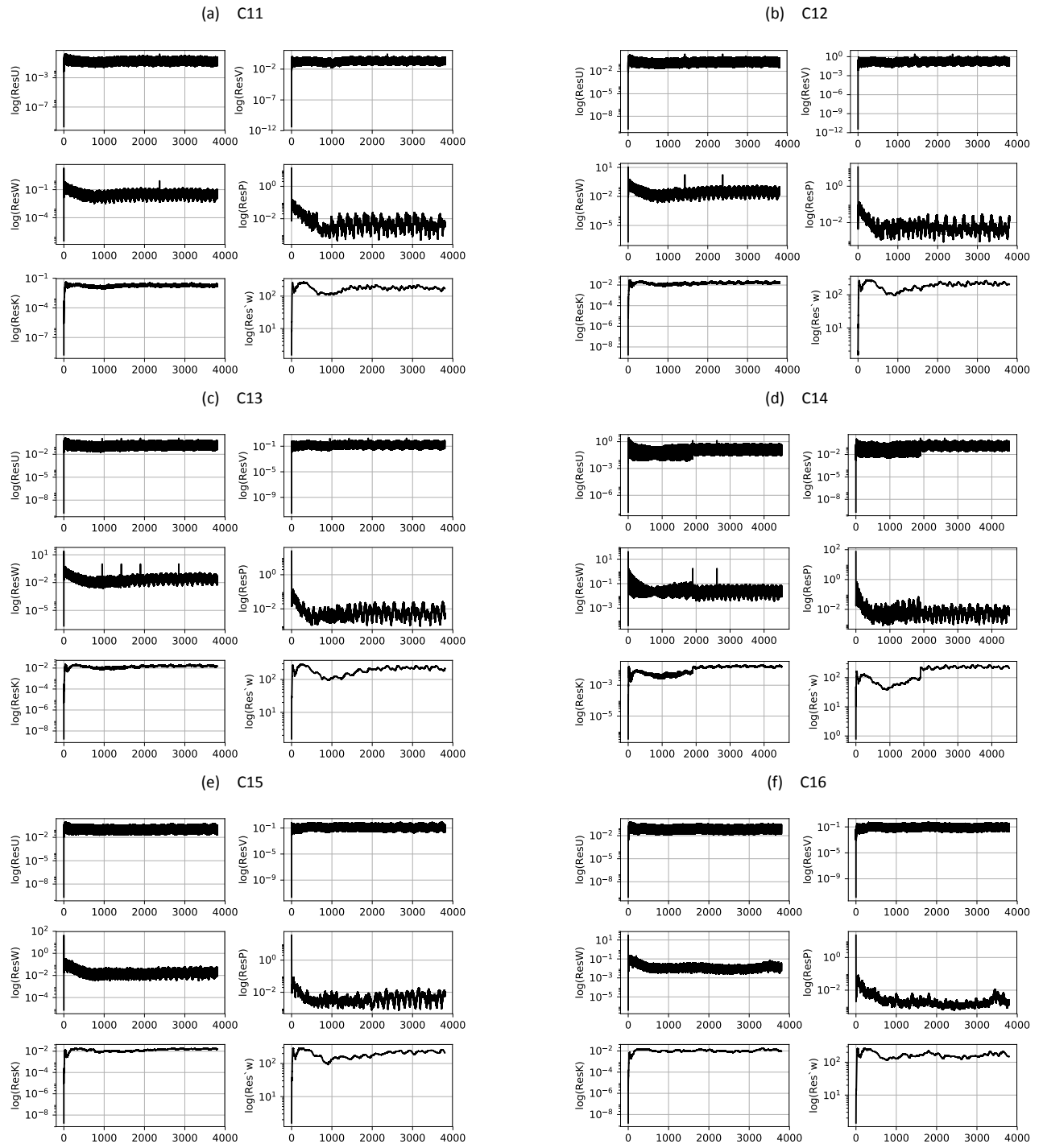


Figure 34 – Time traces of the residuals for cases C21, C22, C23, C24, C25 and C26.

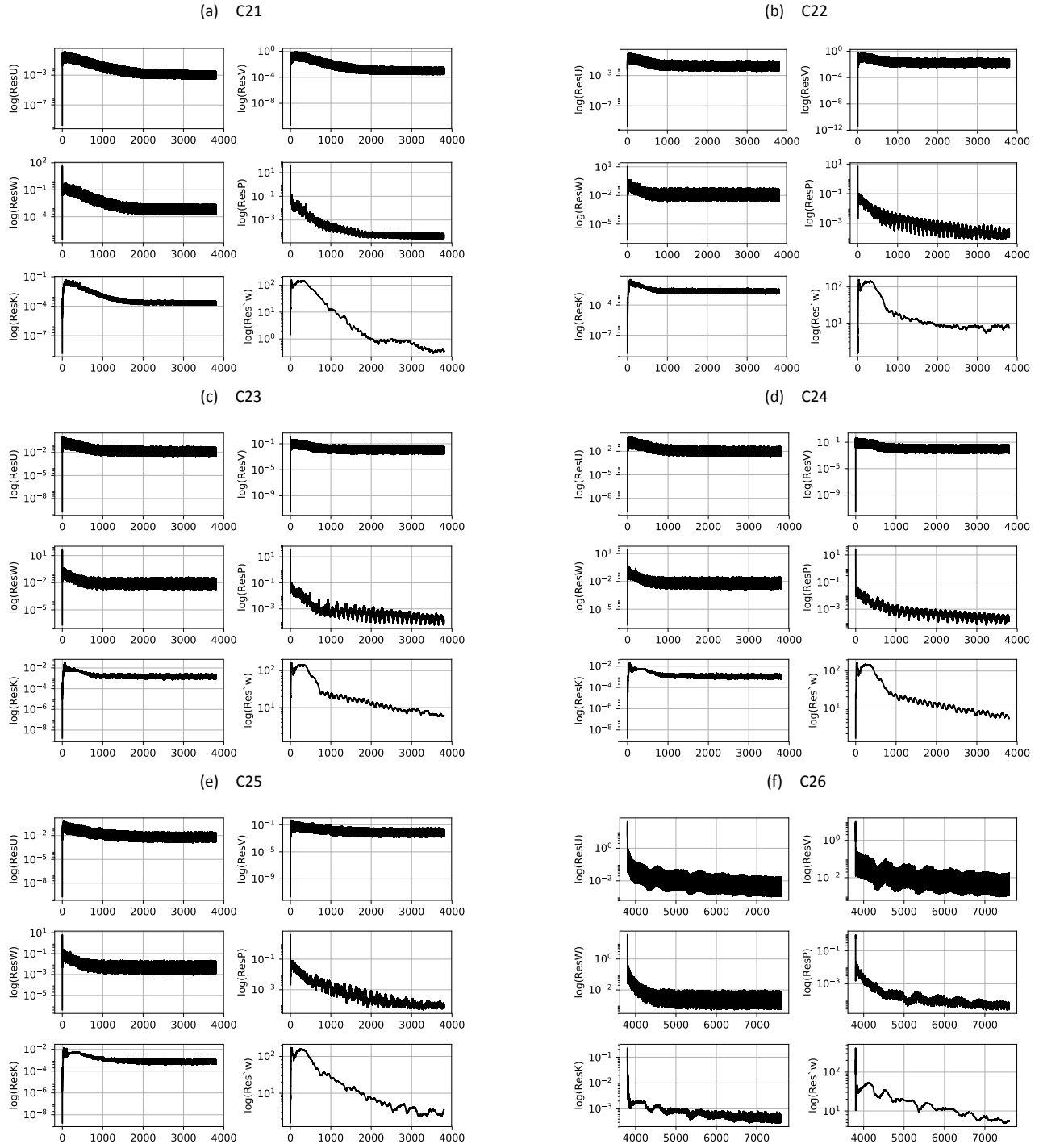
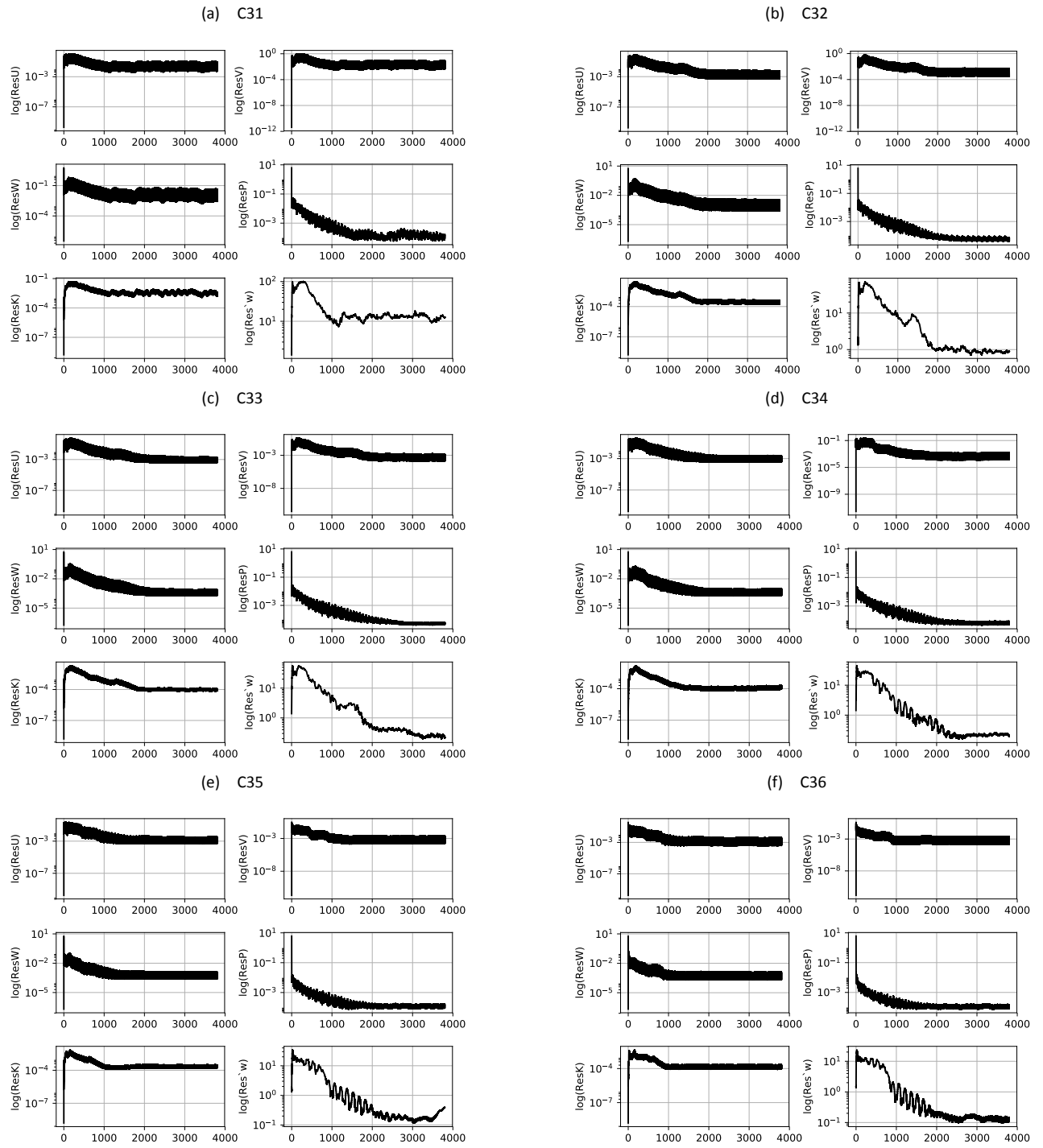


Figure 35 – Time traces of the residuals for cases C31, C32, C33, C34, C35 and C36.



DEPARTMENT **MOBILITY & PUBLIC WORKS**
Flanders hydraulics Research

Berchemlei 115, 2140 Antwerp

T +32 (0)3 224 60 35

F +32 (0)3 224 60 36

waterbouwkundiglabo@vlaanderen.be

www.flandershydraulicsresearch.be