TUTORIALS

FINE™/Marine 10.2

www.numeca.com

CONTENTS

TUTORIAL 1. C-WIZARD RESISTANCE

TUTORIAL 1.

C-WIZARD RESISTANCE

1.1 INTRODUCTION

1.1.1 Problem Description

Ship hull performance calculations have become a classical procedure for marine CFD computations. Decreasing the time of the complete simulation setup would simplify the procedure for computations and provide quick and easy hull performance estimation as well as the resistance curve for a particular hull. C-Wizard mode is introduced into FINE™/Marine software for these purposes.

A full scale resistance computation for the KCS model is performed with the following settings:

- Reference length (Lpp): 230m;
- Velocity: 12, 18 and 24 kt; Froude numbers between 0.13 and 0.26
- Draft of 9m in the full scale ship frame of reference;
- Water density: 1026.021kg/m^3;
- Kinematic viscosity: 1.19e-6m^2/s.

1.1.2 Objectives

The goal of this tutorial is to provide step-by-step instructions for the C-Wizard mode resistance calculation setup including additional numerical and flow parameters: Adaptive grid refinement, actuator disk and external forces. The widely investigated full scale container ship (KCS) hull is employed here for computation performance.

The tutorial gives best practices on the C-Wizard mode computation and mesh setup, giving the flow and mesh settings details information. Indeed, parameters are imposed automatically by the C-Wizard where the minimal user input is required. Geometry patches merging recommendations are developed to support best practices for the challenging geometrical features.

1.1.3 CPU Prerequisites

In order to ensure a smooth FINE™/Marine experience, it is advised to use a computer with the following resources:

- 4GB of RAM;
- 5GB of disk space available to store all files;
- 64bits machine with 8 cores.

1.1.4 Estimated Engineering and Computing Time

Engineering time (user interaction required):

- C-Wizard automatic setup: 7minutes;
- Computation time: 8hours per speed, 24h in total;
- Post-processing: 30minutes.

1.1.5 Preparation

1. Locate and copy the file "KCS_hull_SVA_cf_withnames_cf2.x_t" into your working directory.
2. Start FINE™/Marine v10.2.

How to launch FINE™/Marine

- For Linux systems, you can access the FINE™/Marine v10.2 graphical user interface with the following command line:

  finemarine102 -print

- For Windows systems (Windows 7 and older), you can access the FINE™ /Marine v10.2 graphical user interface from the **Start** menu by going to /Programs/NUMECA software/FineMarine102/FINE. In Windows 8 you can access it by going to the **Start** menu and clicking on **Search**. Under Apps, there will be a section called Numeca software. Click on FINE(#-bits) in order to open the FINE™ /Marine v10.2 graphical user interface.

Click here to start the C-Wizard setup...

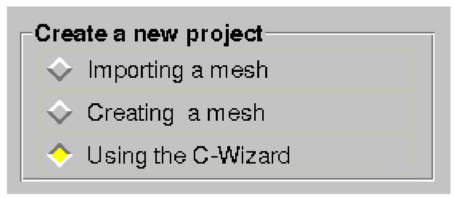
1.2 PART I - PROJECT SETUP

1.2.1 Start C-Wizard

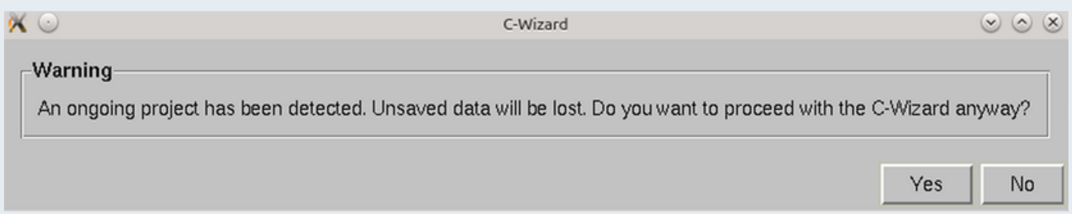
Launching the first part of the **C-Wizard** plugin is performed from the FINE™/Marine interface.

1.1. Open FINE™/Marine software.

1.2. **C-Wizard** plugin creates its own project as a step of the project setup procedure. Select **Using the C-Wizard** to launch the wizard.



When starting the **C-Wizard** when there is a project already opened in the FINE™/Marine interface, a warning window will appear asking for the action to execute.



To save the ongoing project settings, select **No**, then save the project, go to **Project** > **New** and select **Using the C-Wizard** again.

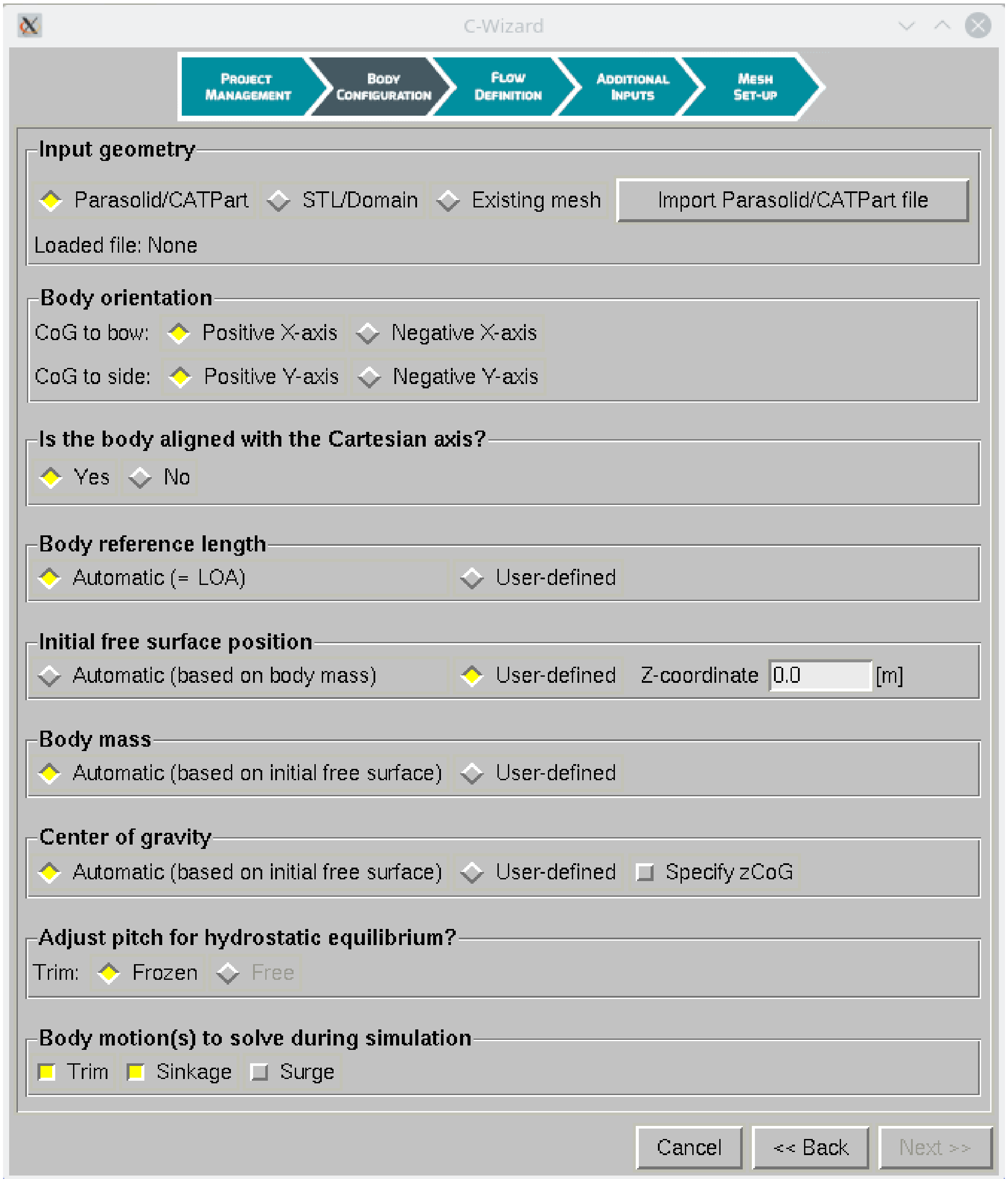
It is recommended to start the **C-Wizard** mode computation from the empty FINE™/Marine interface since there will be a full-chain project setup provided. Selecting **Yes** in the warning will close the opened project without saving and a new setup procedure will be started.

1.2.2 C-Wizard Part I: Create & Set up Project

A. Create Project

2.1.1 Create a project by clicking the **Create project** button. In a browser define the project name in the directory of your choice. Keep the default units.

2.1.2 Select **Resistance** in the **Application** section and click on the **Next>>** button to move to the following step of the setup. In the appearing **C-Wizard** window, the project setup will be available first.



B. Set up project: Body configuration

2.2.1 Under **Input geometry** : select **Parasolid/CATPart** and click on the **Import Parasolid/CATPart file** button ('*.x_t' and '*.CATPart' formats are available here).

2.2.2 Import the previously downloaded Parasolid file ***KCS_hull_SVA_cf_withnames_cf2.x_t*** (available in tutorials folder under ***_beginner\Tutorial_3***). When the geometry is imported, impose the following settings:

2.2.2.1 Select **Half body** to inform the C-Wizard what the loaded configuration is.

The body and flow conditions here are symmetric, thus half of the body is used. It is also possible to import an entire body geometry and create a half-body domain of computation. Running **Half body** will speed up the computation, as the mesh size will be halved.

2.2.2.2 Under section **Body orientation**, activate **Positive-X axis** for CoG to bow and **Positive-Y axis** for CoG to side.

The orientation of the X-axis is required to avoid defining negative speeds in the following entries; Y-axis direction will help to define the domain configuration.

2.2.2.3 Select **Yes** under section **Is the body aligned with Cartesian axis?**

This question sets the Cardan angles. When **Yes** is selected, Cardan angles are automatically set to zero.

2.2.2.4 Select **Automatic (=LOA)** under section **Body reference length**. The reference length will be automatically measured by the **C-Wizard** and set to the Length Over All.

2.2.2.5 Select **User defined** and set the **Z-coordinate** to <9.0> under section **Initial free surface position**.

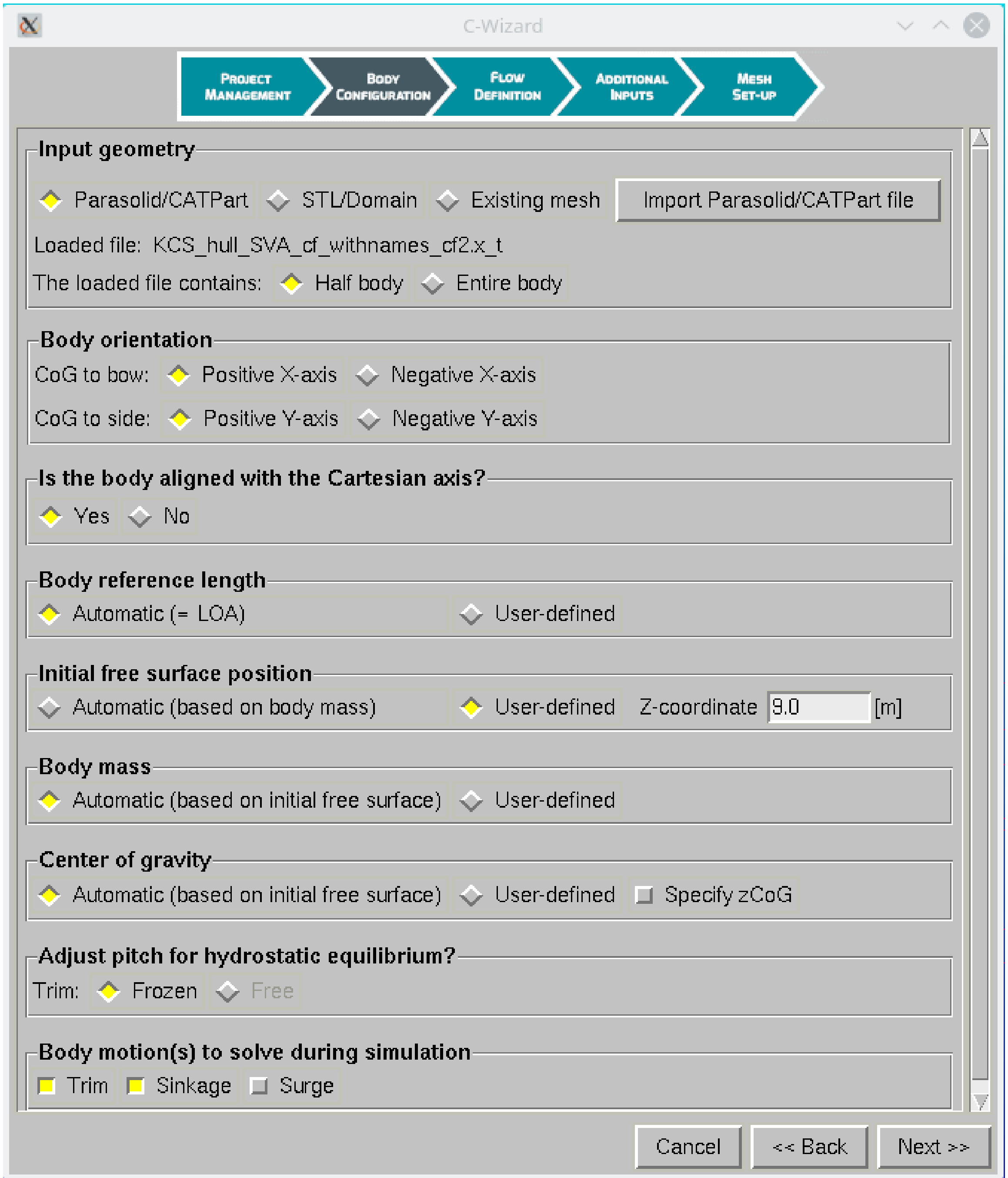
The initial free surface is defined in the geometry coordinate system. In this case Z=0 is located at the hull bottom.

2.2.2.6 Select **Automatic (based on initial free surface)** under section **Body mass**. The body mass will be automatically calculated based on the equilibrium position.

2.2.2.7 Select **Automatic (based on initial free surface)** under section **Center of gravity**. The center of gravity position will be automatically calculated based on the equilibrium position.

2.2.2.8 Keep the trim set to **Frozen** under the **Adjust pitch for hydrostatic equilibrium?** section.

2.2.2.9 Keep the **Trim** and **Sink** in a **Body motion(s) to solve** active.

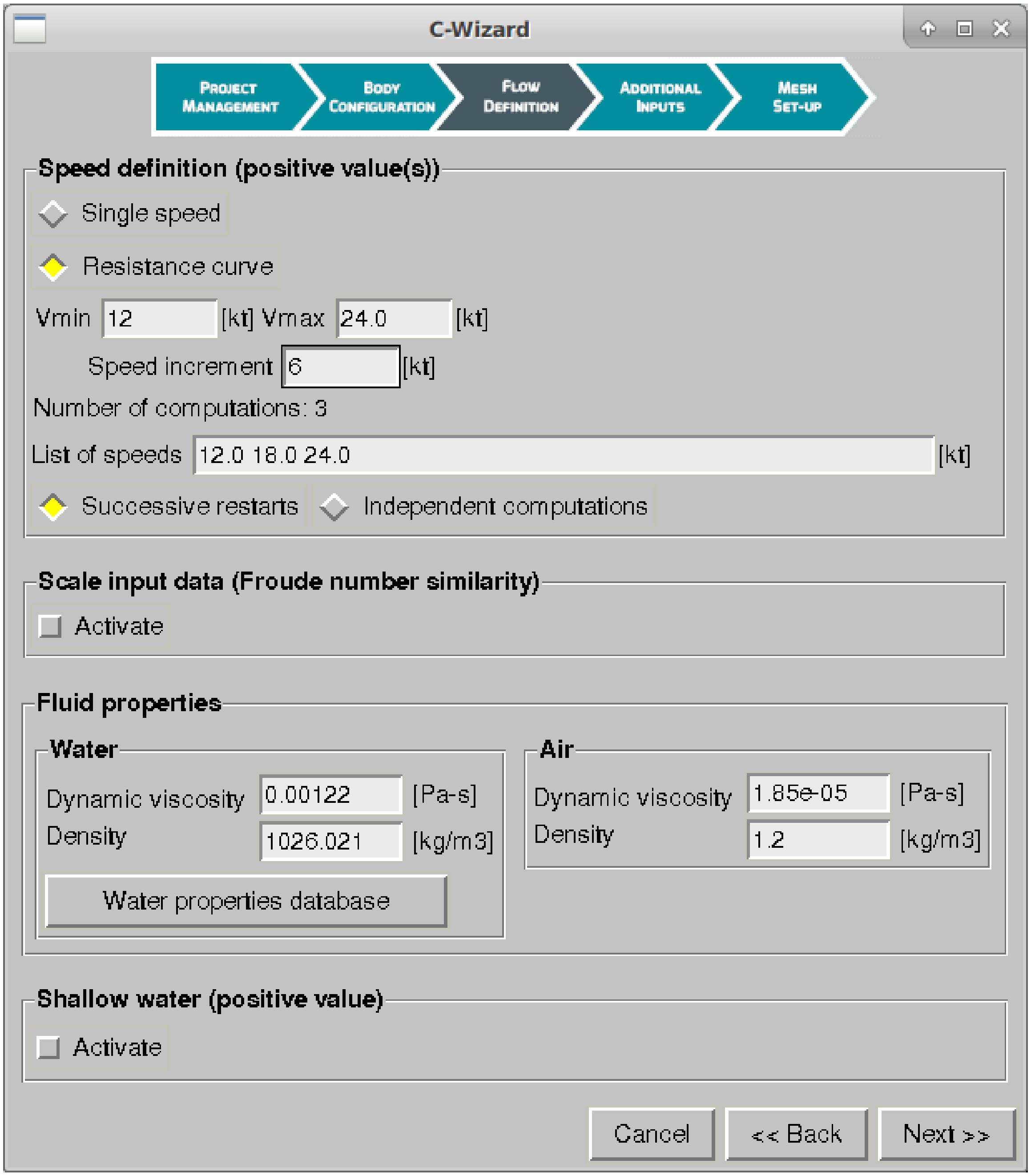


Click on **Next>>**.

C. Set up project: Flow definition

2.3.1 In the section **Speed definition**: activate **Resistance curve**. Define V_{min} = 12kt, V_{max} = 24kt, **Speed Increment** = 6kt and press *Enter*.

Select **Successive restarts** to have each computation start from the result of the previous speed.



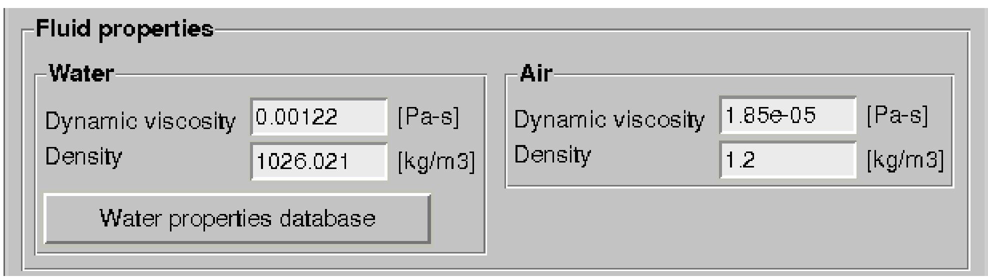
If the increment of speed between computations is not constant the user can directly input a list of values separated by spaces or commas in the **List of speeds** field.

2.3.2 Keep deactivated the **Scale input data** (Froude number similarity).

If the user would like to run this project in model scale, all inputs should be given in full scale, and the Scaling factor set to 0.0316. The C-Wizard would then take care of scaling the geometry and all settings to keep a Froude number similarity.

Here the calculation is performed for the full scale ship and this option should not be activated,

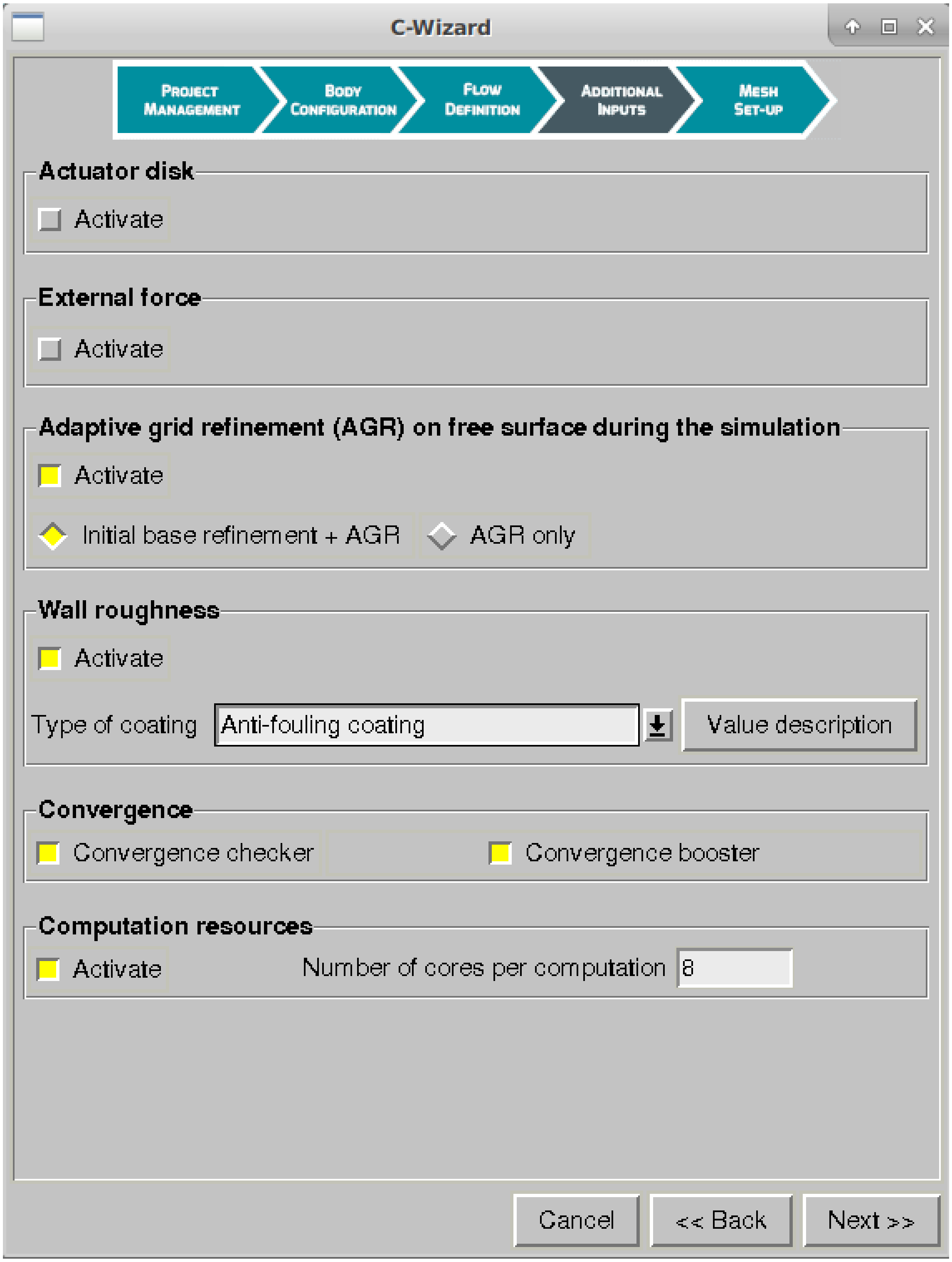
2.3.3 In **Fluid model** section: keep default properties for **Air** and click on the **Water properties database** button to change the water properties to Salt water at 15 °C. Click on **OK** to validate the new properties.



2.3.4 Click on **Next>>** to proceed to the additional input setup.

D. Set up project: Additional inputs

A new window with additional parameters is available here:

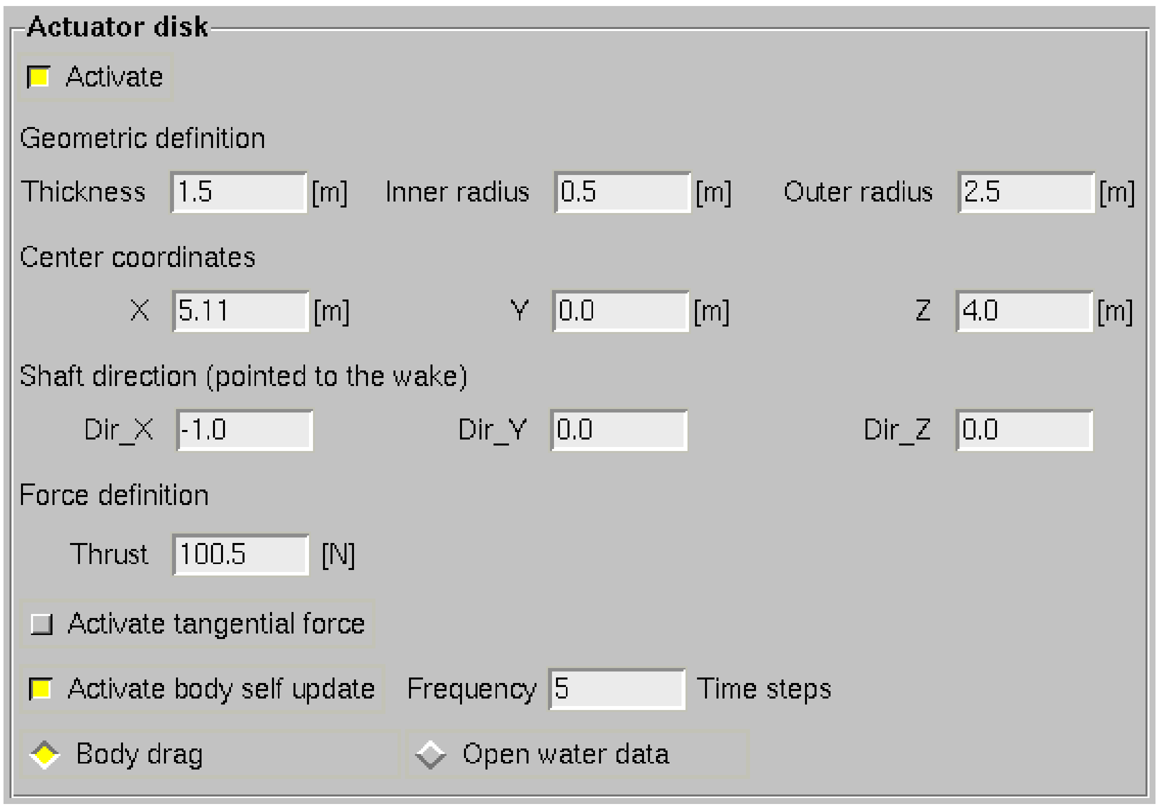


2.4.1 **Activate** the **Actuator Disk** and enter the following parameters:

- Set **Thickness** to <1.5> [m],
- Set **Inner/ Outer radius** respectively to <0.5> and <2.5> [m],
- Set the **Center coordinates** to <5.11 0.0 4.0> [m] for **X**, **Y**and **Z**,

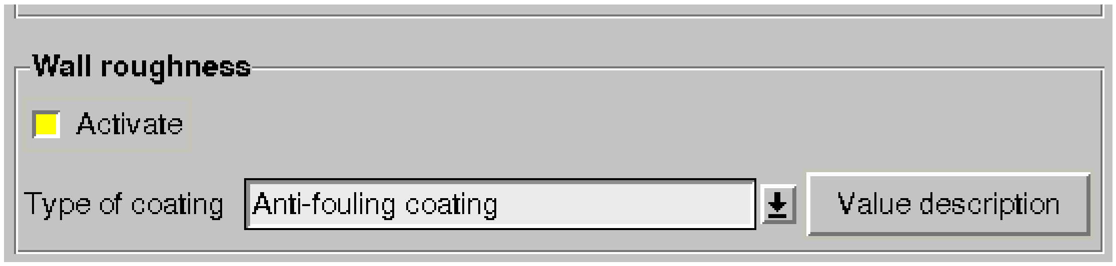
- Set the normal to <-1.0 0 0> [m] for **Dir_X**, **Dir_Y**and **Dir_Z**,
- Set **Thrust** to <100.5> [N],
- Set **Activate body self update** active,
- Set **Frequency** of update to <5> Time steps.

> When **Activate body self update** with **Body drag** is active, the thrust of the actuator disk is automatically updated during the computation such as Thrust = Drag at a prescribed interval. The interval corresponds to the frequency value.



2.4.2. **Activate** the **Adaptive grid refinement on free surface**, leaving the AGR setting to the default **Initial base refinement + AGR**.

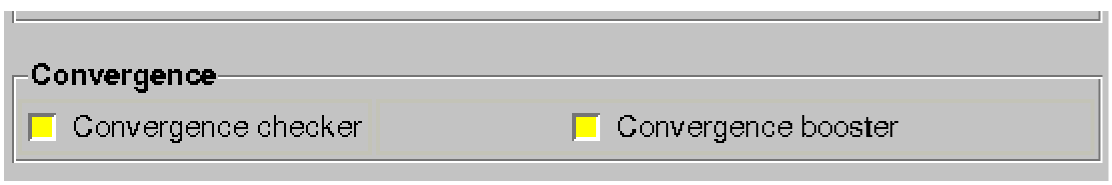
2.4.3 Activate **Wall roughness** and choose **Anti-fouling coating** in the drop-down list. You can click on **Value description** to get more details on the roughness values.



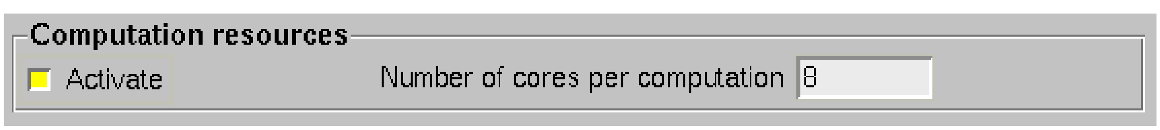
In full scale simulations the hull roughness will have a significant impact in the total resistance value and it is important to define it according to the hull surface real conditions.

2.4.4 Activate **Convergence checker** and **Convergence booster**.

- **Convergence checker** will stop the computation when the drag and solved motions become stable.
- **Convergence booster** will adapt the convergence settings during the computation to speed it up.



2.4.5 Set the **number of cores per computation** to <8> under the **Computation resources**.



2.4.6 Click on the *Next*>> to proceed to the domain creation and mesh setup.

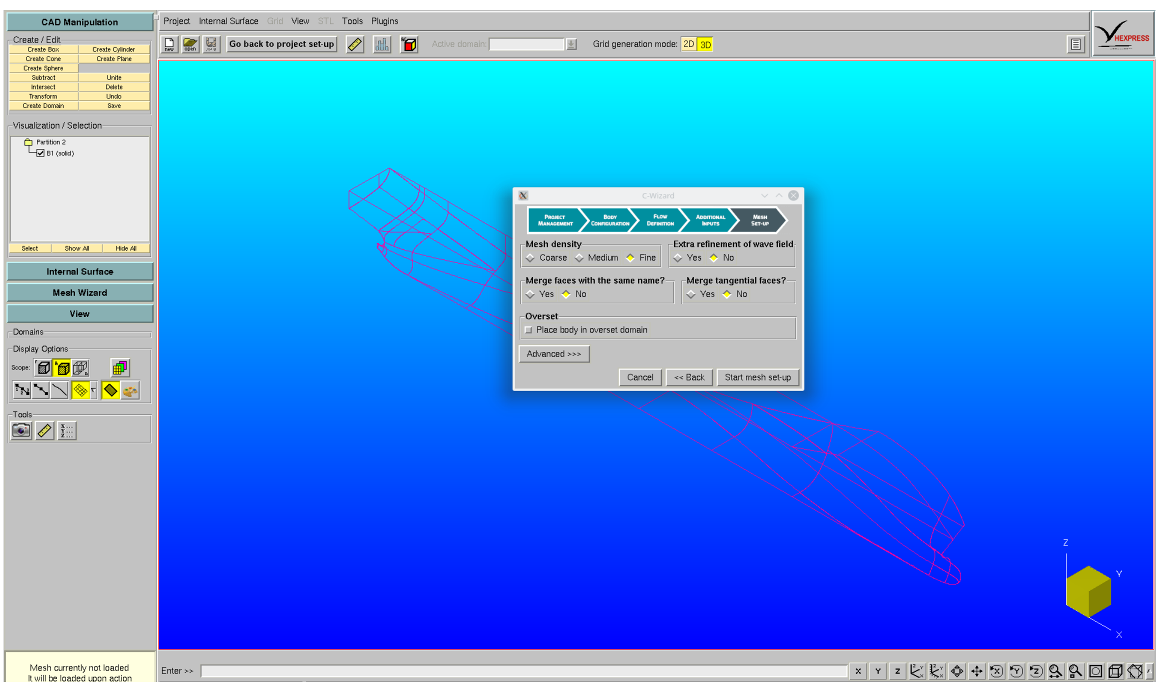
At this step, the file ***wizard.input*** is created and saved into the computation folder next to the ***.iec*** project file. It consists of all the inputs recorded and computed by the wizard. The switch to HEXPRESS™ interface is performed here: C-Wizard will generate domain and mesh automatically respecting the geometry and input parameters (body configuration, orientation, free surface position and etc.).

Click here for the domain and meshing details...

1.3 PART I - DOMAIN & MESH SETUP

1.3.1 Mesh Setup

The last page of the **C-Wizard** gives the user control on the domain and mesh setup.



1.1 Set Mesh density to **Medium**.

Refinements on patches are applied according to their names and to the mesh density level.

The initial cell size in the mesh is defined to have a number of cells per maximum length of the ship (*Loa*). For example, the number of cells per Loa for the initial mesh size is set to 3 if coarse, 4 if medium and 5 if fine.

The refinement level assigned to a patch according to its name is defined in the refinement dictionary.

1.2 Set **Extra refinement of the wave field** to **No**.

This option will create a refinement sector with low aspect ratio cells covering the Kelvin angle area. It will lead to a high resolution of the wave field, but it increases the number of cells in the mesh considerably. For the current Froude number additional refinement of the wave field is not necessary, thus this parameter is deactivated.

1.3 For the current computational project, geometry is checked and proper names were defined in the CADfix software.

For the C-Wizard projects **Merging by name is the priority**. The reason behind this choice is that a ship hull can be generally represented by several common parts like: bow, stern, deck, hull, appendages, etc. Once names are defined, **C-Wizard** will perform merging and apply mesh parameters automatically.

Two options are available from the C-Wizard mode: **Merge faces with the same name** and/or **Merge tangential faces**.

- Set **Merge faces with the same name** to **Yes**.

The **C-Wizard** reads the name of every patch and merges the ones that are adjacent and with identical names: all patches named "*Bow_1*" will be merged to one patch and named "*Bow_1*". When preparing the geometry, the user should give identical names to all patches that need to be merged together.

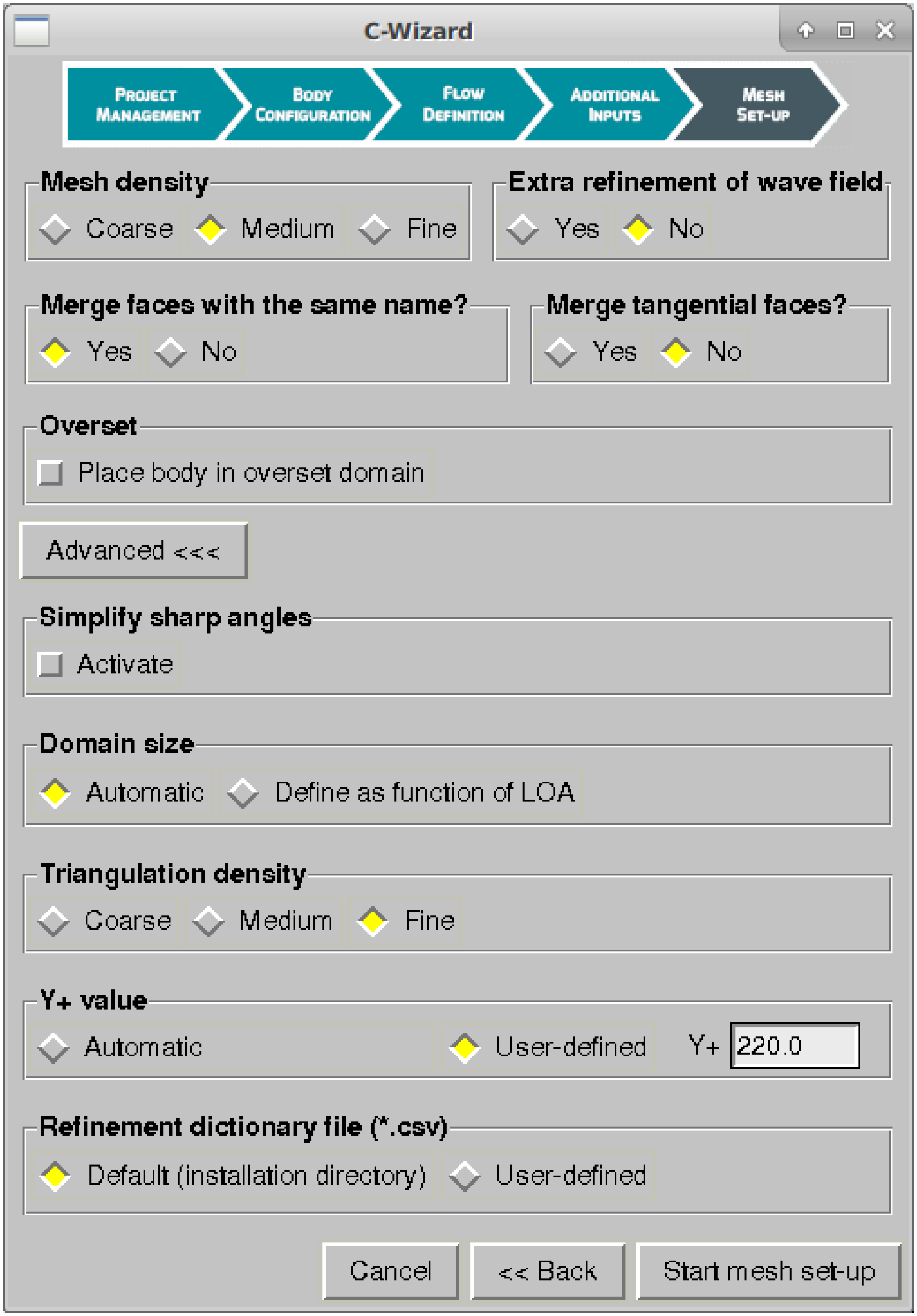
Patches named *Bow_1* and *Bow_2* will not be merged, but they will all get the mesh refinement settings assigned to "Bow " in the refinement dictionary.

- Keep **Merge tangential faces** to **No**.

> This option is exactly the same as for the HEXPRESS™ (**Domain Manipulation**) and when activated it checks if the neighbor patch has a tangent angle greater than a specified one and merge if it is greater. This method is especially helpful when the geometry imported is a *Parasolid* file with big number of patches.

Each time two faces are merged, the new face gets an ID (ID's are incremented one by one). It assigns specific values for the future mesh refinement strategy according to the name of the patch. By default, the name will be the one which leads to the higher refinement.

1.4 Click on the ***Advanced*>>>** button to check the additional parameters.



1.5 Keep the default domain settings: **User-defined domain size** not active.

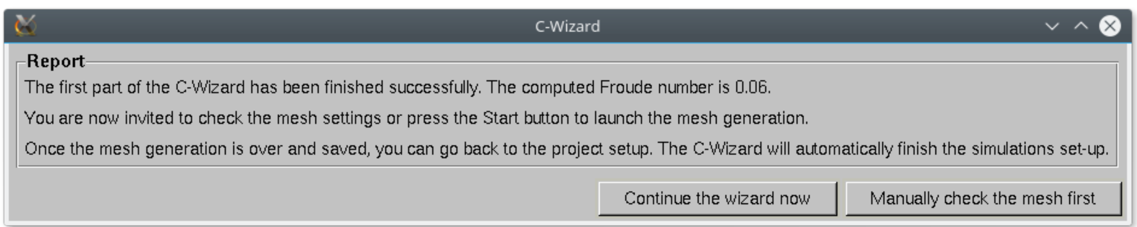
> When not active, the default domain is set to *5Loa x 2Loa x 2Loa*. This domain size is recommended for resistance computations with Froude number under 0.5. For higher Froude numbers the downstream boundary will be placed further from the hull to accommodate the longer wake.

1.6 Set **Triangulation density** to **Fine**.

1.7 Under section **Y+ value** select **User defined** and set Y+ to <220>.

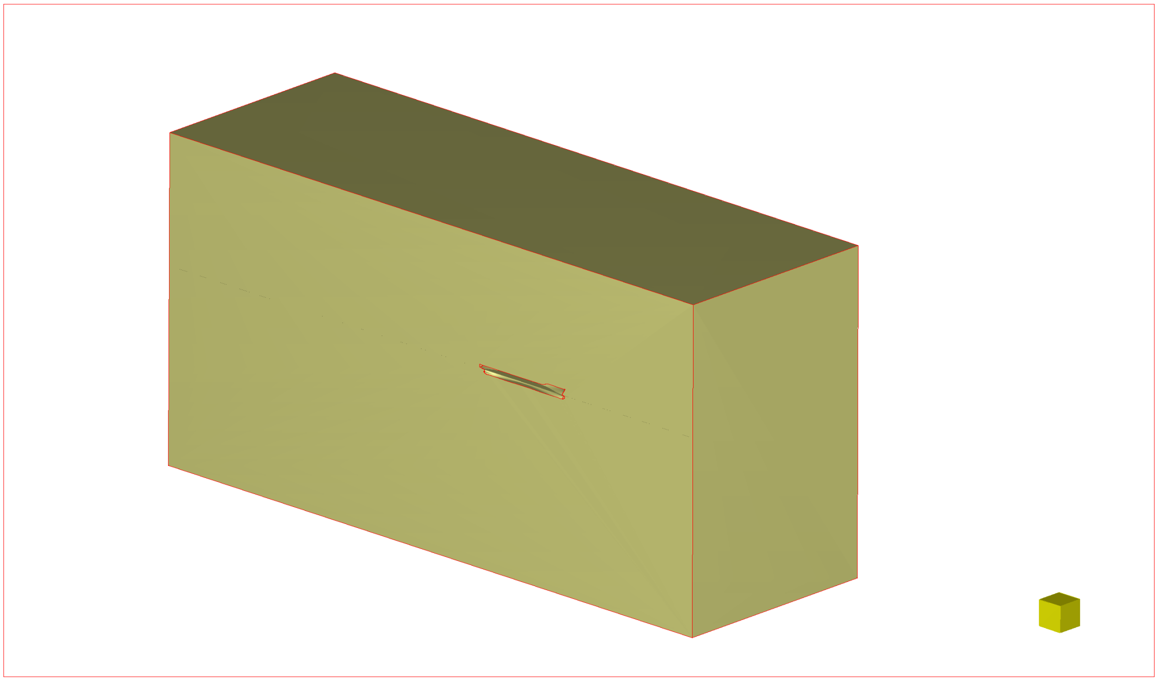
1.8 Click on *Start mesh set-up* button to start the domain creation and the mesh setup. One can check the shell to see the process in action.

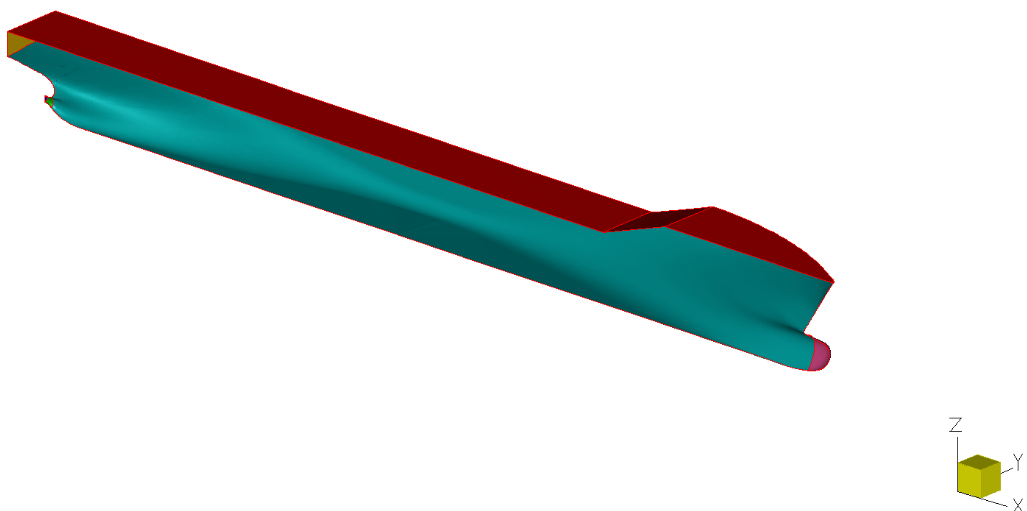
An information window reporting that the first part of the C-wizard has been successfully finished will appear. It also gives information about the computed Froude number and suggests the following actions. Click Manually check the mesh first to activate the HEXPRESS™ interface without immediately starting the mesh generation.



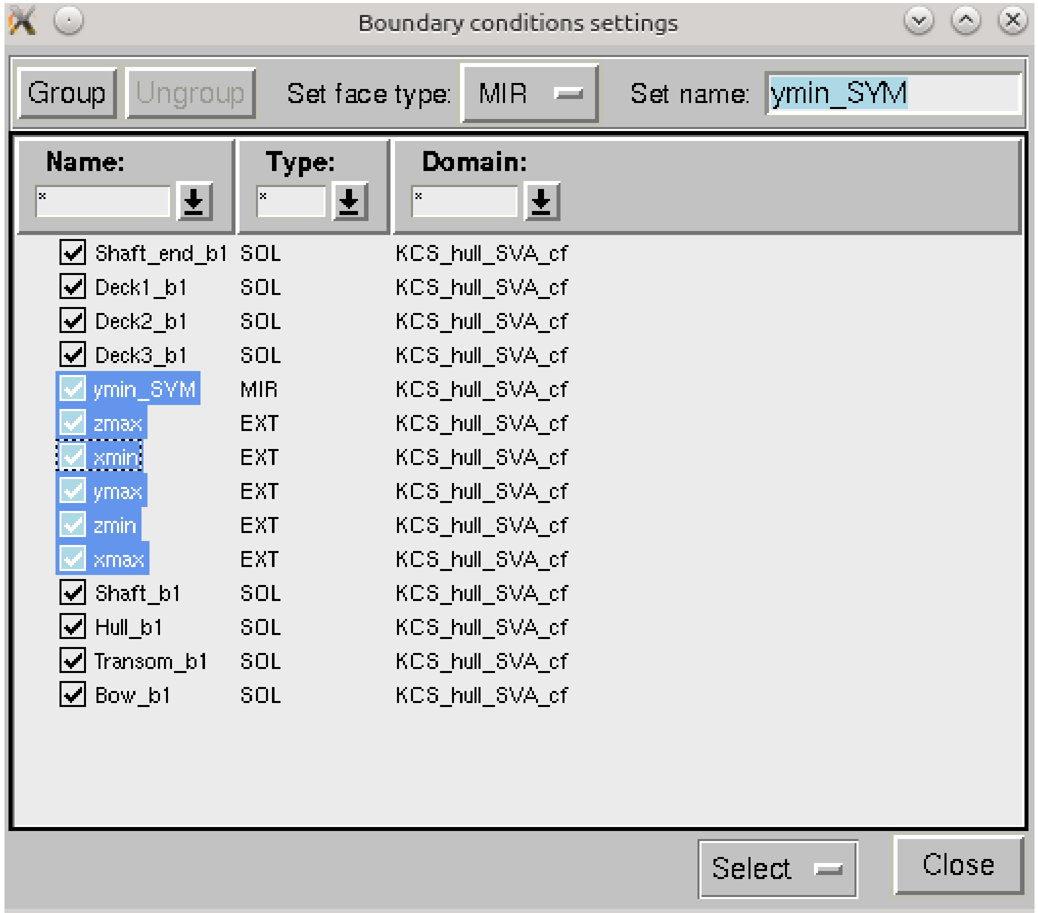
1.9 In HEXPRESS™ , the following parameters based on the previous inputs are imposed:

- Domain with the internal free surface is created, merging by name is performed according to the names of patches.





- External boundary conditions (**Grid/ Boundary Conditions**) of the domain are automatically named and defined as follows.



- The mesh setup is done automatically in accordance to the previously imposed settings (**Medium** density mesh). The parameters can be checked through the HEXPRESS™ **Mesh Wizard** menu:

Initial mesh
2304 cells

Adapt to geometry • Maximum number of refinements: 12 (Global tab) • Curve refinement: — Curve 158: 8 ; Target Cell Size (0,0,0) • Surface refinement: — Deck*: 4 ; Target Cell Size (0,0,0) ; Refinement diffusion: Global ; patches automatically grouped: "Deck1_b1", "Deck2_b1", "Deck3_b1" — Shaft*: 8 ; Target Cell Size (0,0,0) ; Refinement diffusion: Global ; patches automatically grouped: "Shaft_end_b1", "Shaft_b1" — Hull_b1: 6 ; Target Cell Size (0,0,0) ; Refinement diffusion: Global — Transom_b1: 8 ; Target Cell Size (0,0,0) ; Refinement diffusion: Global — Bow_b1: 8 ; Target Cell Size (0,0,0) ; Refinement diffusion: Global Global_FS: 7 ; Target cell size: (X: 74.9089, Y: 74.9089, Z: 0.0) ; Aspect Ratio 128 ; Refinement diffusion: 3 ; Free surface has anisotropic refinement in Z-direction to provide a sufficient mesh resolution for the wave elevation • Box refinement: — Sector #0: for the actuator disk zone; Refinement: 10 ; Target cell size: X=Y=Z=0.0 ; Refinement diffusion: 3 — Sector #1: for the actuator disk wake; Refinement: 9 ; Target cell size: X=Y=Z=0.0 ; Refinement diffusion: 3 • Trimming: all **SOL** patches are set to ***Used for trimming*** and **MIR** and **EXT** patches are defined as ***Not used for trimming***.
Snap to geometry Buffer insertion of **Type II** for all edges on the Mirror plane and External boundaries edges
Optimize Max nb of orthogonality optimization iterations: 5 Minimal orthogonality threshold: 5.0

Viscous layer

Viscous layers are defined and computed for Solid boundaries only and if the face name does not contain the word "deck" since there is usually no need to insert viscous layers (viscous effects from the air part are negligible).

Fixed first layer thickness Method

Floating number of layers:

Minimum number of layers: **11**

Maximum number of layers: **22**

Active with **First layer thickness** <0.005388>

for

Shaft: 13 layers

Hull_b1: 20 layers

Transom_b1: 13 layers

Bow_b1: 13 layers

1.3.2 Mesh Generation

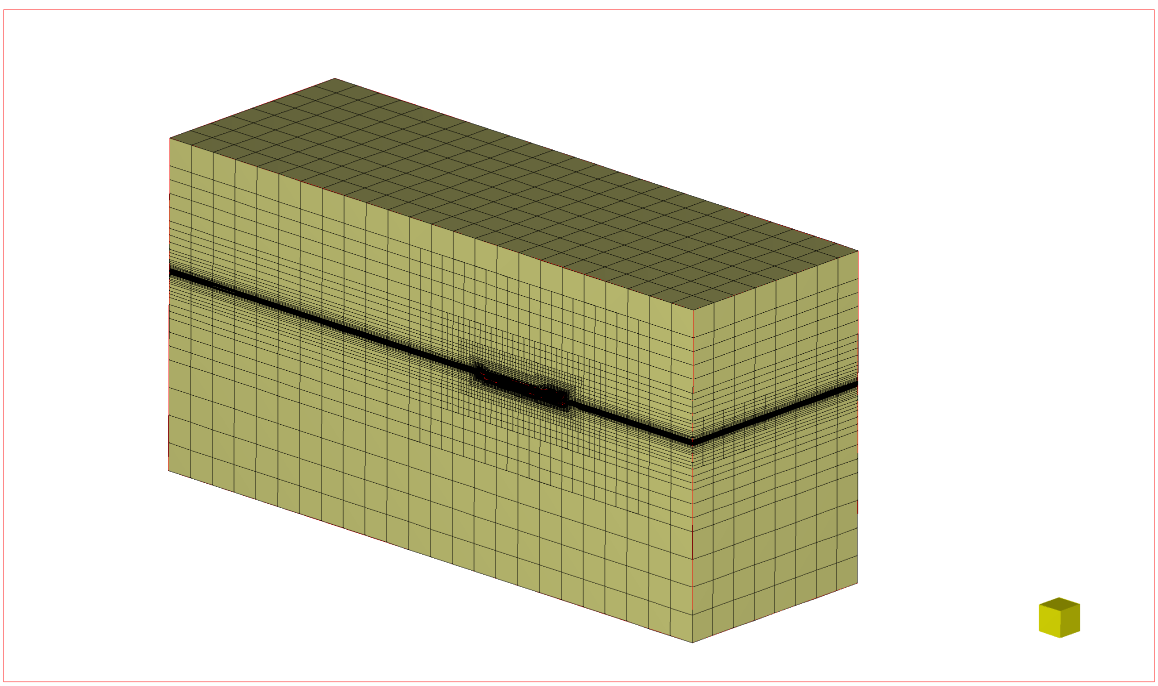
HEXPRESS™ includes a **multi-threading** capability allowing to speed up the Optimization step and the optimization loops included inside the Viscous Layer insertion step.

To activate it, in Menu bar of HEXPRESS™: ***Project > Preference > Mesh generation > Multithreading***

Please refer to the HEXPRESS™ documentation for the advised number of threads.

2.1 Click the ***Start*** button in the **Mesh Wizard** to generate the mesh.

The generated mesh looks like the picture below with around 1.2M cells.



The quality of the mesh can be improved here by generating the **Fine** mesh. For the current sample study a medium mesh is considered to be sufficient.

2.2 Once the mesh is generated, click the **Go back to the project set-up** button to start FINE™/Marine GUI.

2.3 Click on ***Yes*** to save the generated mesh.

Click here to finish the setup...

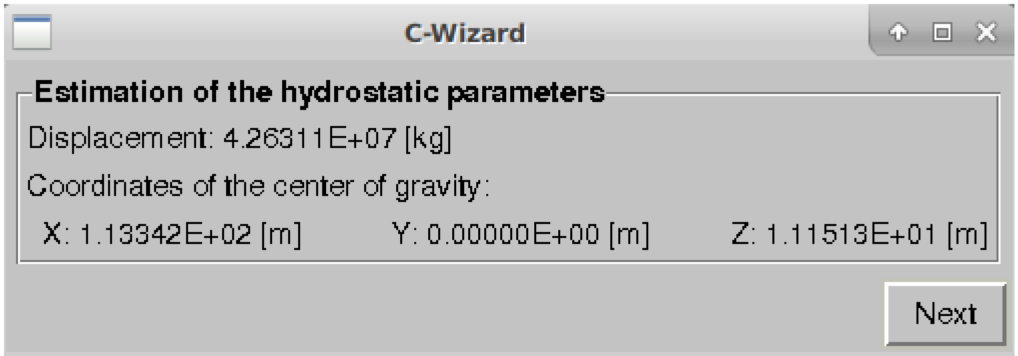
1.4 PART II - FLOW SETTINGS & POST PROCESSING

1.4.1 C-Wizard Part II: Flow Settings

A. Estimate Hydrostatic Values

1.1.1. Check the computed values.

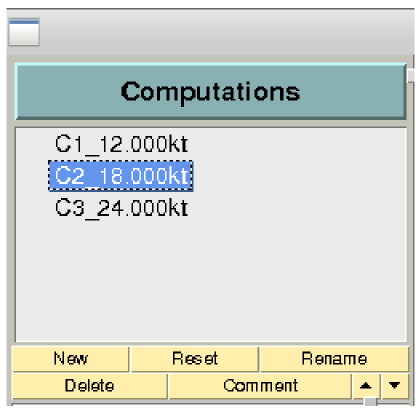
This part of the **C-Wizard** will provide the **Estimation of hydrostatic values** for the present study: the **Displacement** and **Coordinates of the Center of Gravity**. This part refers to the ***domhydro*** calculations (see the FINE™/Marine user guide for more explanations on the ***domhydro*** tool).



1.1.2. Check the computed values and click on *Next* to finalize flow settings setup.

B. Flow Settings

The automated **C-Wizard** setup will create 3 computations :



The computation C2_18.000kt has the following settings. They can be checked in the FINE™/Marine interface.

Time configuration:

Steady

Fluid model:

Salt water at 15°C/Air

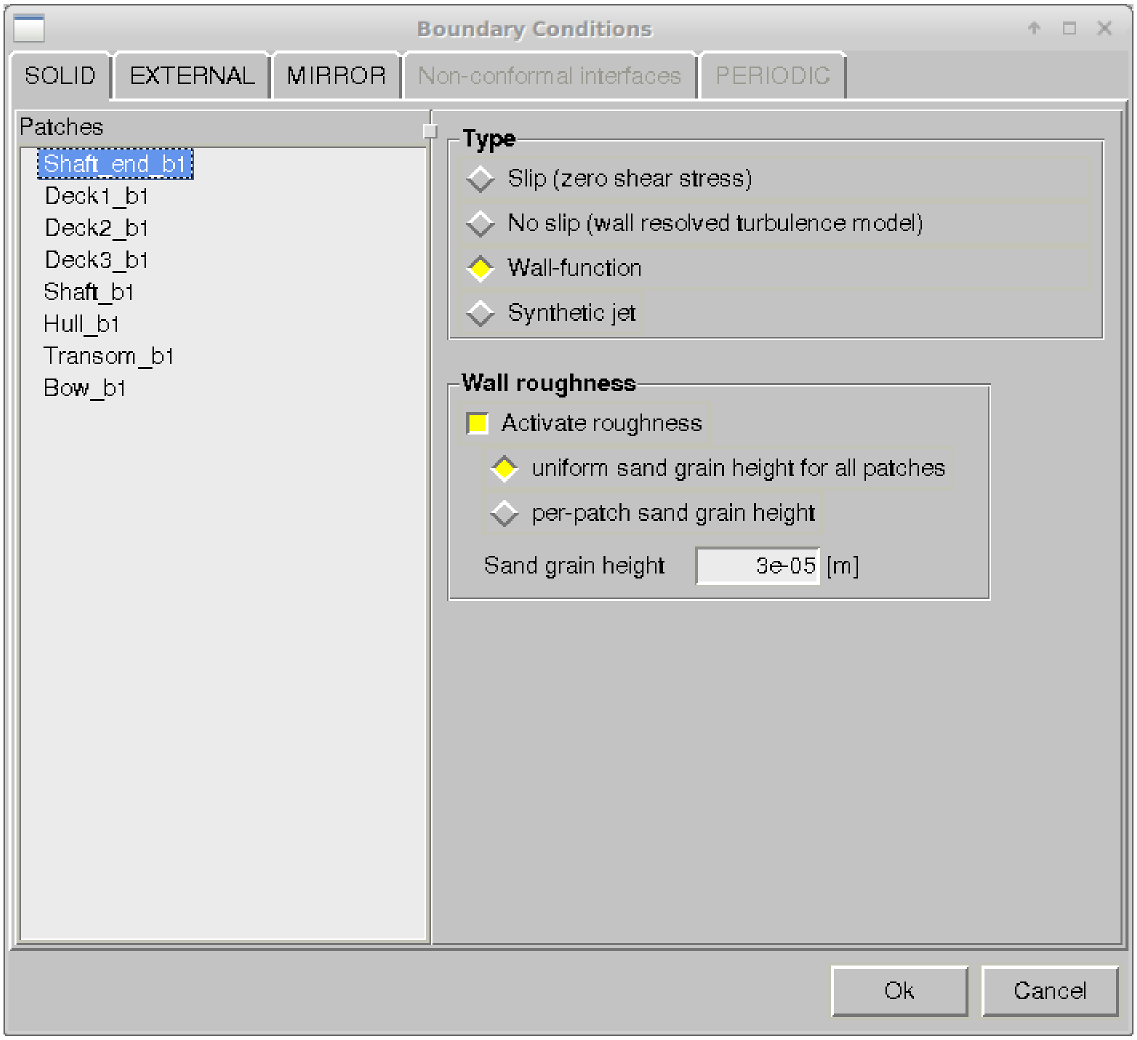
Flow model:

Reference length: 243.844m

Reference velocity: 9.259929m/s

Boundary conditions:

- **SOLID**
  - Deck: *SLIP*
  - all the rest SOLID: *WALL FUNCTION* with a sand grain height of 3e-05m.



- **EXTERNAL**
  - Zmax, Zmin: *PRESCRIBED PRESSURE, updated hydrostatic pressure*
  - Xmin, Xmax, Ymax: *FAR FIELD*
- **MIRROR:** Ymin_SYM

Body definition:

All the solid patches are grouped together and called "Vessel" for the body

Body motion:

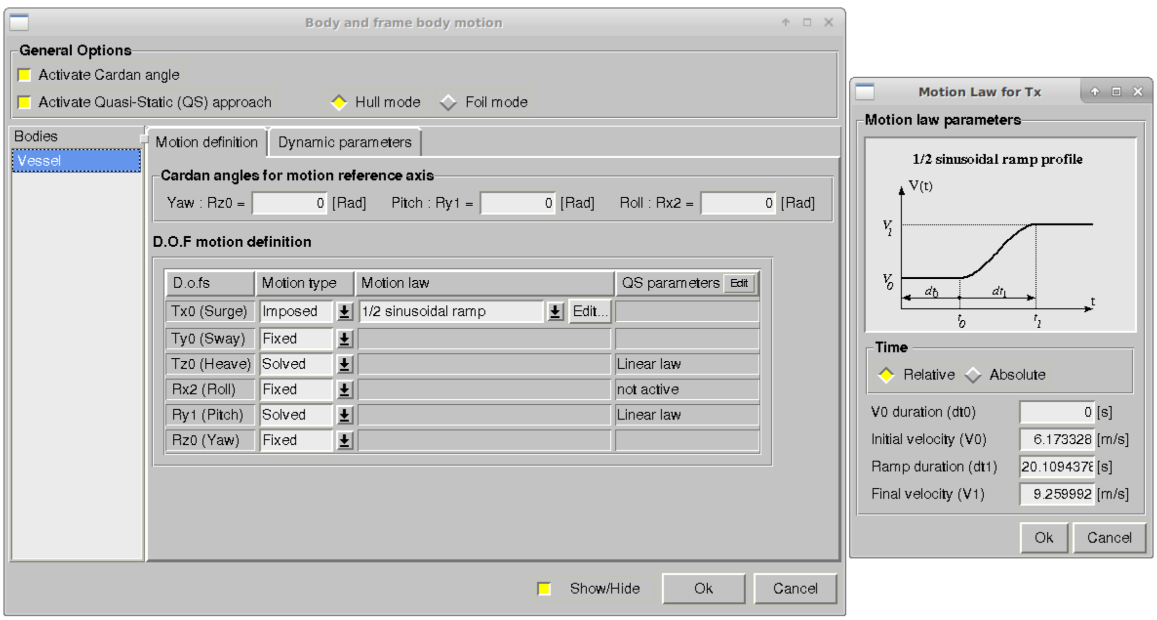
Motion definition:

- **Tz0,Ry1 Solved**: Free trim and sink DOF's
- Tx is imposed as ½ sinusoidal ramp profile: acceleration speed from *6.173328m/s* to the final value of *9.259992m/s*

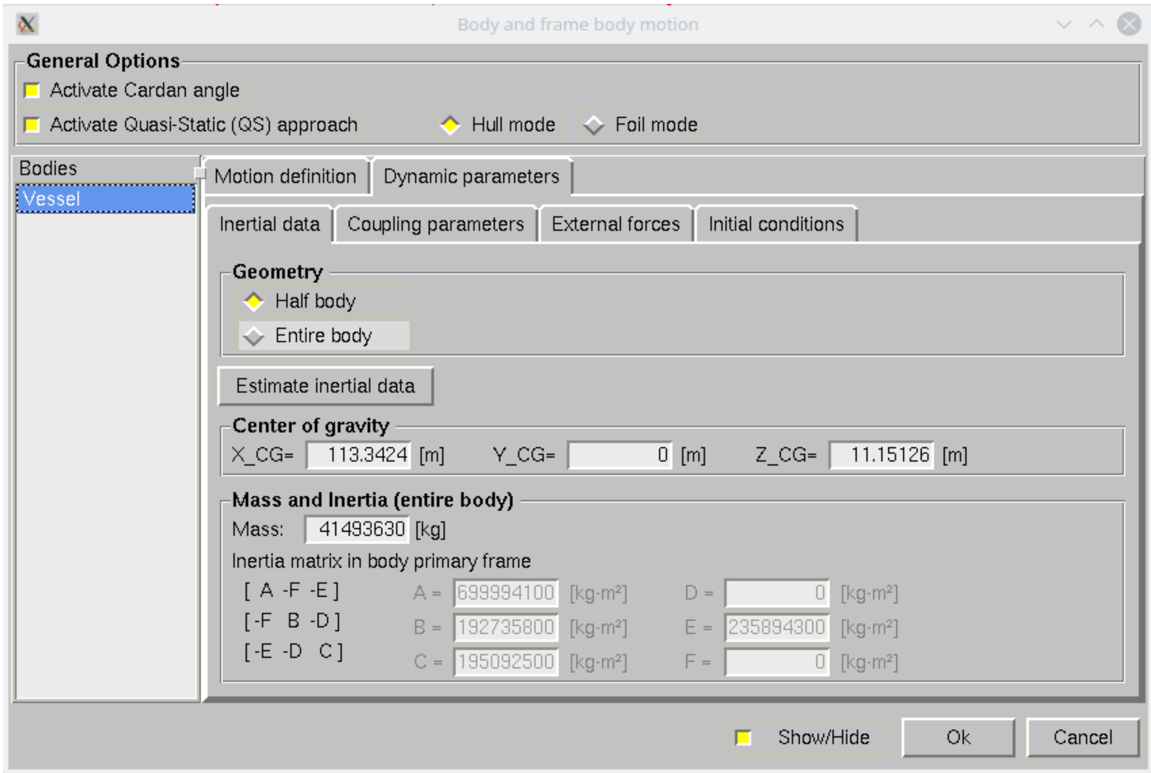
Cardan Angles activated

Quasi-Static (QS) approach activated in Hull mode.

> This method is applied to relax the condition of small time step required by the coupling of the flow motion and the Newton's law. The QS method decreases the CPU time and remain stable even for the larger time step. This method is based on a succession of predicted body attitudes.



Dynamic parameters:



Mesh management: default

Initial solution:

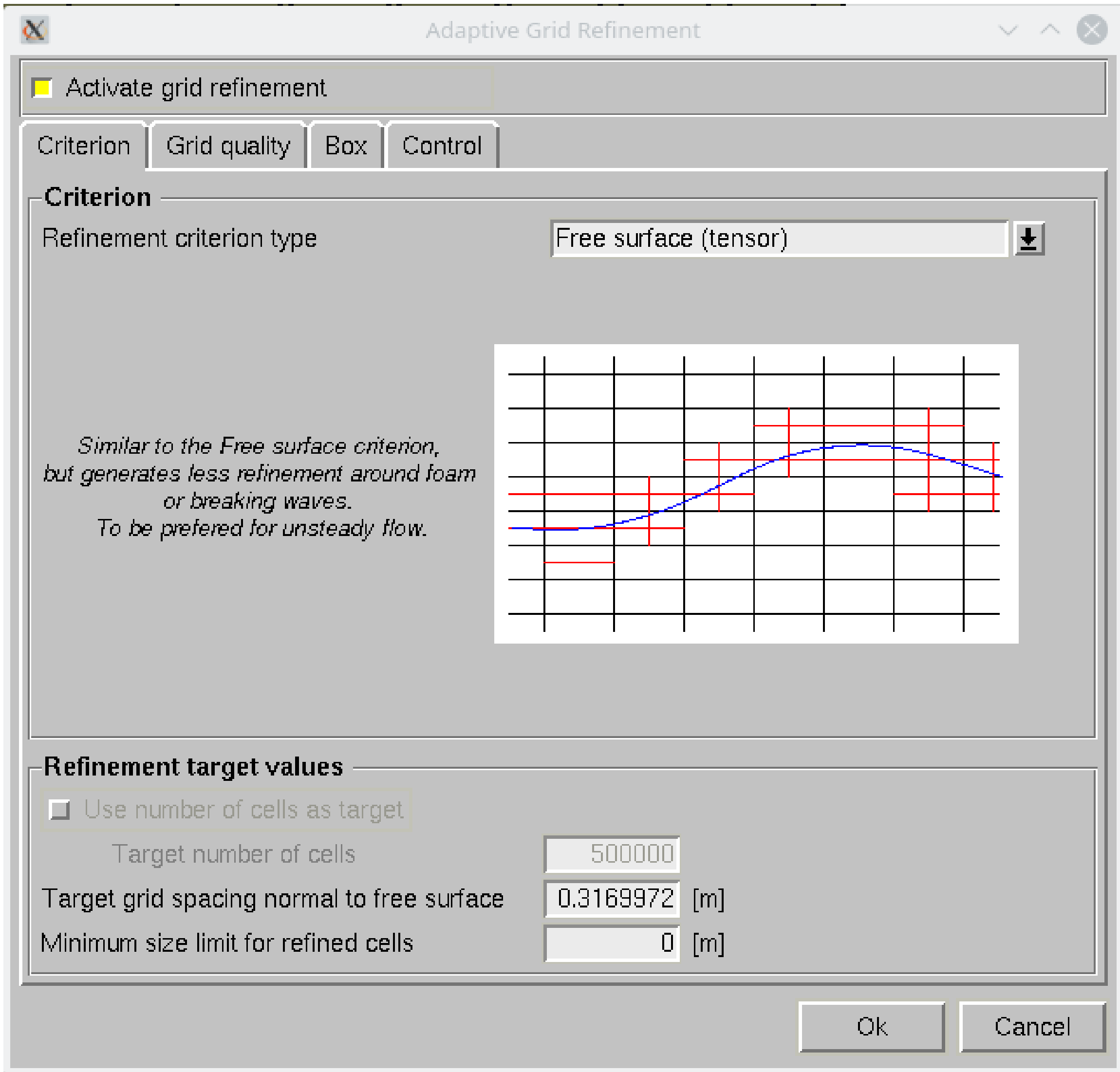
Restart from the previous computation: C1_12.000kt

Additional models:

Actuator disc activated with the settings imposed in the **C-Wizard**

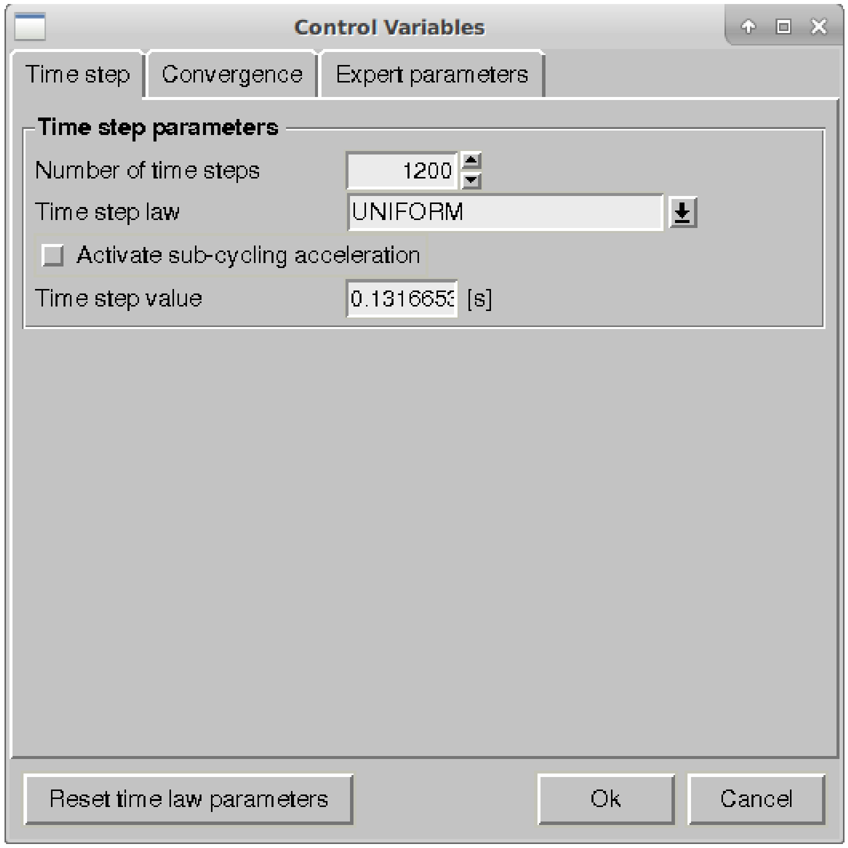
Numerical parameters: Adaptive grid refinement

- *Refinement criterion type:* Free surface (tensor)
- *Target grid spacing normal to free surface:* 0.31967
- *Criterion diffusion:* 2 layers copying full criterion value and 0 for fraction of value
- *Boundary layer protection:* Longitudinal direction only
- *Box:* Directional refinement only (X,Y around the ship area and Z everywhere)
- *Control*
  - 200 steps before the first call of refinement procedure
  - 25 steps between calls to refinement procedure



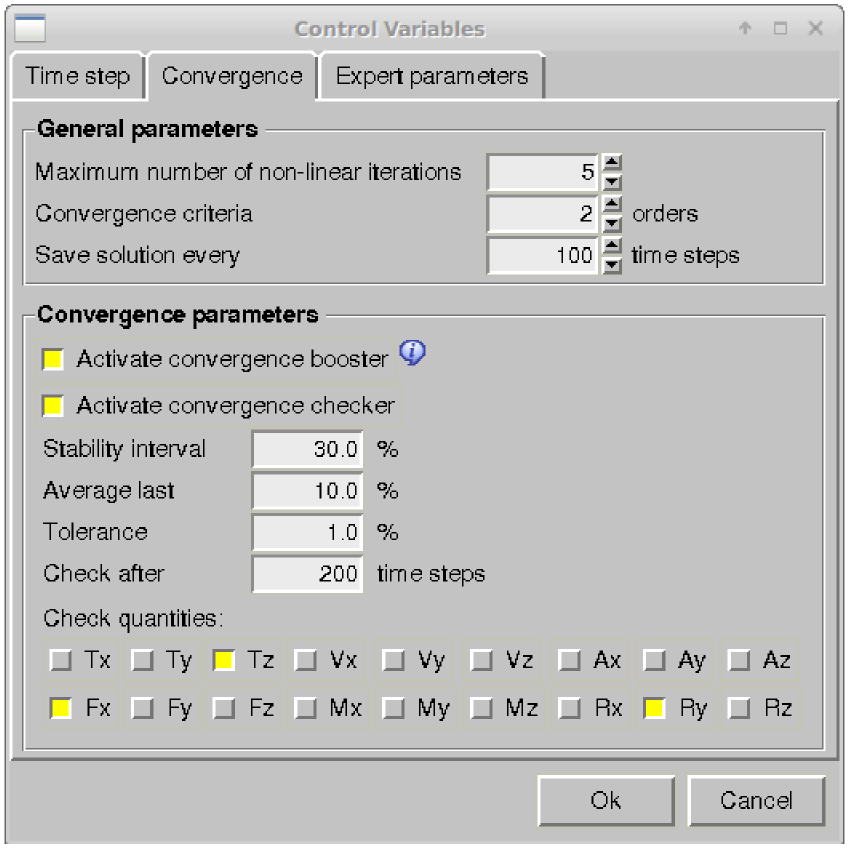
Control variables:

- The **Number of time steps** is set to 1200.
- The **Time step value** is computed based on the reference length and reference velocity:



Convergence tab:

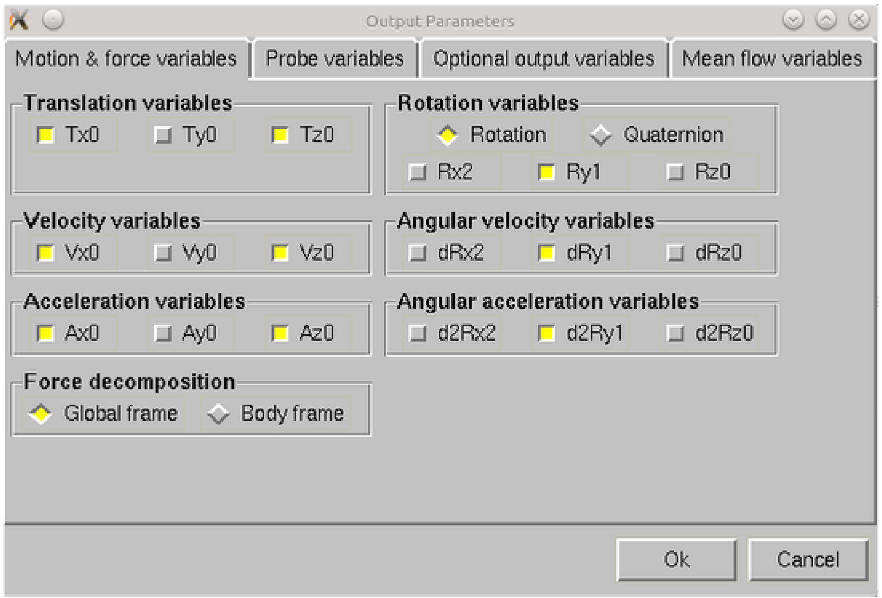
- **Activate convergence booster** is selected.
- The **Convergence checker** is active and will check for the drag, trim and sinkage values oscillations to be less than 1% of their mean value to stop the computation.



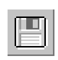
Outputs

Motion & force variables:

- *Translation variables:* Tx0, Tz0
- *Velocity variables:* Vx0, Vz0
- *Acceleration variables:* Ax0, Az0
- *Force decomposition:* Global frame
- *Rotation variables:* Rotation, Ry1
- *Angular velocity variables:* dRy1
- *Angular acceleration variables:* d2Ry1

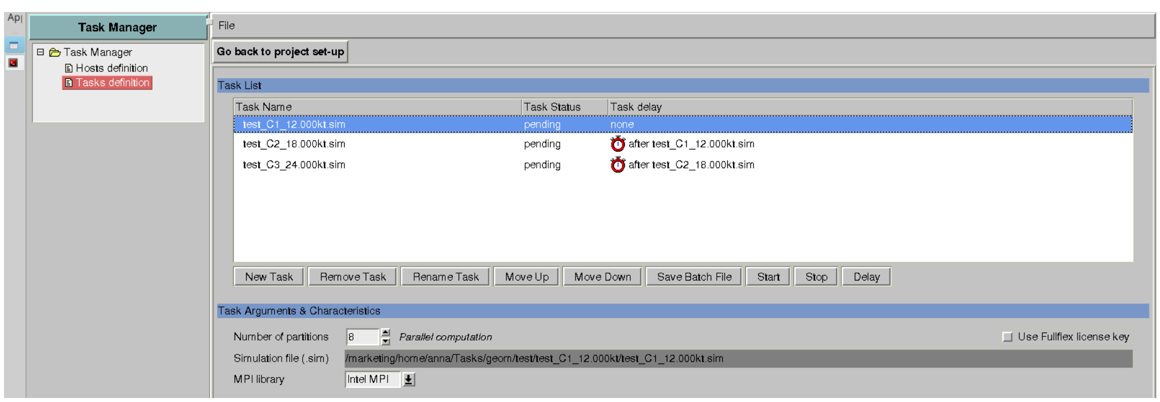


1.4.2 Launch & Control Computation

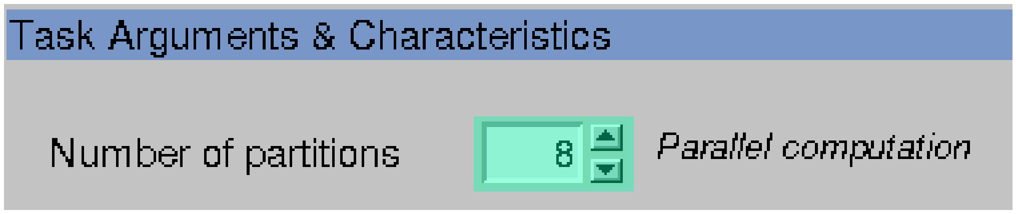
2.1. Click on **Save Project** icon  to save the project.

2.2. Go to the **Task Manager** by clicking on the icon 

2.3. The **Computation resources** options was active in the C-Wizard and now the 3 computations are prepared and queued one after the other:



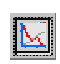
2.4 The number of partitions is automatically set to the earlier specified value of <*8*>, or the maximum number of cores available in the machine. Change this value if needed.



It is not compulsory to set this number of partitions. When set to 7, the computation will take approximately 10,5 hours on Linux 64-bit OS, Intel(R) Core(TM) i7-3370 CPU @ (3.4 GHz, 8 cores, 16 GB RAM)

2.5. Select the computation *C1_12.000kt* and Click on *Start* button to start it.

When the solver finishes the computation, the **Task Manager** will display the computation status it in the **TASK MANAGER INFO** window and the next queued computation will start automatically.

2.6. While the computations are running the **Monitor**  can be used to check the evolution of the quantities.

In the **Quantities to display** menu, it is possible to select the quantities (residuals, forces, moments, motions and actuator disk variables) for which one would like to follow the convergence history or check the computed values.

Multiple components or quantities from different runs can be displayed together as presented below. The button **Add** can be used to open several computations at the same time and compare their results.

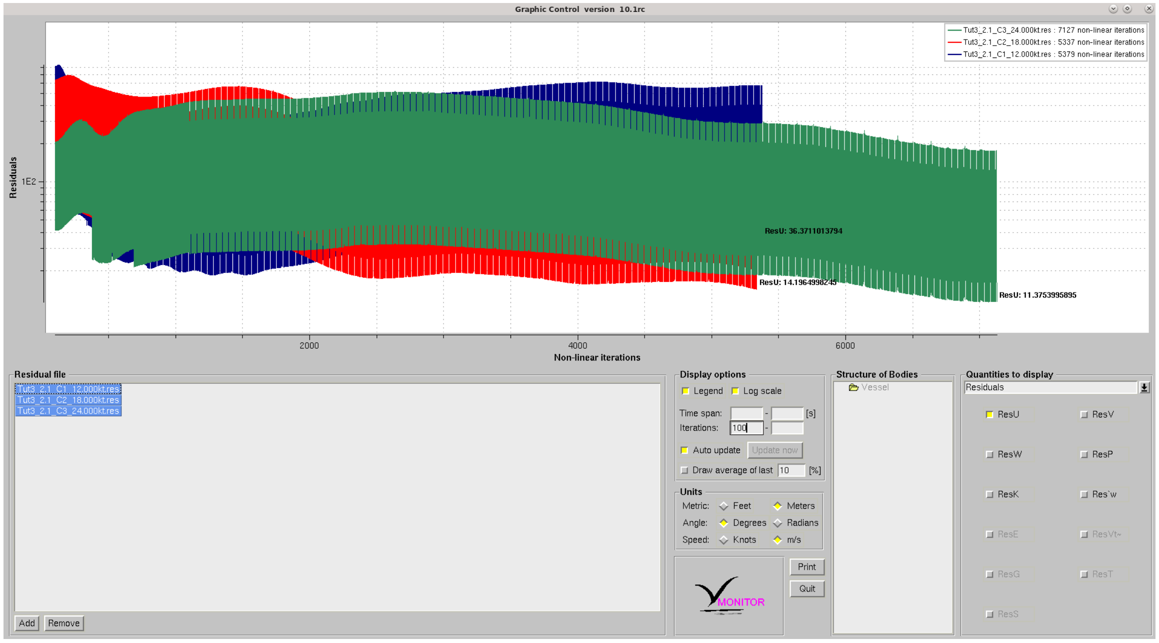


FIGURE 1.1
Residuals evolution

In the **Residuals** page, activate **log scale** to see the orders of magnitude the residuals have decreased. And set the minimum **Iterations** value to remove initial peaks. The residuals by themselves do not give us enough information to check convergence, forces and solved motions should also be checked for stability.

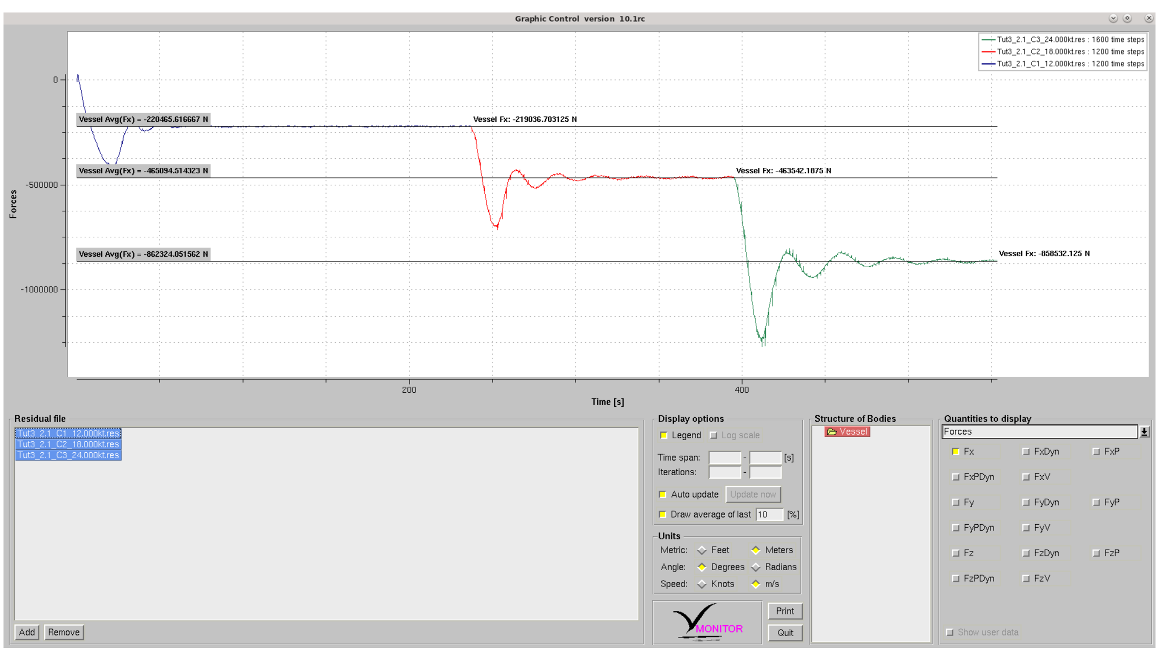


FIGURE 1.2
Drag evolution and averaged value on the last 10% of each computation

The **Draw average of last X%** option should be used over a stable part of the computation to obtain average results and avoid taking the value of a single time step. The **Results analyis tool** will be presented in the next section with more post-processing options.

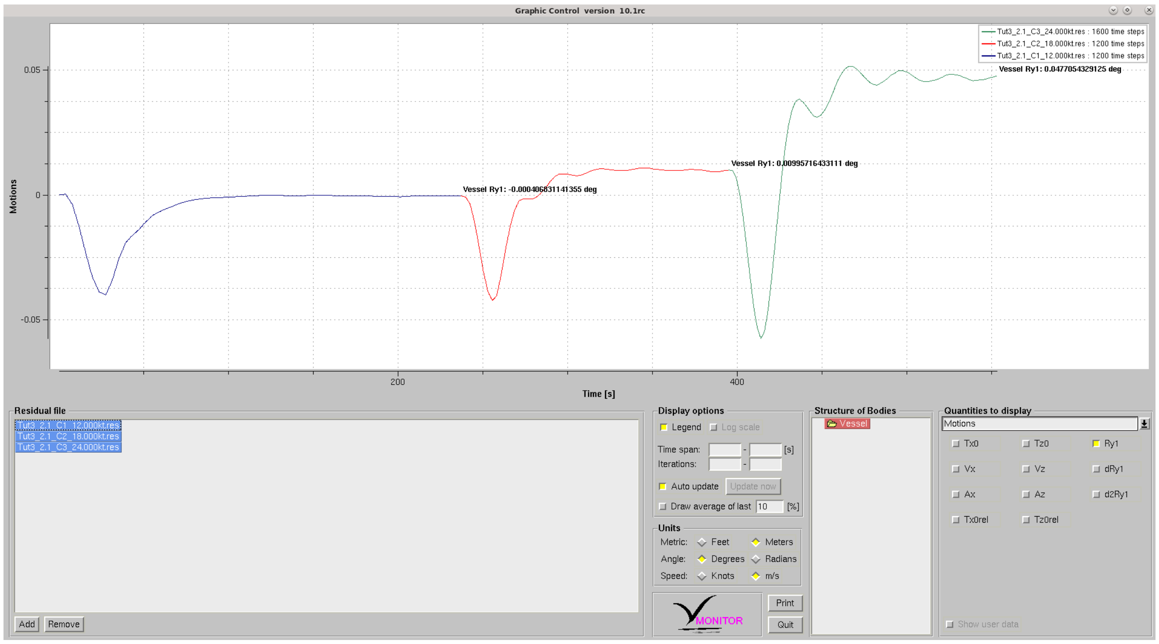


FIGURE 1.3
Trim angle (Ry) evolution in the three computations

The angle units can be switched beteen radians or degrees.

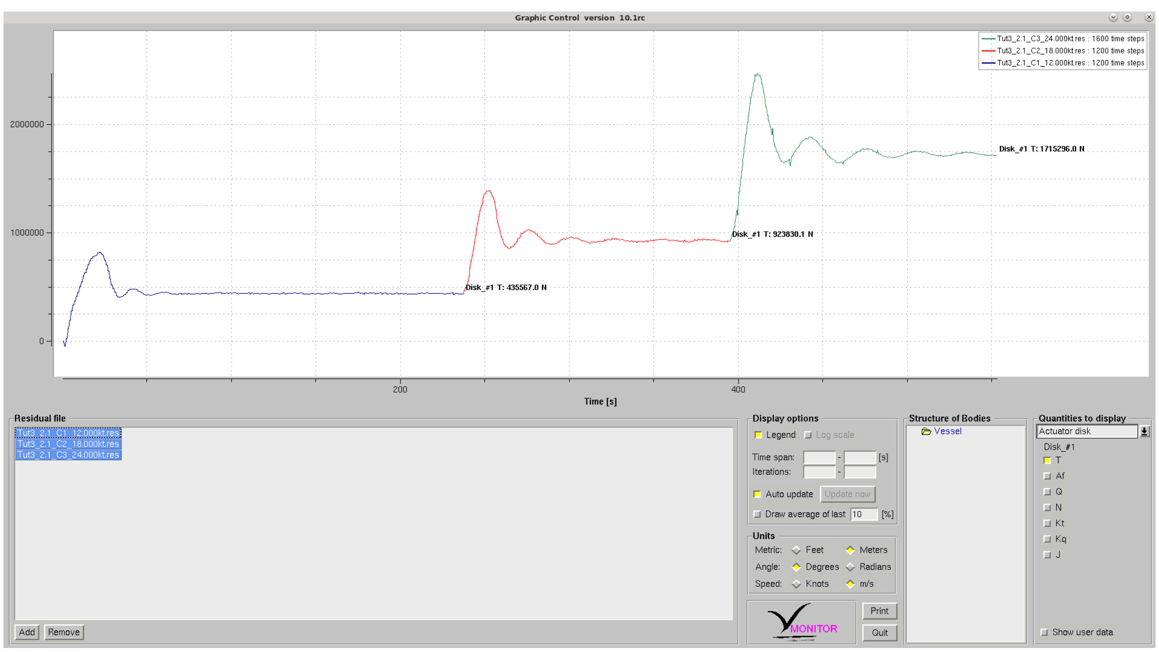


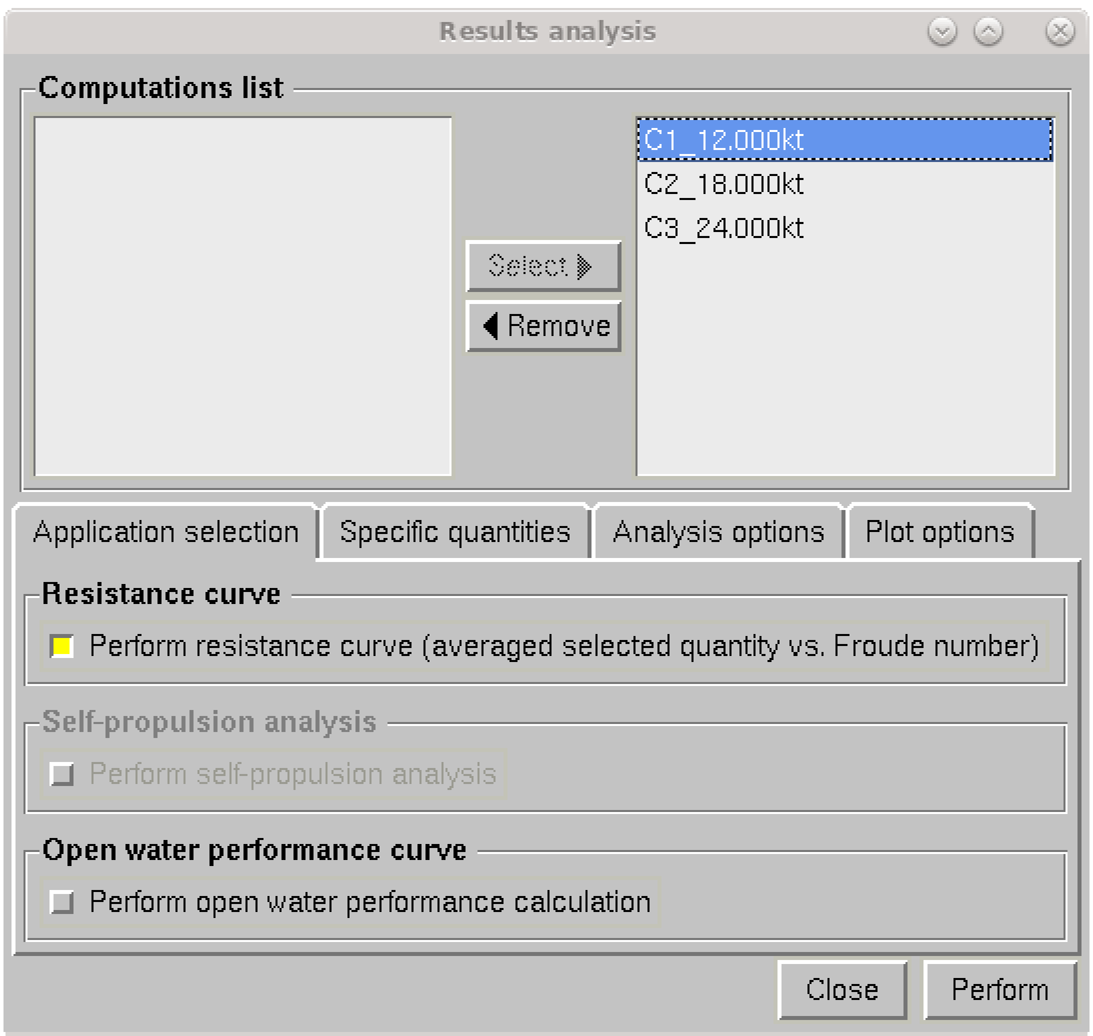
FIGURE 1.4
Actuator disk thrust

In this case the thrust of the actuator disk matches the drag of the hull. Note that the force value is given for half the geometry, while the actuator disk thrust is given for the full disk.

1.4.3 Post Processing

A. Results analysis tool

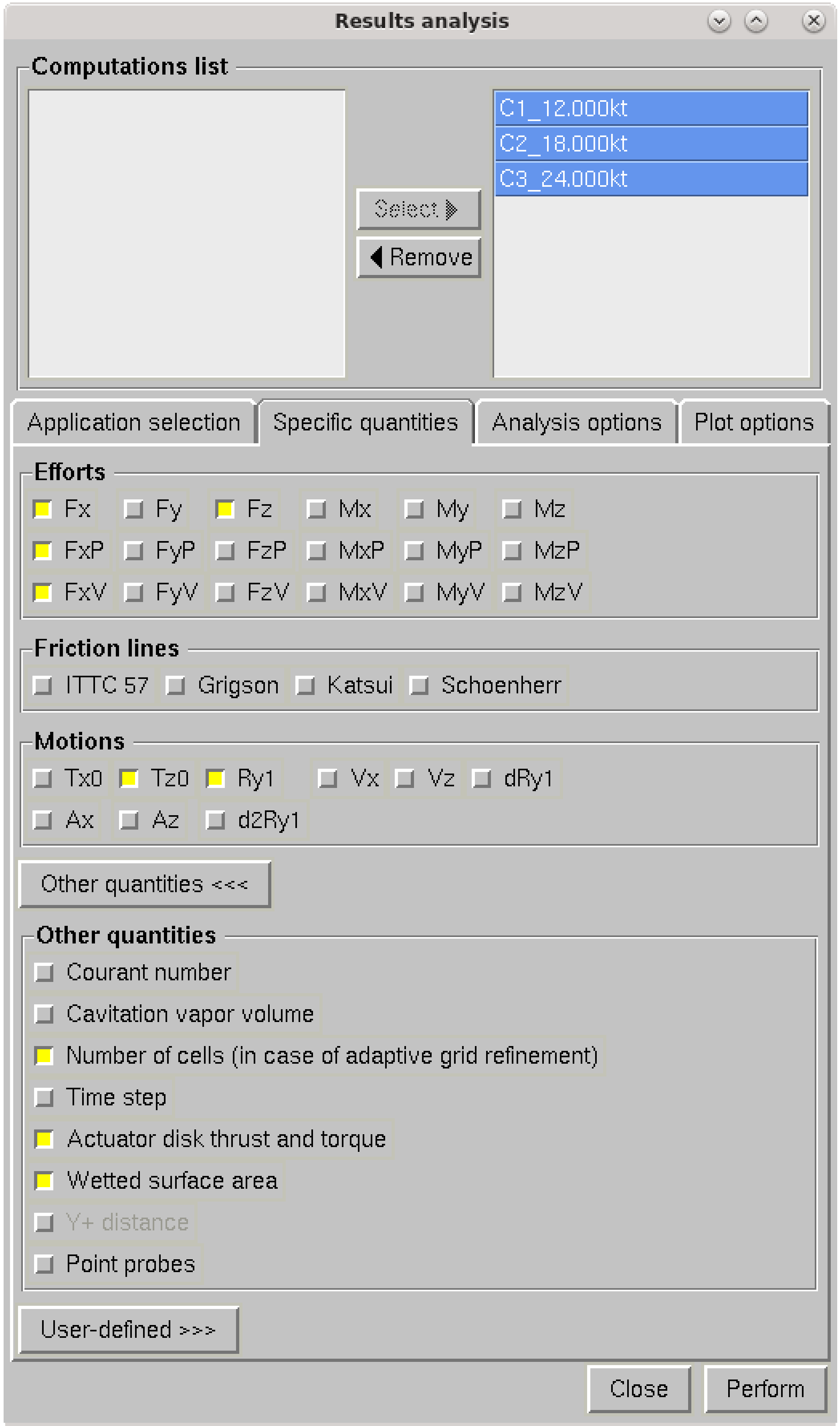
Once the three computations have finished, select them under the list of computations and start the results analysis tool by clicking on the icon .



3.1.1 Activate **Perform resistance curve**

3.1.2 In specific quantities select Fx (total drag), FxV (viscous drag), FxP (pressure drag), Fz and the solved motions Tz0 (sinkage) and Ry1 (trim angle).

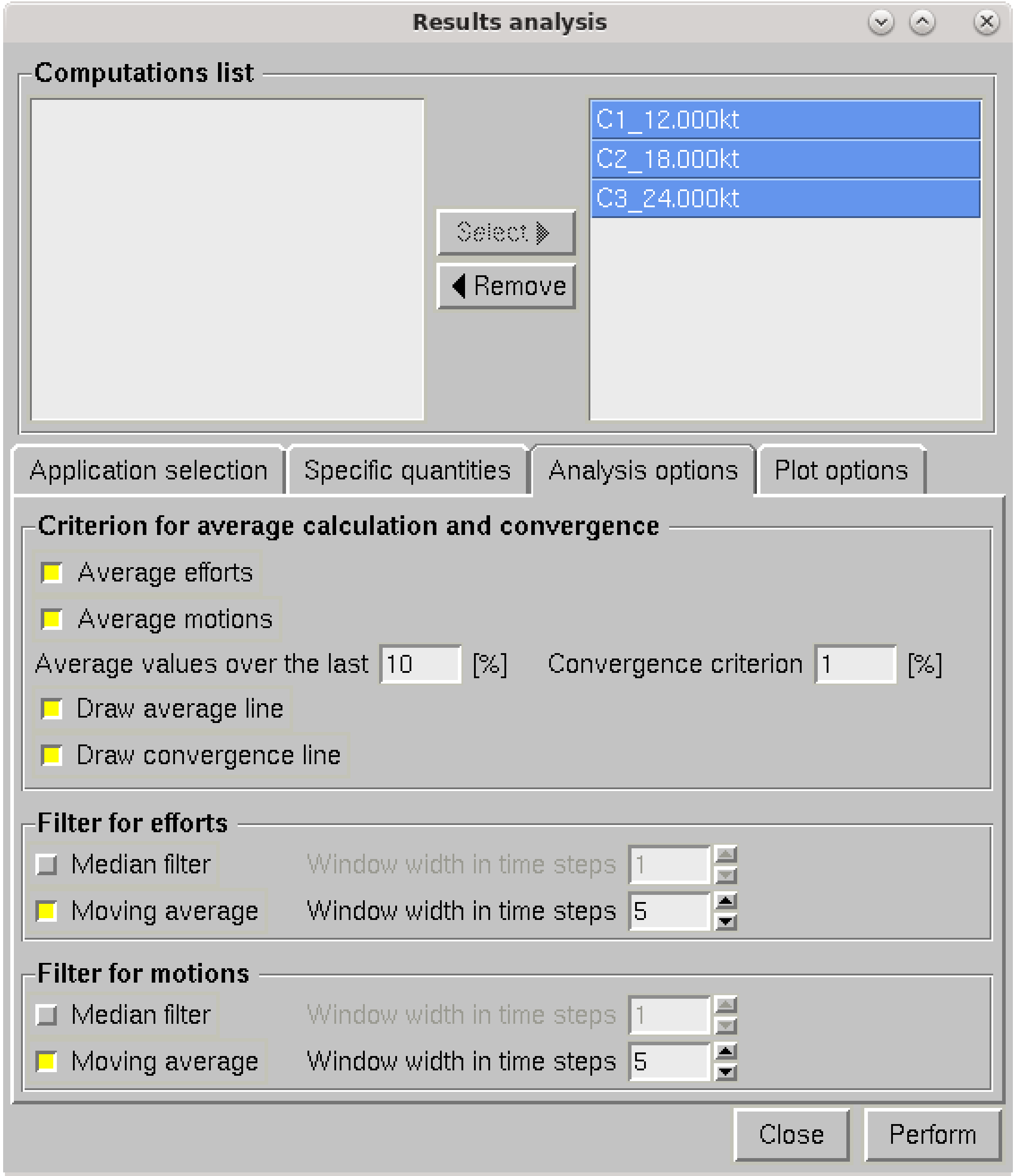
3.1.3 Under **Other quantities** select Number of cells, Actuator disk thrust and torque and Wetted surface area.


Results analysis
Computations list
C1_12.000kt
C2_18.000kt
C3_24.000kt
Select
Remove
Application selection
Specific quantities
Analysis options
Plot options
Efforts
Fx Fy Fz Mx My Mz
FxP FyP FzP MxP MyP MzP
FxV FyV FzV MxV MyV MzV
Friction lines
ITTC 57 Grigson Katsui Schoenherr
Motions
Tx0 Tz0 Ry1 Vx Vz dRy1
Ax Az d2Ry1
Other quantities <<<
Other quantities
Courant number
Cavitation vapor volume
Number of cells (in case of adaptive grid refinement)
Time step
Actuator disk thrust and torque
Wetted surface area
Y+ distance
Point probes
User-defined >>>
Close
Perform

3.1.4 In **Analysis options** activate the options shown in the image:

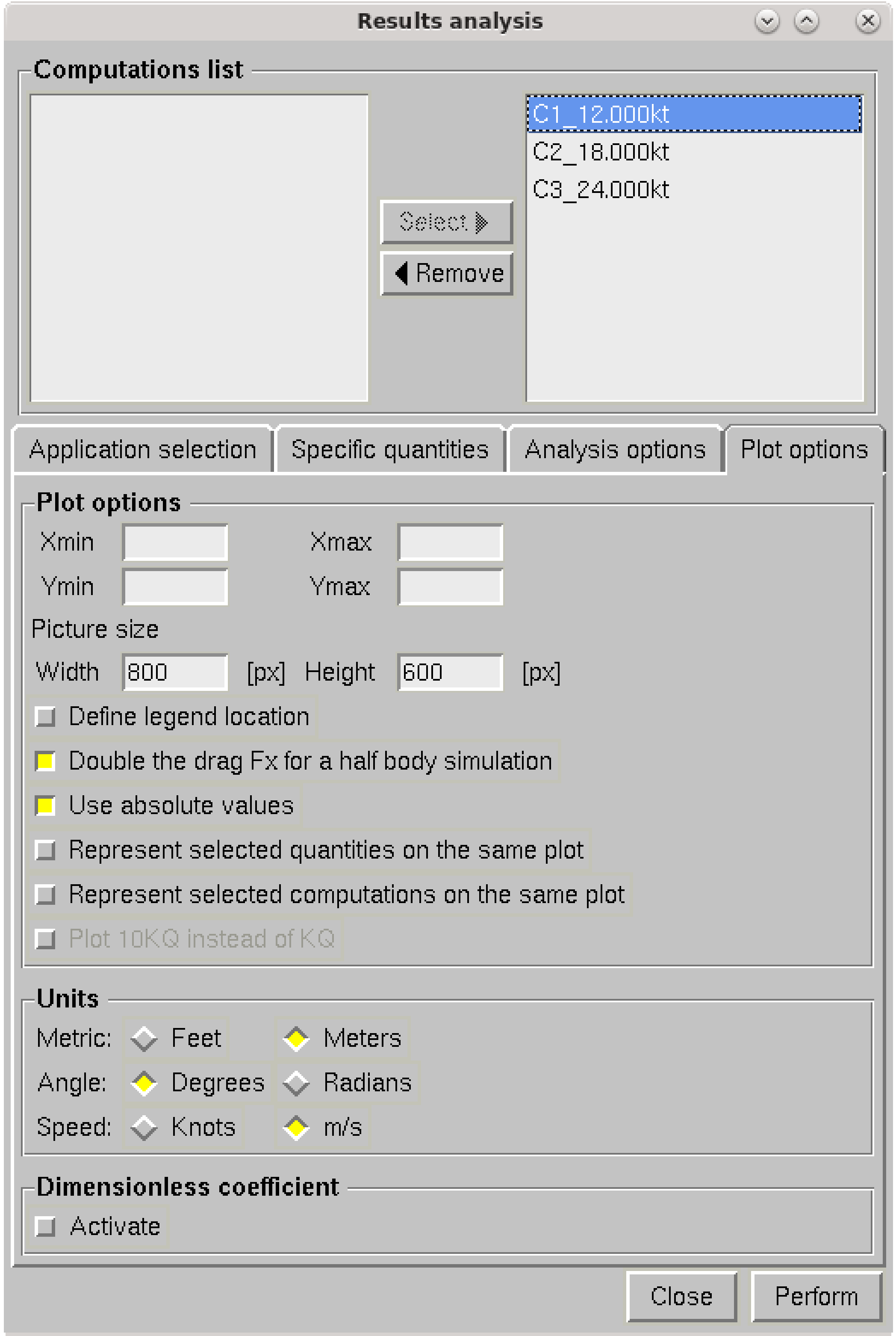
- Average efforts and motions **over the last** 10% with a **Convergence criterion** of 1%. In the plots a green dotted line will be shown where this condition is first met: the amplitude of oscillation of the signal remains below 1% of the average during 10% of the computed time.

  The convergence is checked separately for each quantity selected: some quantities will stabilize very quicly while others might not reach this level of convergence.
- **Draw average line**
- **Draw convergence line**
- Apply a **Moving average** with a window of 5 time steps to efforts and motions to display a smooth signal.



3.1.5 In **Plot options** select

- **Double the drag Fx for a half body simulation** to obtain plots with the complete hull drag.
- **Use absolute values** to have plots showing drag as a positive value.
- Change the angle unit to **Degrees**.



3.1.6 Click on Perform to run the analysis. It will create a new folder in the computation directory named **Convergence_report_<date>_<time>**.

On this folder the user will find the plots of all selected quantities versus Froude number.

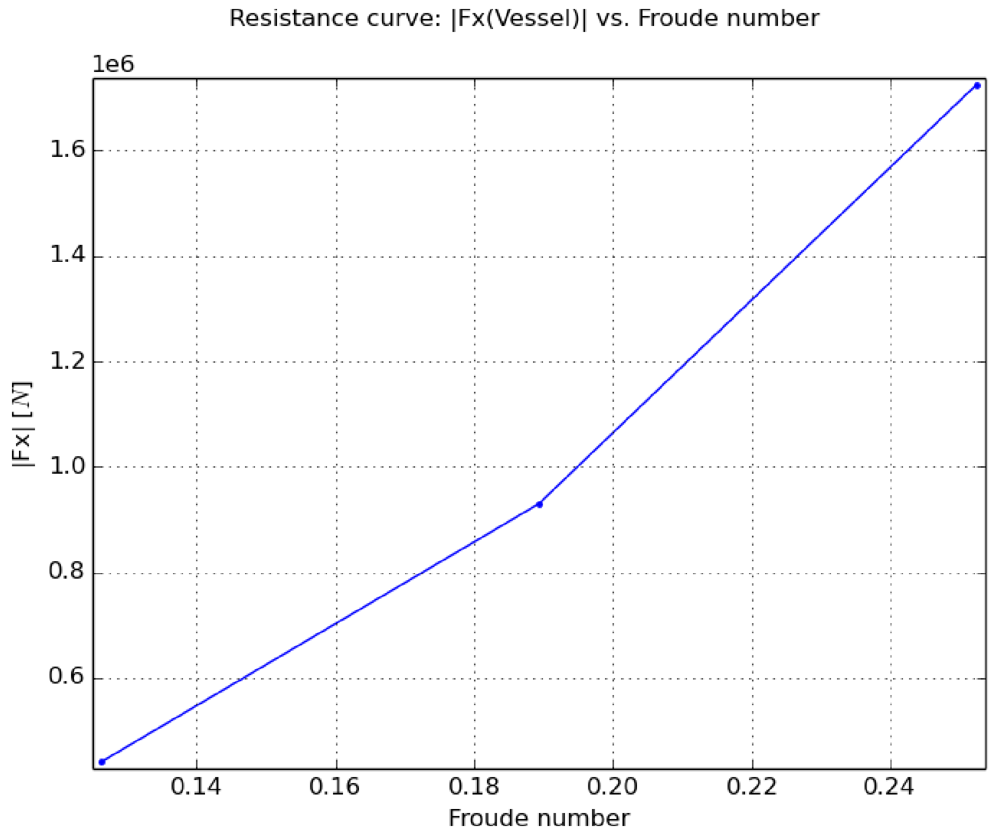


FIGURE 1.5
Drag vs Froude number plot.

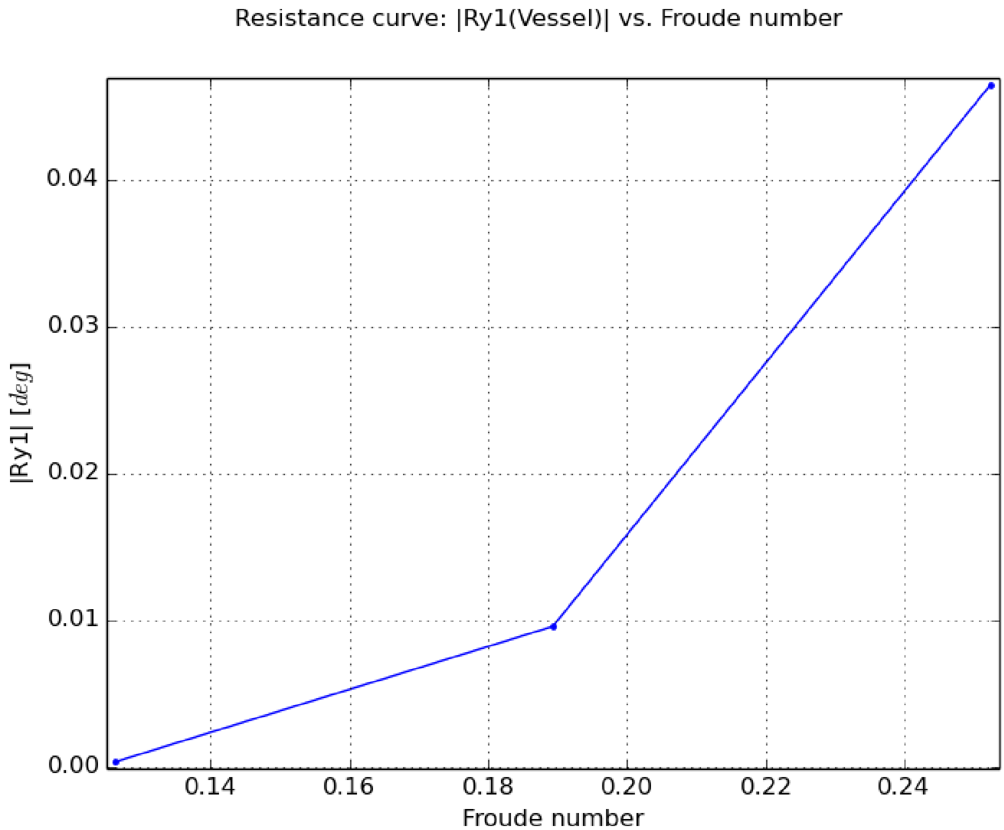


FIGURE 1.6
Trim angle vs. Froude number plot.

Inside the global folder, a folder per computation is created. In this sub-folders, like *C1_12.000kt*, the file **computed_data.dat** is saved. It contains the averaged values for each quantity and the time and number of time steps needed to reach convergence for each quantity. Besides, the plots of each quantity evolution during the computation are stored.

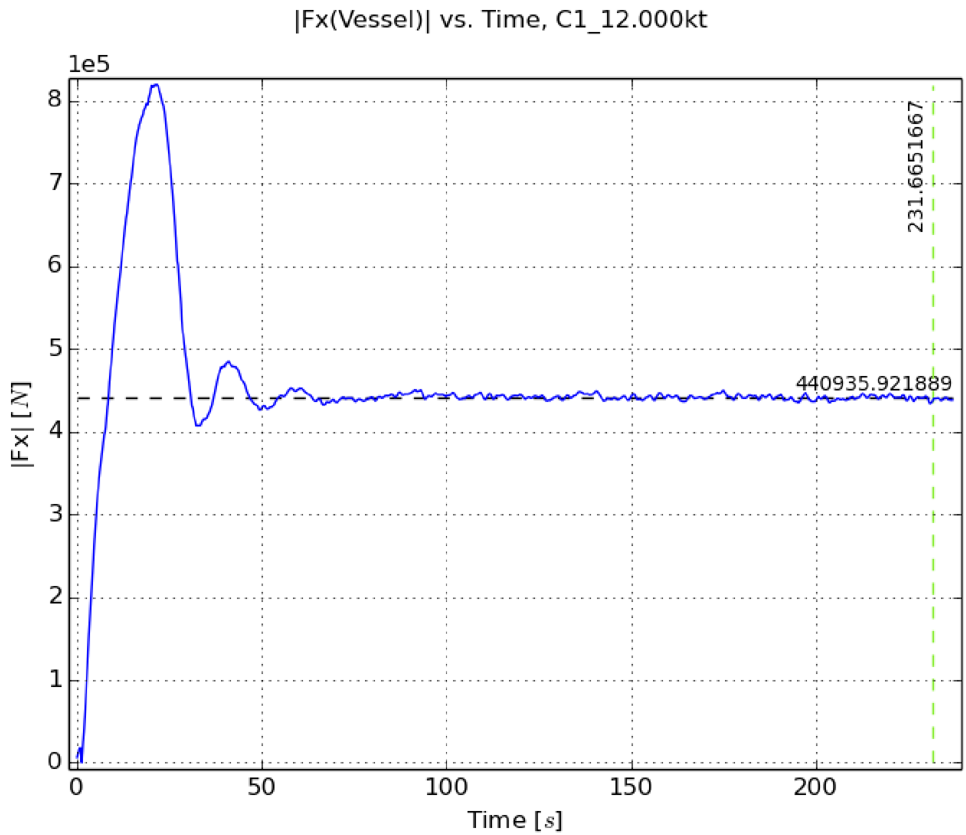


FIGURE 1.7

Evolution of drag during the 12kt computation (blue), average value over the last 10% of time steps (black) and point where convergence for drag was reached (green).

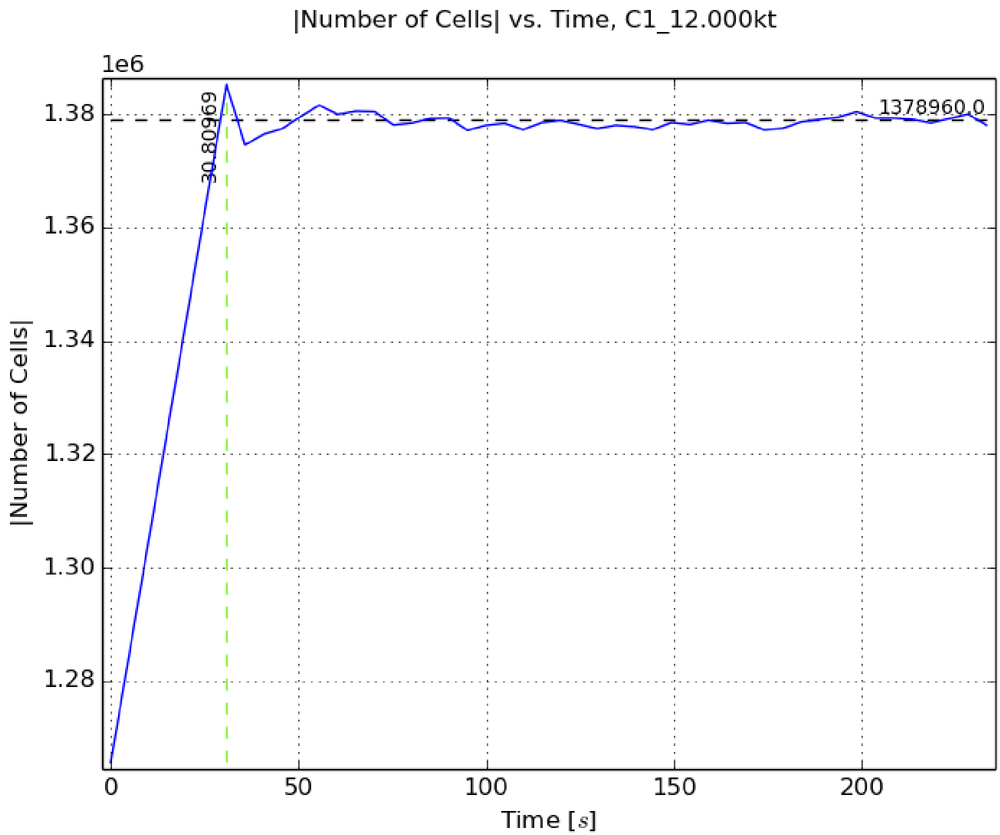
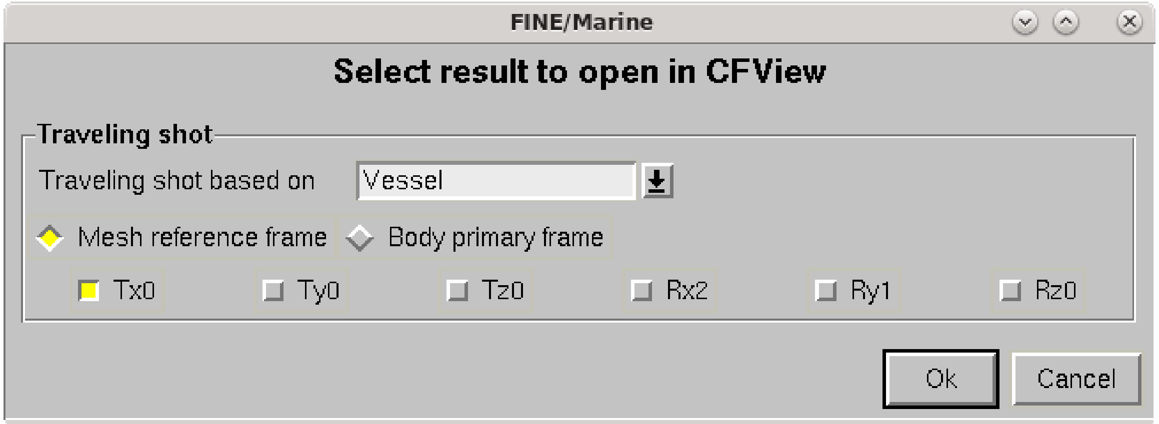


FIGURE 1.8
Evolution of the number of cells in the mesh due to Adaptive grid refinement in the 12kt computation.

B. CFView™

3.2.1. To start the Post-processing, click on the CFView icon of the FINE™/Marine toolbar.

3.2.2. In the appearing selection menu, keep the **Traveling shot based on** Vessel and activate *Tx0* to open. This way the post-processing "camera" will remain centered on the ship. Click the *Ok* button.



3.2.3 Once in CFView, use the Open button to load the other two computation's results. The .cfv file in each computation folder needs to be selected.

3.2.4 Use **Window** > **Tile Views** to show the three computations results at the same time.

3.2.5 Click on one view to select it (the view frame will be highlighted in red) and use **View** > **Maximize** if at some point during the next steps you want to work on only one result.

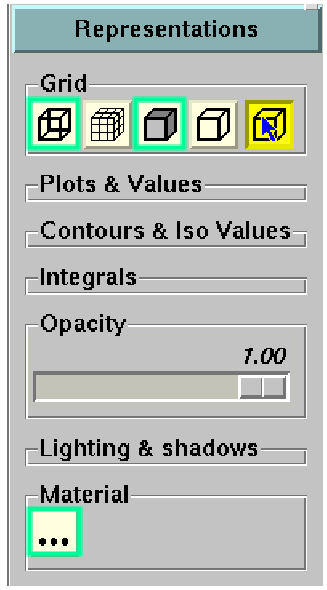
In the current tutorial, the free surface, hydrodynamic pressure and wake flow parameters are chosen for the visualization.

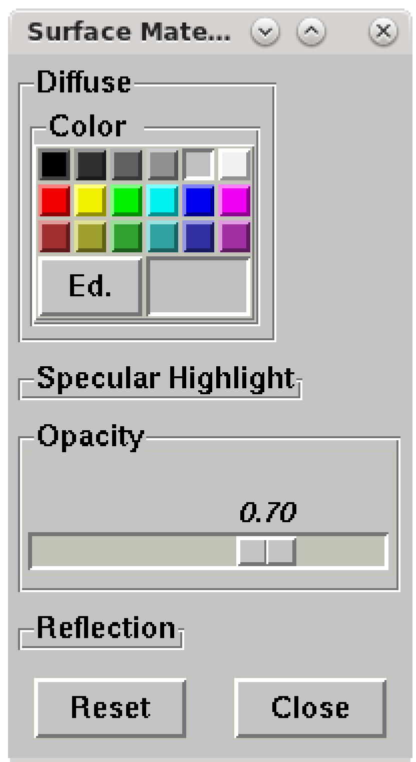
The following steps should be performed on each result:

3.2.6 Mirror the results with *Shift+R*. This option is also available in **Geometry** > **Repetition on/off**.

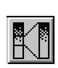
3.2.7 Solid body rendering:

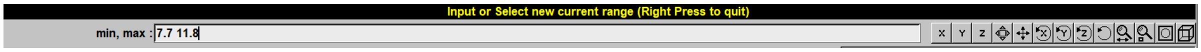
- Use **Macros** > **Group patches by type** and select the Solid patches group
- Remove the boundary edges and activate solid rendering. Under **Material**, open the menu and change the rendering color to a lighter gray. Lower the solid opacity to 70%.



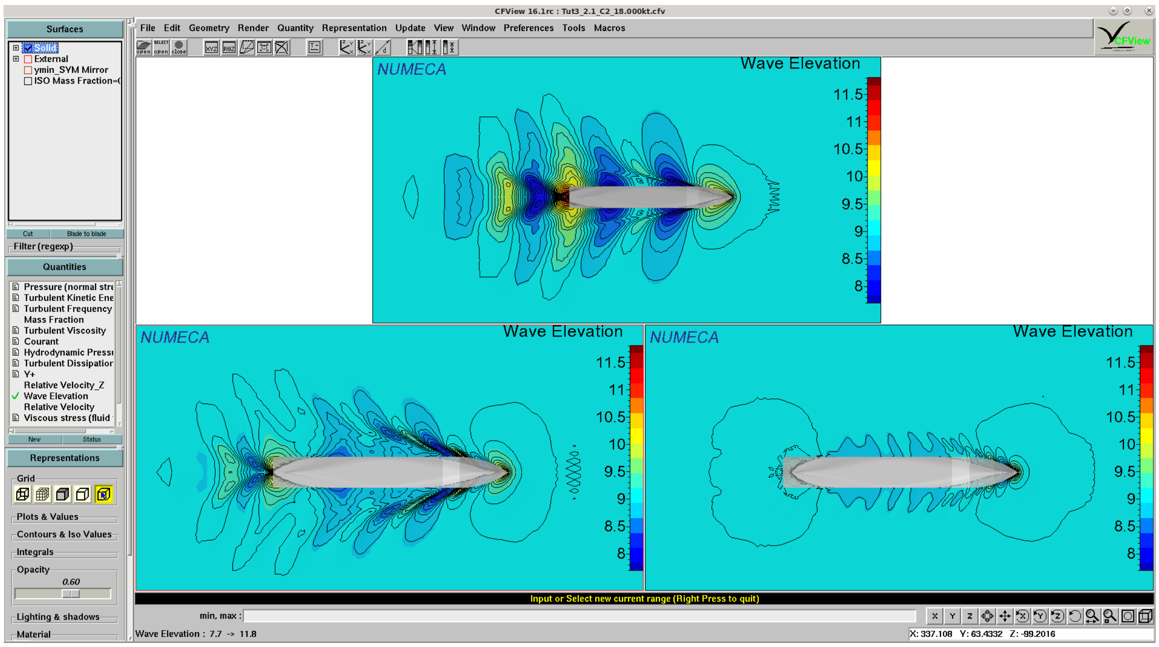


3.2.8 Free surface representation:

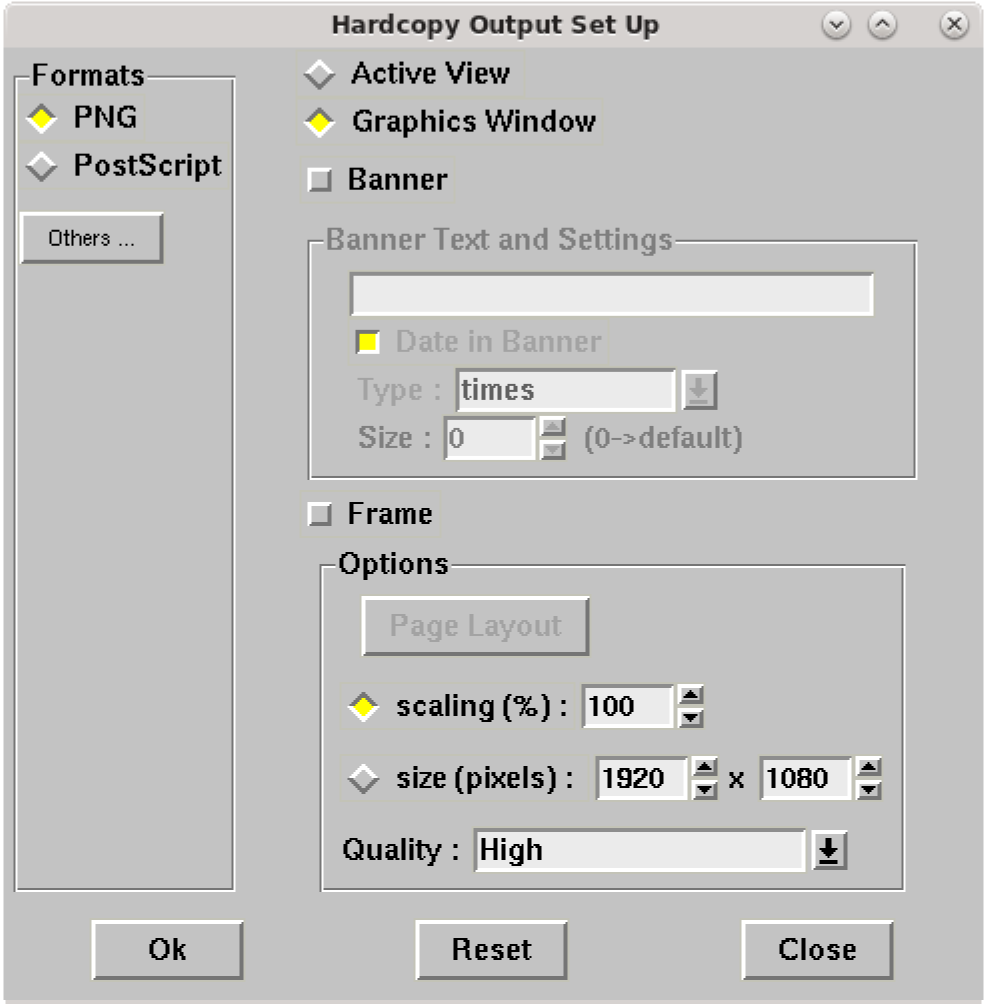
- Use **Macros** > **Represent free surface**
- Use the Set range button  to set the same range from 7.7 to 11.8 in all results. Press *Enter* to apply it.



- To apply the same view in all results, activate one view, click on the = key and select the reference view.

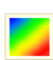


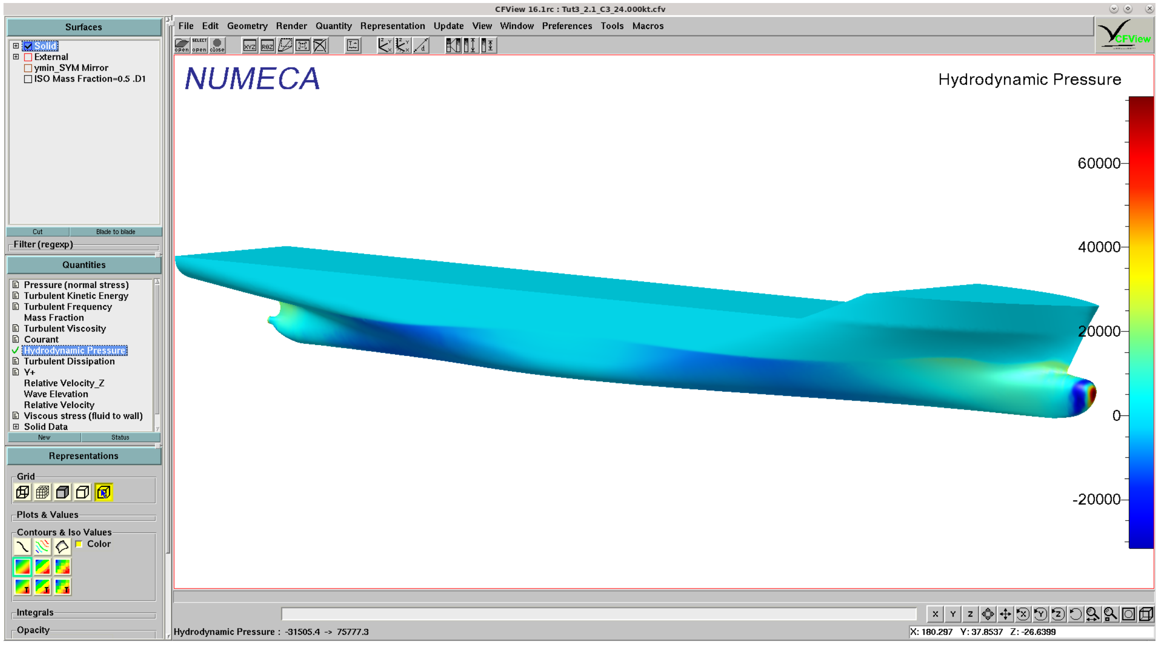
- When all views are ready go to **File** > **Print** to save images. Select **Active View** to save only one result image or **Graphics Window** to save the three results image. Deactivate **Frame** and adjust the **Quality** value depending on the image's purpose.
- All steps performed can be recovered as python macros in **File** > **Macro** > **Save All**. This can become useful to automate post-processing when you gain experience with the program.



- Press *Ctrl+d* or go to **Update** > **Delete** > **All** to restore each view to its initial clean state.

3.2.9 To represent the hydrodynamic pressure, the steps to perform are:

- select all Solid patches on the **Surfaces** menu,
- select **Hydrodynamic pressure** in **Quantities**,
- click on  in the **Contours & Iso Values.**
- Change the colormap with: **Update** > **Colormap** > **Matlab style** > **Reset jet**

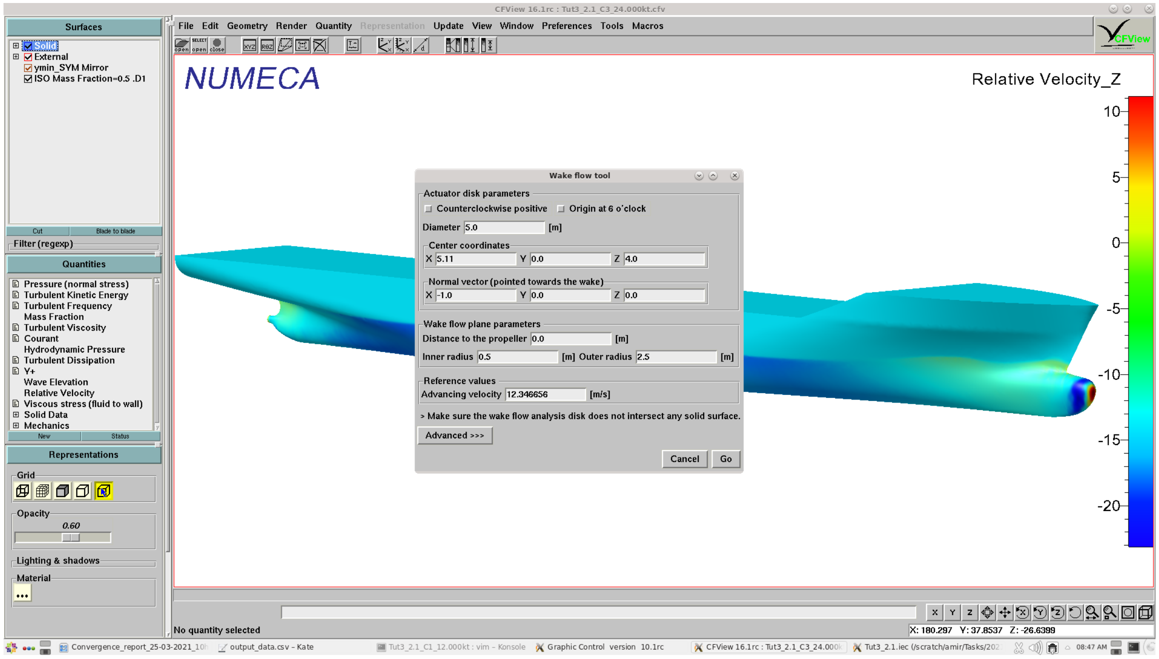


3.2.10 The next step will be using the Wake flow tool to visualize the velocity field in the propeller location.

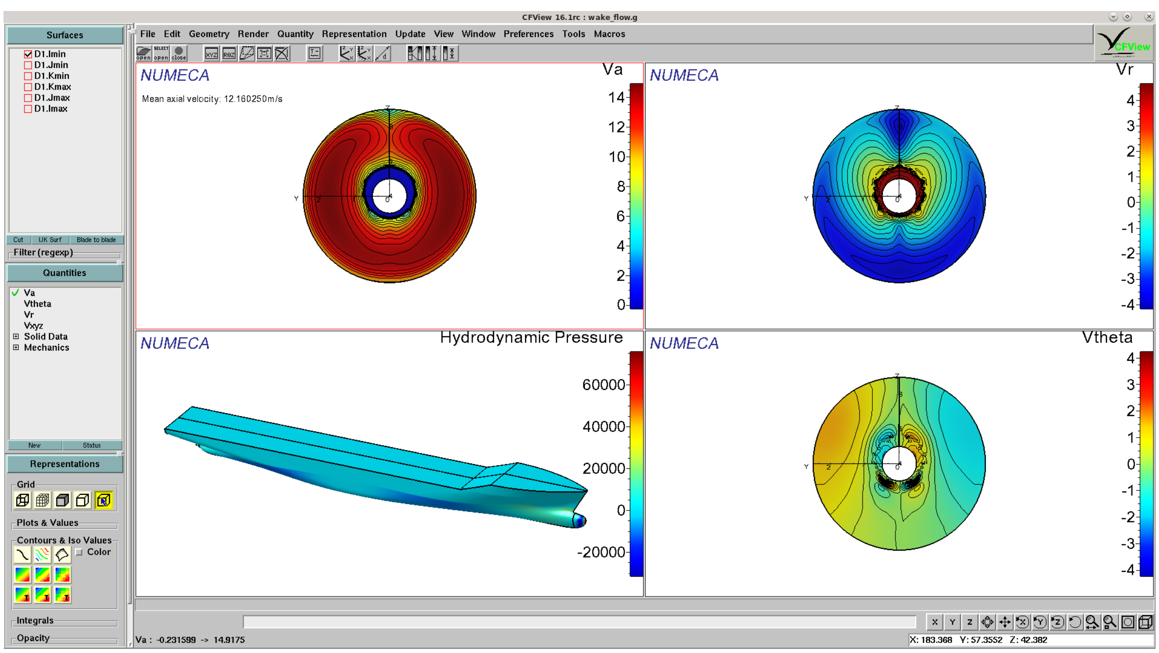
- Use or **Window** > **Close** to close all views except the 24kt result.
- Select the **Macros > Wake_flow_tool**.

> As an **Actuator disk**was activated in the C-Wizard setup, the tool proposes the disk location by default

- Keep these values and press **Go**.



- The tool will recompute the flow field in a cylindrical cut and open three new views with the axial, radial and tangential flow components.
- The information is also stored in the file *wake_flow.txt* in the computation folder.



10.2