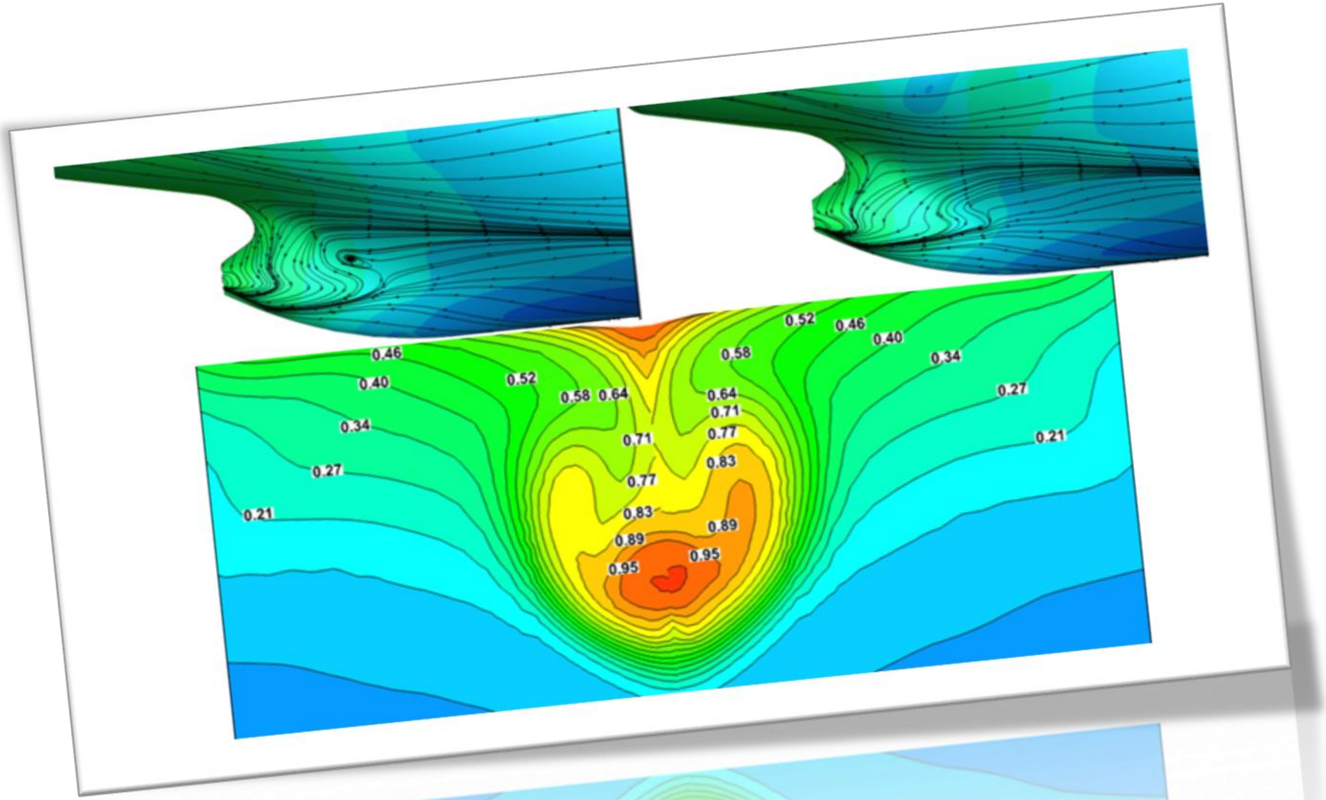
ITU



“A theory is something nobody believes except the person proposing the theory and an experiment is something everybody believes except the person doing the experiment”

Albert Einstein

NUMECA FINE MARINE - DTMB

Ship Hydrodynamics

Contents

- General introduction to CFD
  - Why do we do this tutorial?
  - What is CFD?
  - How it works?
  - Why we use it?
  - Some images of CFD applications
  - Advantages & Disadvantages?
  - Procedure & Methodology
  - Verification & Validation
- A literature study on choosing the different mesh generation, the choice of the turbulence models and the discretization schemes
- Numerical ship hydrodynamics (hull form optimization study)
- DTMB bare hull tutorial with free surface

Why do we do this tutorial?

- Empirical equations
- Wave resistance via Michell’s integral
- Experimental results
- CFD

The aim of this tutorial is to provide step-by-step instructions of the simulation of a typical marine case in order to learning the process of FINE™/Marine. The tutorial gives guidelines/best practices on the complete mesh set-up, flow settings and post-processing, also applicable to other marine cases.

What is CFD?

«CFD is the art of replacing the differential equation governing the Fluid Flow, with a set of algebraic equations (the process is called discretization), which in turn can be solved with the aid of a digital computer to get an approximate solution.»

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, chemical reactions and related phenomena by solving the mathematical equations which govern these processes using numerical process.

The result of CFD analyses is relevant engineering data used in: conceptual studies of new designs detailed product development troubleshooting redesign CFD analysis complements testing and experimentation. Reduces the total effort required in the laboratory.

It is very important to know velocity, pressure and temperature fields in a large no. of applications involving fluids i.e liquids and gases. The performance of devices such as turbo machinery and heat exchangers is determined entirely by the pattern of fluid motion within them.

$$\rho * \frac{\partial w}{\partial t} + \rho * u * \frac{\partial w}{\partial x} + \rho * v * \frac{\partial w}{\partial y} + \rho * w * \frac{\partial w}{\partial z} = -\frac{\partial \rho}{\partial z} + \mu[\frac{\partial w^2}{\partial x^2} + \frac{\partial w^2}{\partial y^2} + \frac{\partial w^2}{\partial z^2}]$$

How it works?

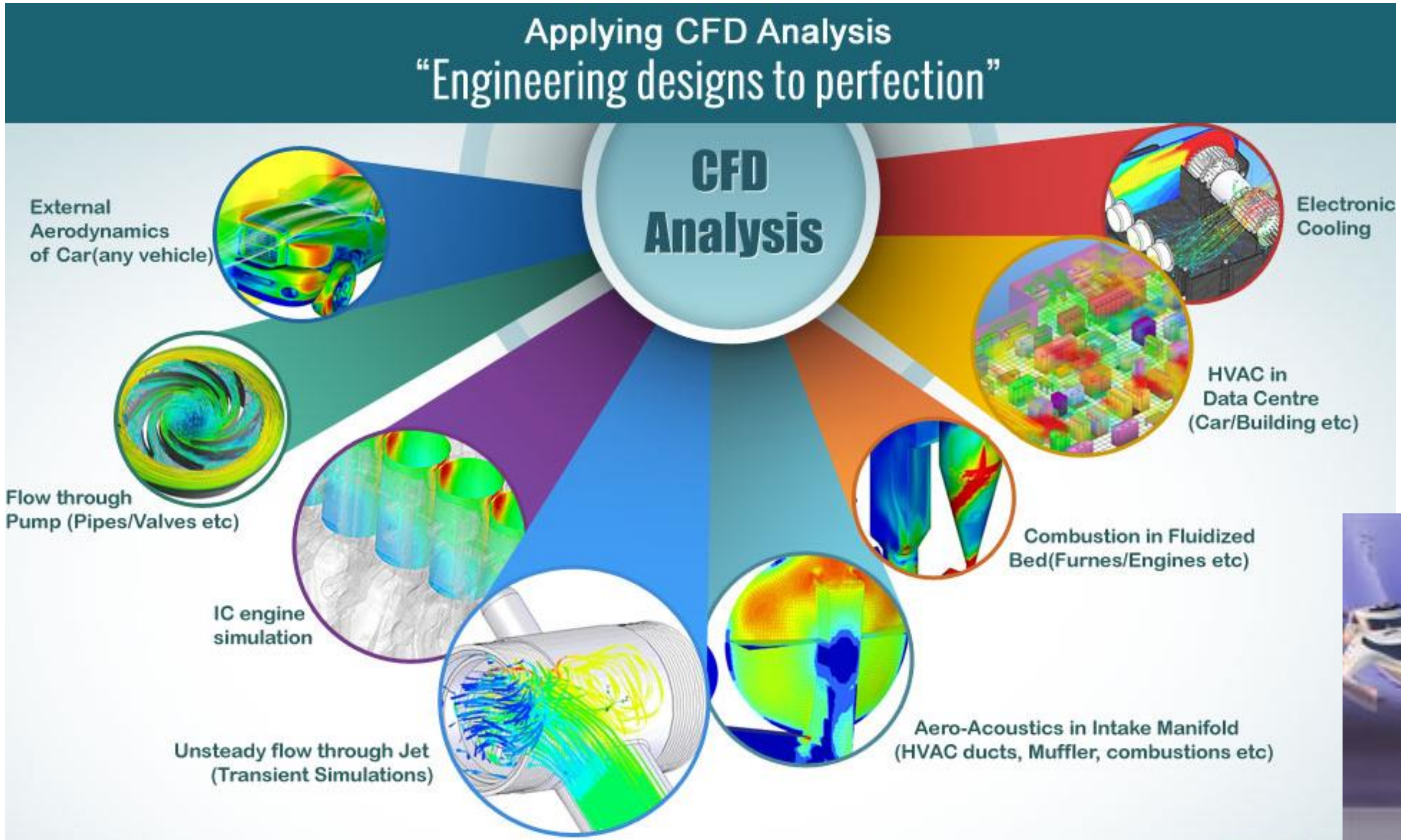
The fundamental basis of almost all CFD problems are the Navier-Stokes Equation, which defines any single phase fluid flow. (By some simplifications equation can be seen above.)

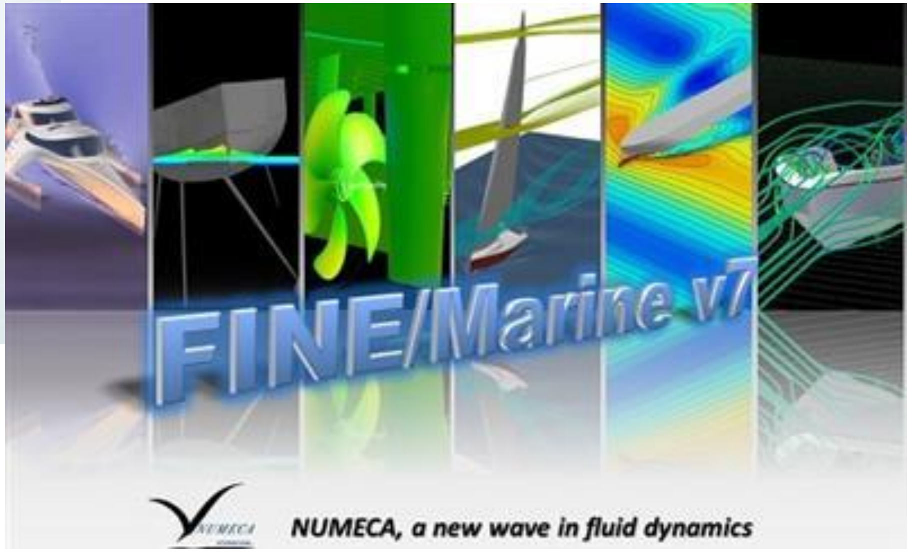
Analysis begins with a mathematical model of a physical problem.

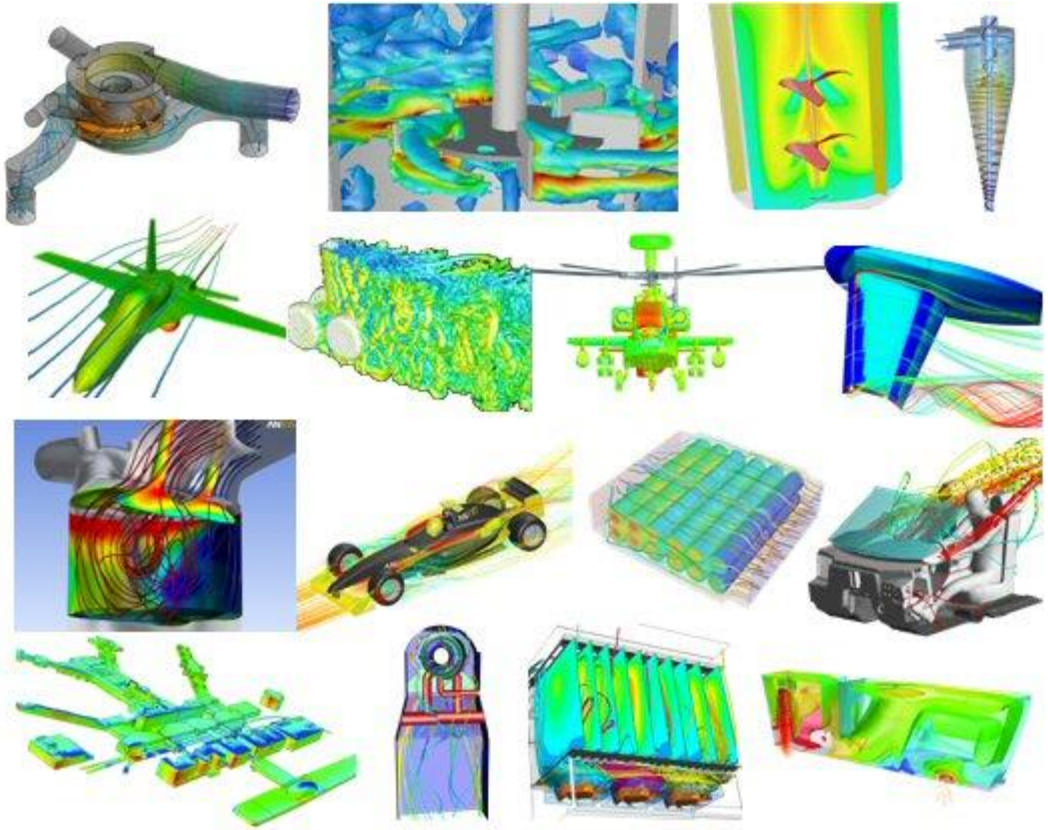
- **Conservation** of matter, momentum and energy must be satisfied throughout the region of interest.
- **Simplifying** assumptions are made in order to make the problem tractable (e.g. Steady-state, incompressible, inviscid, two-dimensional)
- Provide appropriate **initial and/or boundary conditions** for the problem.
- CFD applies numerical methods called **discretization** to develop approximations of the governing equations of fluid mechanics and the fluid region to be studied.
  - Governing differential equations are algebraic and the collection of cells is called the grid or mesh.
- The set of approximating equations are **solved numerically** on a computer for the flow field variables at each node or cell.
  System of equations are solved simultaneously to provide solution.
- The solution is **post processed** to extract quantities of interest (e.g. Lift, drag, heat transfer, separation points, pressure loss etc.)

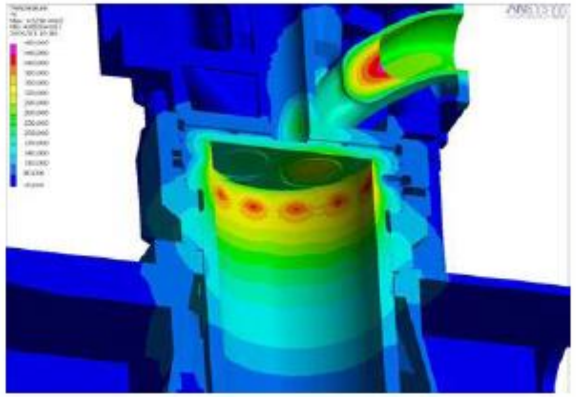
Why CFD?

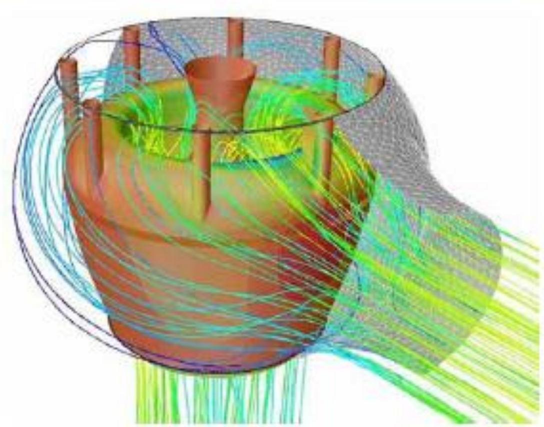
- Absence of analytical solutions
- Need for quick solutions of moderate accuracy
- Complements actual engineering testing
- Reduces engineering testing costs
- Provides comprehensive data not easily obtainable from experimental tests.
- Reduces the product-to-market time and costs
- Helps understand defects, problems and issues in product/process
- Growth in complexity of unsolved engineering problems
- The prohibitive costs involved in performing even scaled laboratory experiments
- Efficient solution algorithms
- Developments in computers in terms of speed and storage
- Serial/parallel/web computing
- Sophisticated pre and post processing facilities

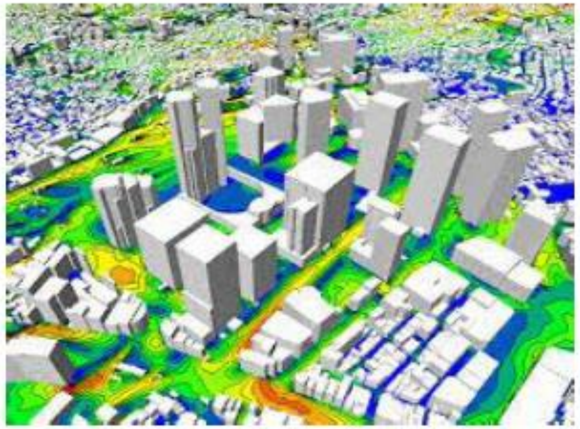

Applying CFD Analysis
“Engineering designs to perfection”
CFD Analysis
External Aerodynamics of Car(any vehicle)
Electronic Cooling
HVAC in Data Centre (Car/Building etc)
Flow through Pump (Pipes/Valves etc)
Combustion in Fluidized Bed(Furnes/Engines etc)
IC engine simulation
Aero-Acoustics in Intake Manifold (HVAC ducts, Muffler, combustions etc)
Unsteady flow through Jet (Transient Simulations)

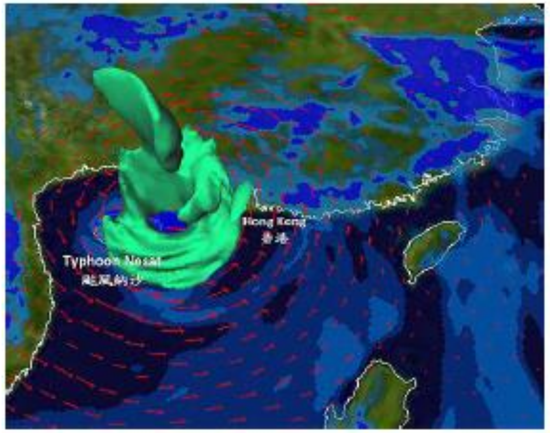

FINE/Marine v7
NUMECA, a new wave in fluid dynamics










Typhoon Nesat
Hong Kong

Advantages & Disadvantages

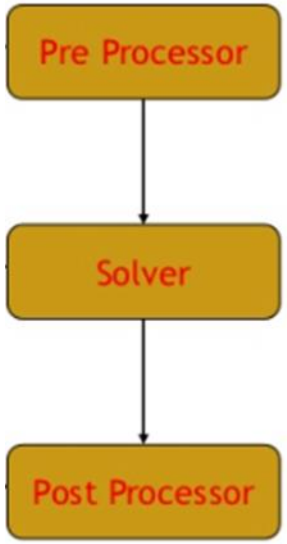
Comparison of experimental, analytical and numerical methods of solution

Name of Method	Advantages	Disadvantages
Experimental	Capable of being most realistic	Equipment required
		Scaling problem
		Measurement difficulties
		Probe errors
		High operating costs
Analytical	Clean, general information which is usually in formula form	Restricted to simple geometry and physics
		Usually restricted to linear problems
		Cumbersome results-difficult to compute
Numerical	No restriction to linearity	Truncation and round-off errors
	Ability to handle irregular geometry and complicated physics	Boundary condition problems
	Low cost and high speed of computation	

- CFD today is equal partner with pure theory and pure experiment in the analysis and solution of fluid dynamic problems.
- It nicely and synergistically complements the other two approaches of pure theory and pure experiment, but it will never replace either of these approaches.
- CFD carry out numerical experiments. Numerical experiments carried out in parallel with physical experiments in the laboratory can sometimes be used to help interpret physical experiment.

Procedure

- Virtual model
- The flow region or calculation domain is divided into a large number of finite volumes or cells
- Partial differential equations are discretized using a wide range of techniques: finite difference, finite volume or finite element
- Algebraic equations gathered into matrices which are solved by an iterative procedure
- Numerical solution gives the values of the dependent variables at discrete locations



- Pre processor
  - Geometry generation
  - Geometry cleanup
  - Meshing
- Solver
  - Problem specification
  - Additional models
  - Numerical computation
- Post Processor
  - Line and Contour data
  - Average Values
  - Report Generation

Verification & Validation

- Verification and validation increase our confidence in the simulation
- No computer software can be proved to have no errors.
- We can state that software is wrong if evidence to this effect can be collected
- Verification is solving the chosen equations right
- Numerical techniques for verification involves finding out sources of error in spatial & temporal discretization, iterative convergence, and rounding off errors
- Checking out if time steps adequate for all situations
- Validation is solving the right equation
- Is the simulation matching with experimental data & Scientific literature
- Experimental data helps validation of similar simulations

Attention!

CFD is a powerful tool to solve complex flows in engineering systems. However:

Extreme care should be taken while:

- Generating geometry and grids
- Choosing flow model (turbulence models)
- Boundary conditions
- Material properties
- Convergence criteria (grid independence)

Unless proper inputs are given and solution is checked, the solution we get may not be the real solution!

Turbulence modeling?

Type	Flow type	Reynolds Number
Internal	Laminar regime	up to Re=2300
	Transition regime	2300<Re<4000
	Turbulent regime	Re>4000
External	Laminar to Turbulence	$Re>3\times10^5$

Whenever turbulence is present in a certain flow it appears to be the dominant over all other flow phenomena. That is why successful modeling of turbulence greatly increases the quality of numerical simulations. All analytical and semi-analytical solutions to simple flow cases were already known by the end of 1940s. On the other hand **there are still many open questions on modeling turbulence and properties of turbulence it-self. No universal turbulence model exists yet.**

Solving CFD problem usually consists of four main components: geometry and grid generation, setting-up a physical model, solving it and post-processing the computed data. The way geometry and grid are generated, the set problem is computed and the way acquired data is presented is very well known. Precise theory is available. Unfortunately, **that is not true for setting-up a physical model for turbulence flows. The problem is that one tries to model very complex phenomena with a model as simple as possible. Therefore an ideal model should introduce the minimum amount of complexity into the modeling equations, while capturing the essence of the relevant physics.**

Complexity of different turbulence models may vary strongly depends on the details one wants to observe and investigate by carrying out such numerical simulations. Complexity is due to the nature of Navier-Stokes equation (N-S equation). N-S equation is inherently **nonlinear, time-dependent, three-dimensional PDE**. Turbulence could be thought of as instability of laminar flow that occurs at high Reynolds numbers (Re). Such instabilities origin form interactions between nonlinear inertial terms and viscous terms in N-S equation. These interactions are rotational, fully time-dependent and fully three-dimensional. Rotational and three dimensional interactions are mutually connected via vortex stretching. Vortex stretching is not possible in two dimensional space. That is also why no satisfactory two-dimensional approximations for turbulent phenomena are available. **Furthermore turbulence is thought of as random process in time. Therefore no deterministic approach is possible.** Certain properties could be learned about turbulence using statistical methods. These introduce certain correlation functions among flow variables. However it is impossible to determine these correlations in advance. Another important feature of a turbulent flow is that vortex structures move along the flow. Their lifetime is usually very long. Hence certain turbulent quantities can not be specified as local. This simply means that upstream history of the flow is also important of great importance.

Turbulence models (NUMECA)

One-Equation Model (Spalart-Allmaras) The Spalart-Allmaras turbulence model ([1]) is a model with integration to wall based on a transport equation for the turbulent viscosity. The aim of this model is to improve the predictions obtained with algebraic mixing-length models to develop a local model for complex flows, and to provide a simpler alternative to two-equation turbulence models. The model uses distance to the nearest wall in its formulation, and provides smooth laminar-turbulent transition capabilities, provided that the location of the start of transition is given. It does not require as fine a grid resolution in wall-bounded flows as two-equation turbulence models, and it shows good convergence in simpler flows. This model does not provide good predictions in jet flows, but gives reasonably good predictions of bidimensional mixing layers, wake flows, and flat-plane boundary layers and shows improvements in the prediction of flows with adverse pressure gradients compared with the k-epsilon and k-omega models, although not as much as the SST model.

Wilcox Two-Equation Model The k-omega model is a widely tested two equation eddy viscosity model with integration to the Wall. The numerical wall boundary conditions require the specification of the distance from the wall to the first point off the wall. In the logarithmic region, the model gives good agreement with experimental results for mild adverse pressure gradient flows.

Menter Two-Equation Model (SST) SST k-w combines several desirable elements of existing two-equation models. The shear stress transport modeling also modifies the eddy viscosity by forcing the turbulent shear stress to be bounded by a constant times the turbulent kinetic energy inside boundary layers (a realizability constraint). This modification improves the prediction of flows with strong adverse pressure gradients and separation.

Reynolds-Stress Modeling Whereas second-order closures using Reynolds stress transport model (RSTM) are based on the solution of a modeled form of the full Reynolds stress transport equations, the explicit algebraic stress models (EASM) are derived from equilibrium hypotheses imposed on the convective and diffusive terms in the RST equations. Both models are able to provide an anisotropic description of the turbulence but the RSTM closures include additional physical terms such as the convection and the turbulent diffusion of the Reynolds stresses which makes possible a more accurate description of turbulent complex flows. However, the current generation of RSTM closures does not perform outstandingly well on flows far from equilibrium.

Explicit Reynolds-Stress Modeling A quadratic Explicit Algebraic Stress Model (EASM k-w) that takes into account the variation of production to dissipation rate ratio is also avaliable. A new implementation of ASM model where the turbulent eddy viscosity provided by the explicit solution is employed is found to be robust. It has been validated for ship flows at model and full scale of different models. If RSM model provides better prediction in the region dominated by convex curvature, however, no much improvement is observed near the concave surface.

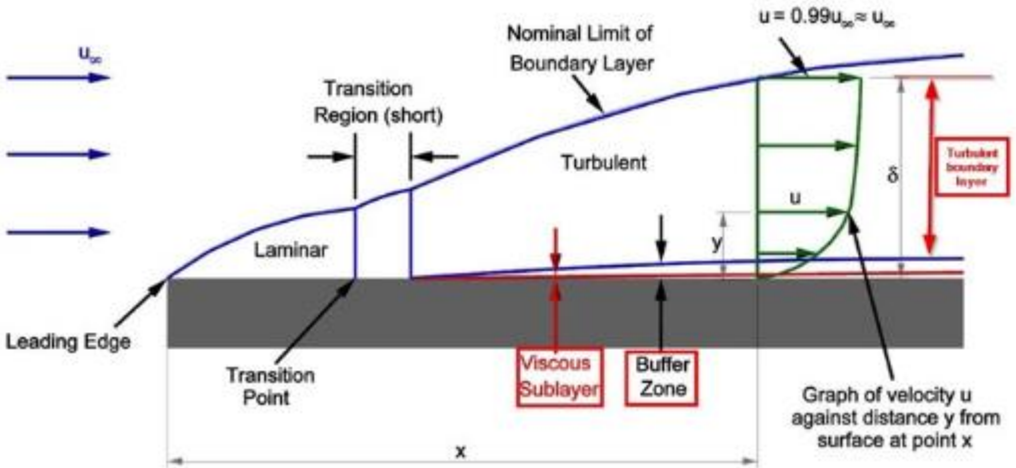
DES models The DES approach is based on a implicit splitting of the computational domain into two zones. In the first region near solid walls, the conventional RANS equations are solved. Within the second region, the governing equations are the filtered Navier-Stokes equations of the LES approach. The DES model was originally based on the Spalart-Allmaras one equation RANS turbulence model [2]. The hybrid nature of DES is not linked to any specific turbulence model [3] and the model employed in the present study is a variant based on the SST k-w turbulence model.

Roughness

Transition model The transition model is the model proposed by Menter et al. (2015) [1]. It is based on the LCTM ("Local Correlation-based Transition Model") concept. The starting point for the model is the γ-Reθ model [2]. Some of the deficiencies of this model, like the lack of Galilean invariance have been removed. Furthermore, the Reθ equation was avoided and the correlations for transition onset prediction have been significantly simplified.

Flow models

Model	Advantages	Disadvantages
Mixing-Length Model	• Easy to implement, requiring less computing resources • Well-established • Suitable for thin shear layers (e.g. jets, wakes)	• Incapable for flows with separation/recirculation
Spalart-Allmaras	• Algebraic calculation of length scale provides economic computations • Good results in flows with adverse pressure gradients • Suitable for external aerodynamics	• Definition of length scale difficult given complex geometries
Standard k-ε	• Need only supply initial and/or boundary conditions • Performs well for many industrially-relevant flows • Most validated model	• Extremely poor performance in flows with adverse pressure gradients leading to separation/recirculation
Wilcox k-ω	• Integration of flow solution to the wall does not require extra damping functions for low-Re regimes • Turbulent boundary conditions prescribed at the wall (k=0, $\omega \to \infty$)	• Prediction of separation is early and excessive
SST k-ω	• Blending functions allows use of k-ω model near-wall and k-ε in fully turbulent region far from wall • Excellent agreement for flows with or without adverse pressure gradients	• Extra functions increase complexity and computational resources required



y+ is a non-dimensional distance. It is often used to describe how coarse or fine a mesh is for a particular flow pattern. It is important in turbulence modeling to determine the proper size of the cells near domain walls. The turbulence model wall laws have restrictions on the y+ value at the wall. For instance, the standard K-epsilon model requires a wall y+ value between approximately 300 and 100. A faster flow near the wall will produce higher values of y+, so the grid size near the wall must be reduced.

A wall function is needed to accurately predict the flow in the boundary layer. Boundary layer is the thin region near a wall where the velocity gradient in the direction normal to the wall is high (the velocity goes from zero at the wall to ~ the mainstream velocity at a certain distance away from the wall). The boundary layer exists for all flows, let it be laminar, transitional or turbulent. A laminar flow would have a laminar boundary layer, which is straightforward. On the other hand, a turbulent boundary layer would have a very thin viscous sub-layer next to wall, a transition layer (buffer zone) and the turbulent boundary layer (log-layer). The turbulent boundary layer on a flat plate and the velocity profile across a turbulent boundary layer is shown in the image.

Why do we need a wall function?

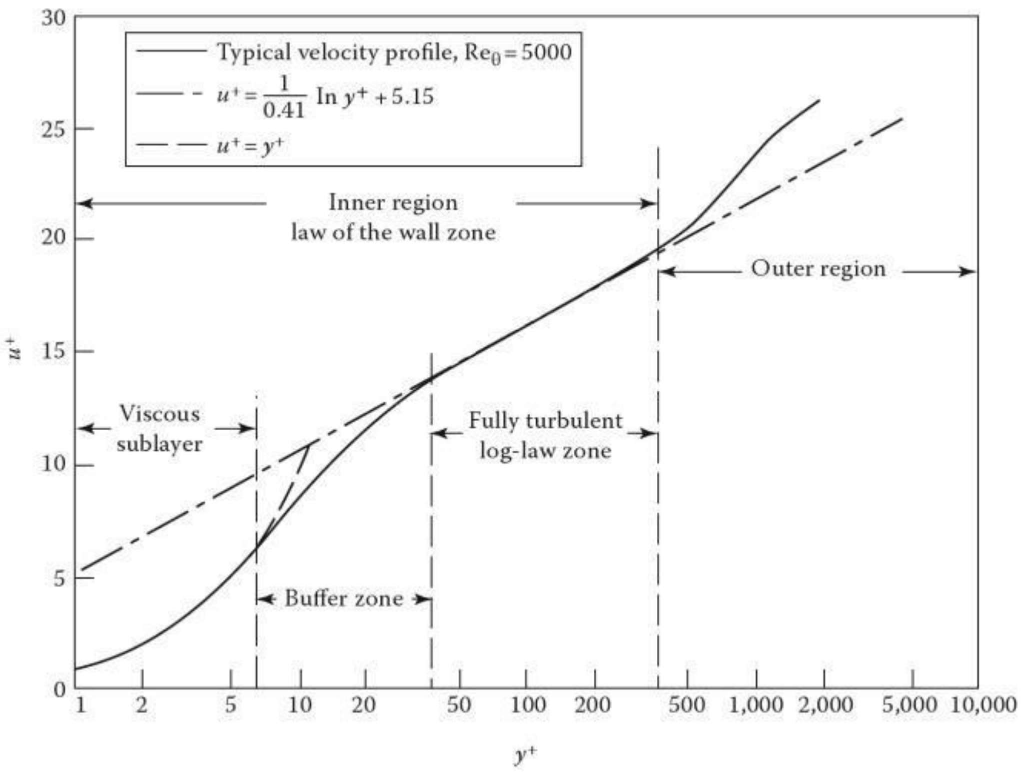
From CFD standpoint, it is essential that we create a mesh/grid which can accurately predict the velocity gradient across boundary layer. **For turbulent flows, we would ideally want the first cell from the wall to lie within the very thin viscous sub-layer. Though this might be possible for certain flow scenarios, this criteria cannot be fulfilled for complex flows in complicated geometries as it would require a very fine mesh resolution near the wall which would extensively increase the time required for solving the problem.** To deal with this requirement, a wall function is introduced which allows the use of a "relatively" larger mesh near the vicinity of the wall. Now depending on the region of the boundary layer you want to capture, you would need the appropriate mesh and appropriate wall function**. Another important parameter when it comes to wall functions is y-plus (y+). y+ is the non-dimensional distance from the wall to the first node from the wall. The different regions of the turbulent boundary layer based on y+ would be: laminar sub-layer (y+ <5); transition or buffer layer (5 < y+ <30); and turbulent or log-layer (y+ >30). The velocity is linear in the viscous sub-layer while its logarithmic for the log-layer. One important point is that the first cell adjacent to the wall should not lie in the buffer zone, i.e., y+ should not be between 5 and 30 (Not good: 5 > y+ > 30).**

Near Wall Treatment for k-e

Standard wall function:
If the first cell cannot be placed within the viscous sub-layer and it lies in the log-layer region, standard wall functions can be employed. "They provide reasonably accurate predictions for the majority of high-Reynolds-number, wall-bounded flows." The y+ value should be between 30 and 300 (valid: 30 > y+ >300).

Non-equilibrium wall function:
Under severe pressure gradient and strong non-equilibrium flows, the standard wall functions do not predict the flow accurately. So the non-equilibrium wall function provides a better prediction. "Because of the capability to partly account for the effects of pressure gradients and departure from equilibrium, the non-equilibrium wall functions are recommended for use in complex flows involving separation, reattachment, and impingement where the mean flow and turbulence are subjected to severe pressure gradients and change rapidly." The y+ value should be between 30 and 300, just like the standard wall function (valid: 30 > y+ >300).

Enhanced wall treatment:
When the viscous sub-layer needs to be captured in cases like transitional flow, separation, heat transfer, frictional drag prediction etc., the first cell has to be located within the viscous sub-layer and the y+ should be less than 1 (valid: y+ < 1). For complicated geometries, the y+ can go up to 5 (valid: y+ < 5).



The first parameter to be specified is the thickness of the first layer.

A tool allows to "best estimate" the first cell size (**First layer thickness**) that should be inserted at the solid wall and the recommended number of layers (**Number of layers**) to reach an optimal inflation. When clicking on the *Compute* button, a dialog box that allows the computation of the first cell size on the wall appears. (See FIGURE 48) The required inputs are:

- **Reference length** (in m),
- **Kinematic viscosity** (in m^2/s) of the fluid (the heaviest fluid in case of multi-fluid simulation) and **Reference velocity** (in m/s) or the **Reynolds number**,
- y+.

The relation between the parietal coordinate y^+ and width of the first cell close to the wall y is driven by the Blasius equation, expressed as follows for turbulent flows :

$$Y_{wall} = 6\left(\frac{V_{ref}}{\nu}\right)^{-\frac{7}{8}}\left(\frac{L_{ref}}{2}\right)^{\frac{1}{8}} Y_1^+ \ldots$$

where :

y_{wall} is the distance of the nearest grid point to the wall (in m),

V_{ref} is a reference velocity of the flow, for instance the inlet velocity (in m/s),

ν is the kinematic viscosity of the fluid (in m^2/s), i.e. the dynamic viscosity divided by the density,

L_{ref} is a reference length of the test case (in m),

$Y_1{}^+$ is a non-dimensional value.

The *Estimate* button is available only in FINE™/Marine package and allows to estimate the y^+. The y^+ is estimated through the equation shown:

$$y^+ = max(y^+_{min}, min(30 + \frac{(Re-1e^6)*270}{1e^9}, y^+_{max}))$$

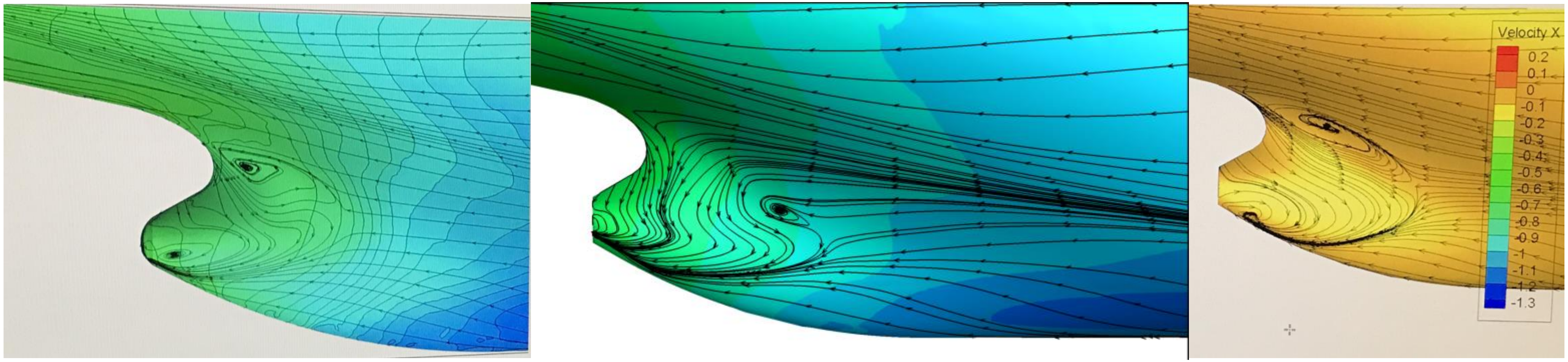
where:

y^+_{max}=300,

y^+_{min}=30,

Re is the Reynolds number.

K-omega & k-epsilon flow models



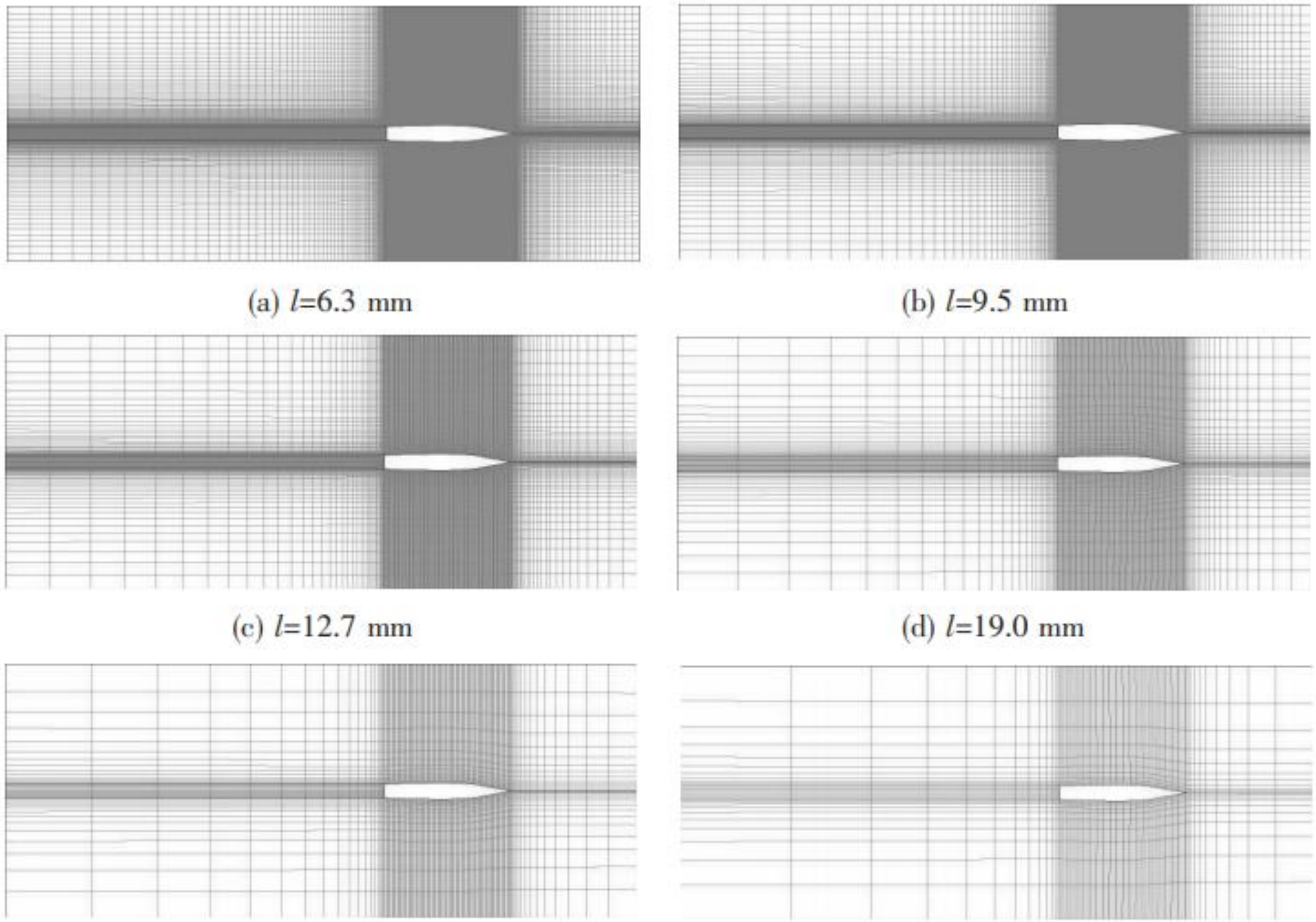
Which turbulence model?

Vol.17 No.6 2013 Journal of Ship Mechanics Jun. 2013 doi: 10.3969/j.issn.1007-7294.2013.06.004

Investigation on Some Factors Effecting Ship Resistance Calculation with CFD Code FLUENT

Though a lot of new measures[3-4] are adopted in the CFD method, some of the difficult problems[5-6] can be solved by it, and it is successfully used[7], **the simulation result is affected by the mesh generation, the choice of the turbulence models and the discretization schemes.** In the present work, combined with the comparison with model test data, some factors that affect the calculation of the resistance, such as the mesh generation, the choice of the discretization schemes and turbulence models in the simulation of a mono-hull vessel are investigated, and some suggestions are put forward.

- The model test is performed at the towing tank in Harbin Engineering University. The length, width and water depth of the towing tank are respectively 108.0, 7.0 and 3.5 m. The carriage is controlled by the computer, and the speed of it is between 0.100-6.500 m/s. In this paper, the model used is of a mono-hull, and the length, width and depth of the full scale ship are 65.0, 8.3 and 2.7 m. The scaling factor is 1/20, so the length of the water line of the test mode is 2.8 m. The model is shown in Fig.1. The surging and rolling are restrained during the experiment.
- In this study, model construction and mesh generation are performed by the pre-processing software GAMBIT, and 0.92 million non-structural grids are used to discrete the flow field. The boundary conditions are non-slip wall, velocity inlet, free stream outlet, vertical symmetrical face and free surface. The VOF method is adopted to calculate the free surface, and the k-ε turbulence model is used. The method used for the coupling of the pressure and velocity is SIMPLE, and the discretization scheme of the convection is the first order upwind difference.
- The velocities in the calculations are 0.854, 1.281, 1.601, 1.922, 2.349, 2.776, 3.203, 3.630 m/s, respectively, and the comparison between numerical result and the experimental data will be shown in Fig.9, and the numerical result agrees with the experimental data generally at high speed, and it suggests the feasibility of the simulation. There is a considerable discrepancy between the calculation result and the experimental data, which may be caused by the method of mesh generation, size and quantity of the grid, choice of discretization scheme 第 6 期 DENG Rui et al: Investigation on Some Factors Effecting … 617 and turbulence model. All those factors should be discussed, attempting to get some suggestions that could be useful in practical application.



(a) l=6.3 mm (b) l=9.5 mm

(c) l=12.7 mm (d) l=19.0 mm

(e) l=25.3 mm (f) l=40.0 mm

Fig.3 Grid distribution of the free surface

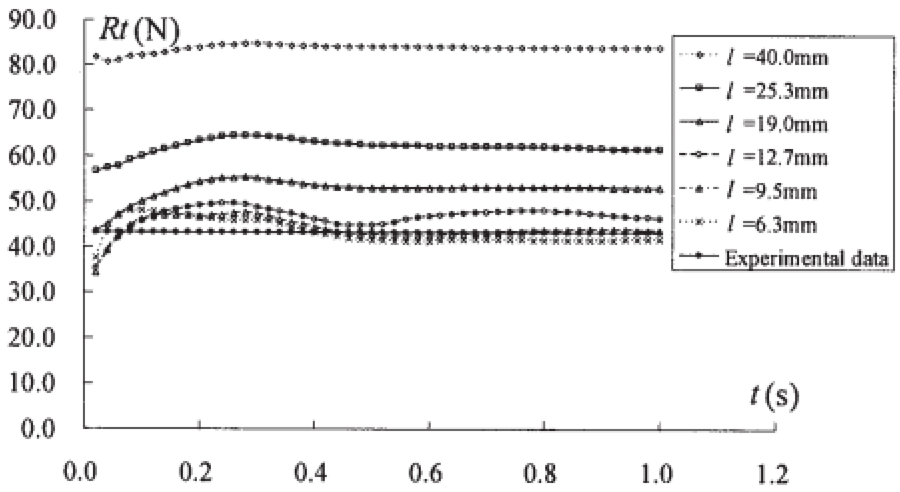


Fig.4 Comparison of the resistance with different grid sizes on hull surface

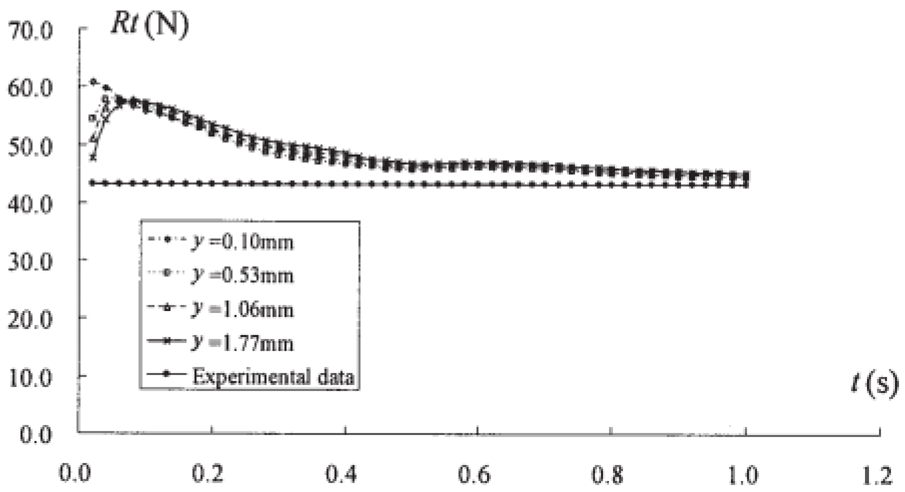


Fig.5 Comparison of the results of resistance with grids of different heights of first layer

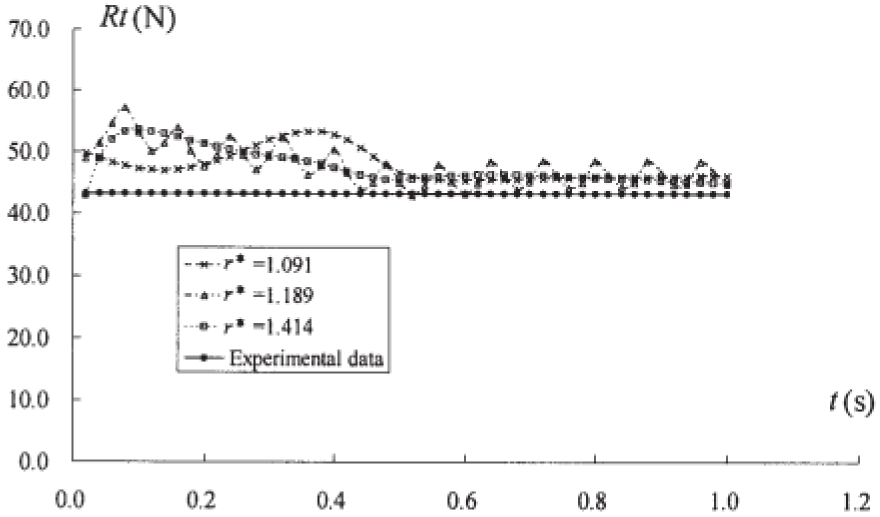


Fig.6 Comparison of the resistance of the grid in the case of different grid distribution ratio

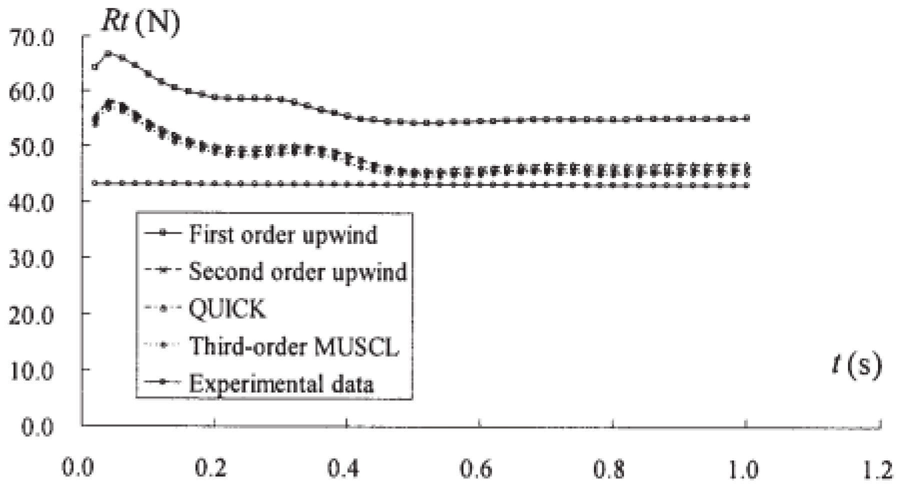


Fig.7 Comparison of the results of different discretization scheme

Conclusions

- The factor that affects the numerical result of resistance most is the grid size on the hull surface for the mono-hull vessel in this paper, and the proper grid size is 0.34% of the water line length, the accuracy of the result can thereby be improved and the cost of the calculation is not increased.

- The alteration of the height of the first grid layer does not affect the resistance obviously when the grid size on the hull surface is about 0.34% of the waterline length, and the result does not change much while the non-dimensional parameter y+ is altered between 11.5 and 200.

- The effect of the grid distribution ratio to the resistance is not significant if structural grid is used for the flow field and the techniques in (1) and (2) are adopted, but a smaller ratio r * is suggested.

- A proper accuracy of the simulation can be obtained for the mono-hull vessel in the paper, if the calculation is carried out with the second-order upwind discretization scheme, and the accuracy of the calculation is similar to those from QUICK or third -order MUSCAL scheme.

- **The k-ω SST turbulence model is suggested for the numerical calculation of the monohull vessel on the basis of the comparison of several turbulence models**

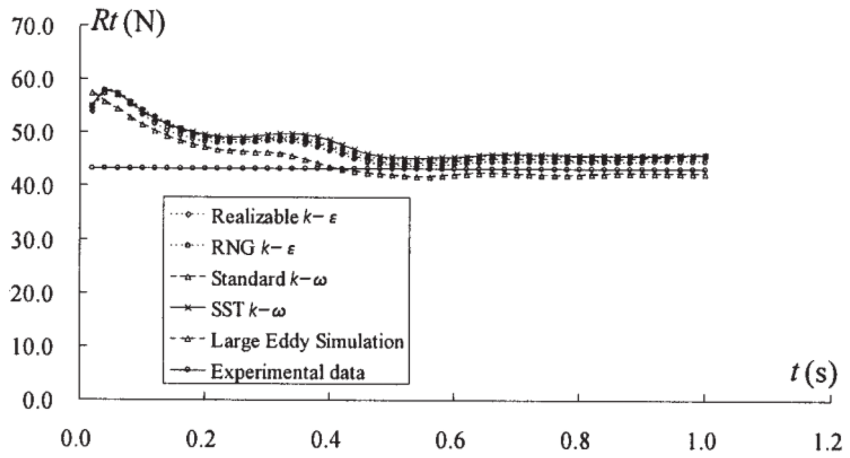
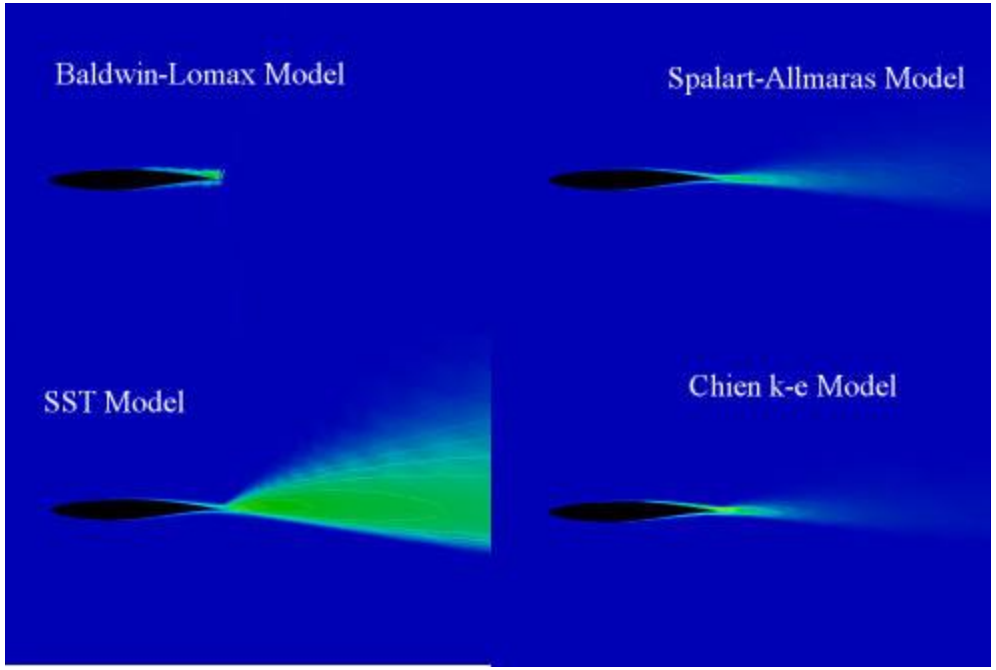


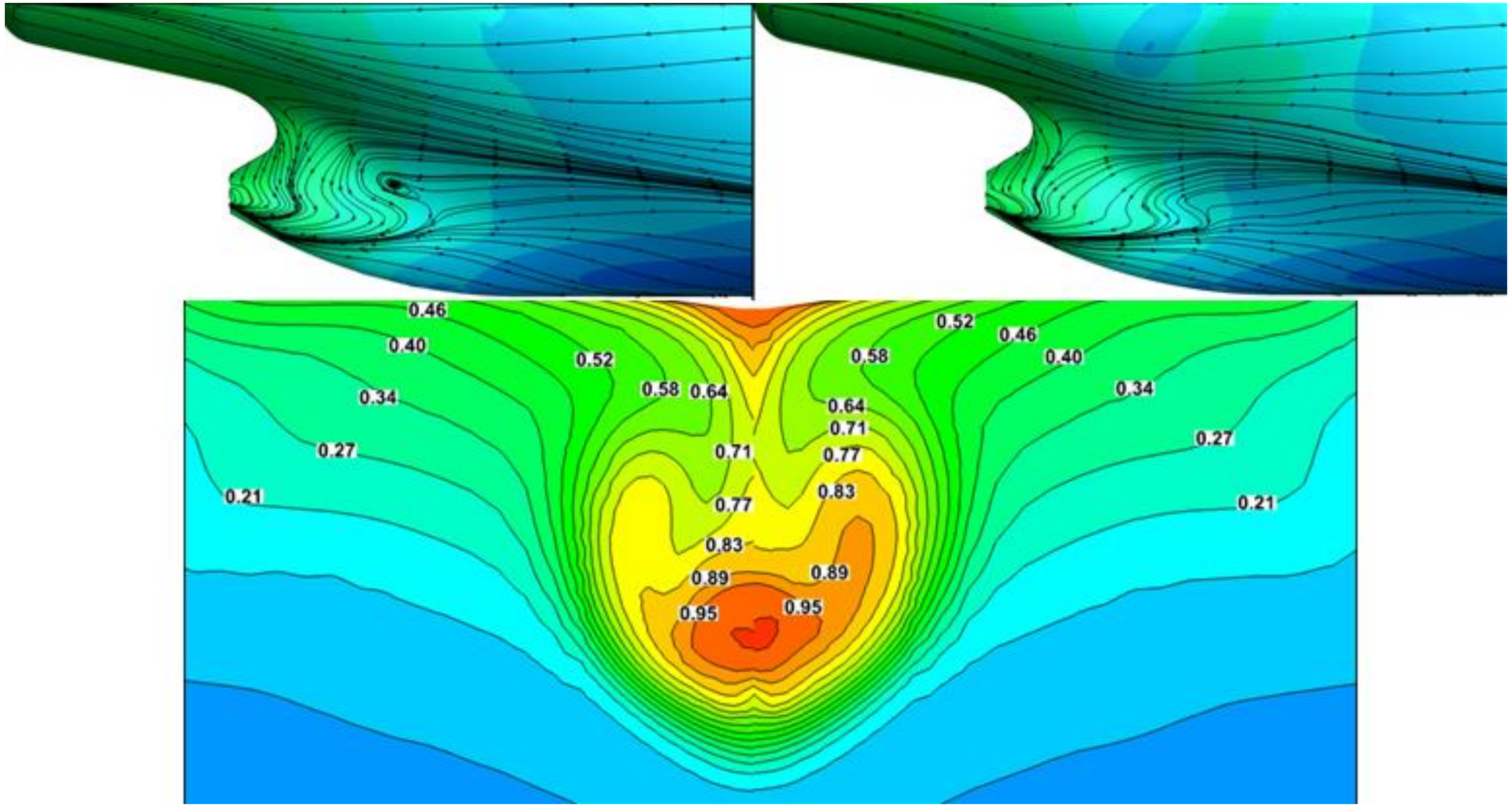
Fig.8 Comparison of the resistance results using different viscous models

Another example

The Effect of Turbulence Model on Flow Over an Airfoil



Numerical ship hydrodynamics



PhD study on: *Aft form optimization via using surrogate modelling*

Simulation settings

- Reference length (LOA) of 6.17m;
- Velocity of 1.59m/s - Froude number of 0.20;
- Draft of 0 m in the reference model frame;
- Water density 1025.321kg/m³;
- Dynamic viscosity 1.131e-3Pa.s;
- Since the flow settings are symmetric, only half of the geometry will be meshed.

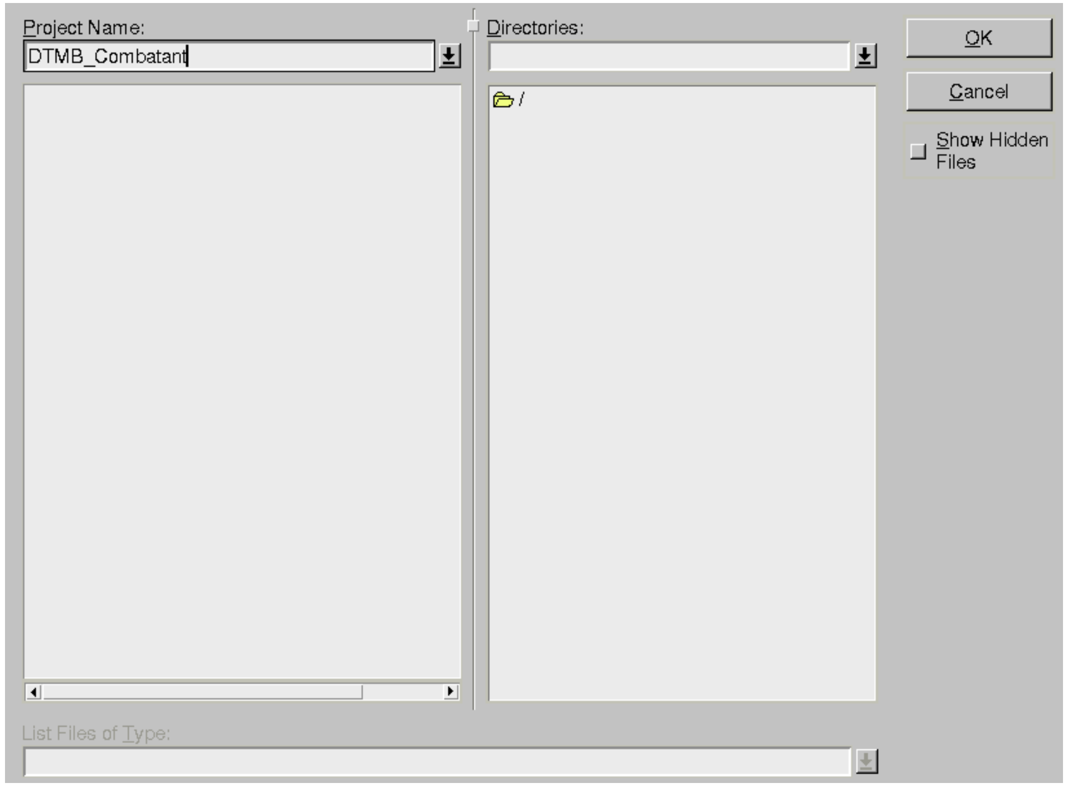
CPU Prerequisites & Estimated Time

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

- Mesh setup and mesh generation: 1hour;
- Computation setup: 15minutes;
- Computation time: 3hours;
- Post-processing: 1hour.

Domain Preparation

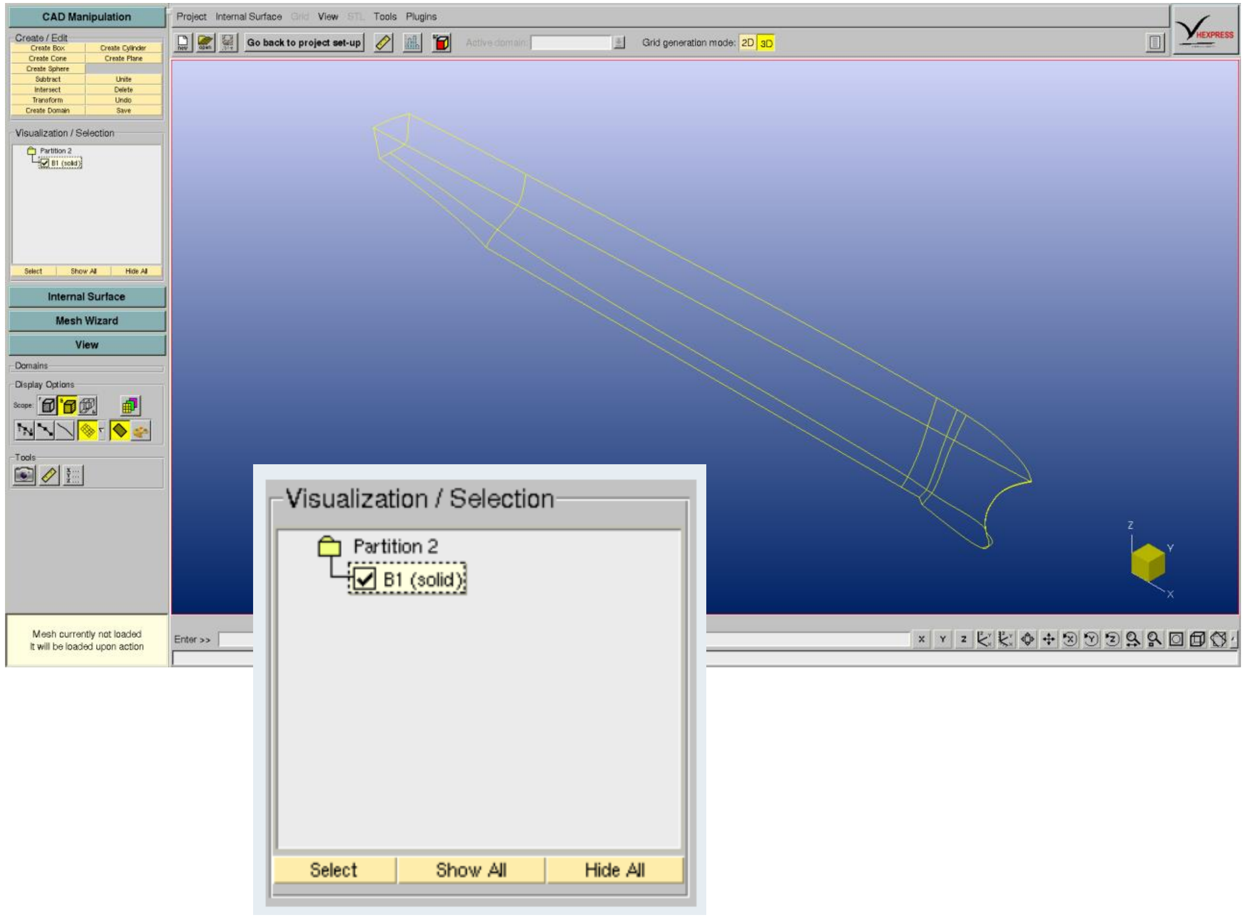
- Open FINE™/Marine, select **Create a new project/Creating a mesh** and press **Ok**.
- Select a working directory, enter the project name **DTMB_Combatant** and press **Ok**.



- Press **Yes** to start HEXPRESS™ .
- Click on **Import Parasolid model <.x_t> ....**
- Select the tutorial file **DTMB_ Combatant_ full_ scale.x_ t** from the folder _
- ***beginner/Tutorial_2/_geom/***, and confirm your choice. One Parasolid™ body is loaded.

Check the hull geometry!

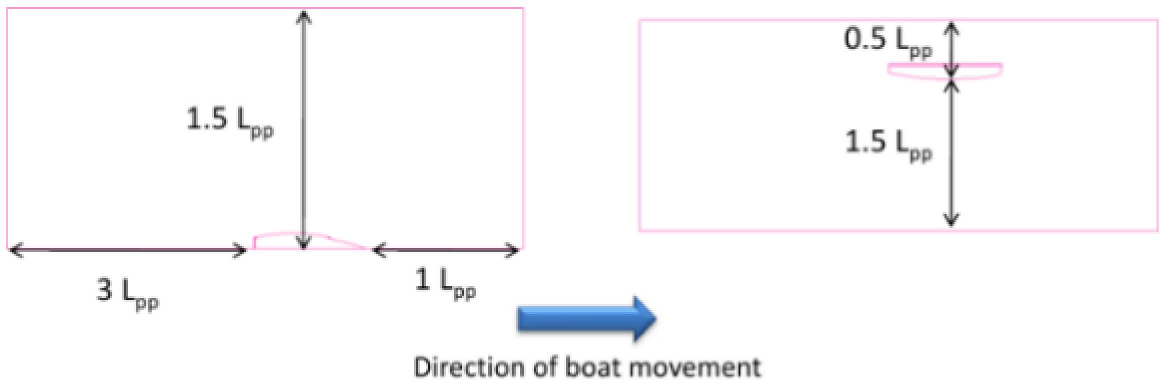
- Imported Parasolid™ model may be unclean (gaps, intersecting geometries, etc.). A checking algorithm is included in HEXPRESS™ so as to confirm the geometry validity:
- Select the **B1** body by left-clicking on it;
- Right-click on the highlighted body to access the options menu and release on **Check**. Wait a few seconds to have confirmation that the geometry is clean.
- Press **Ok** to continue

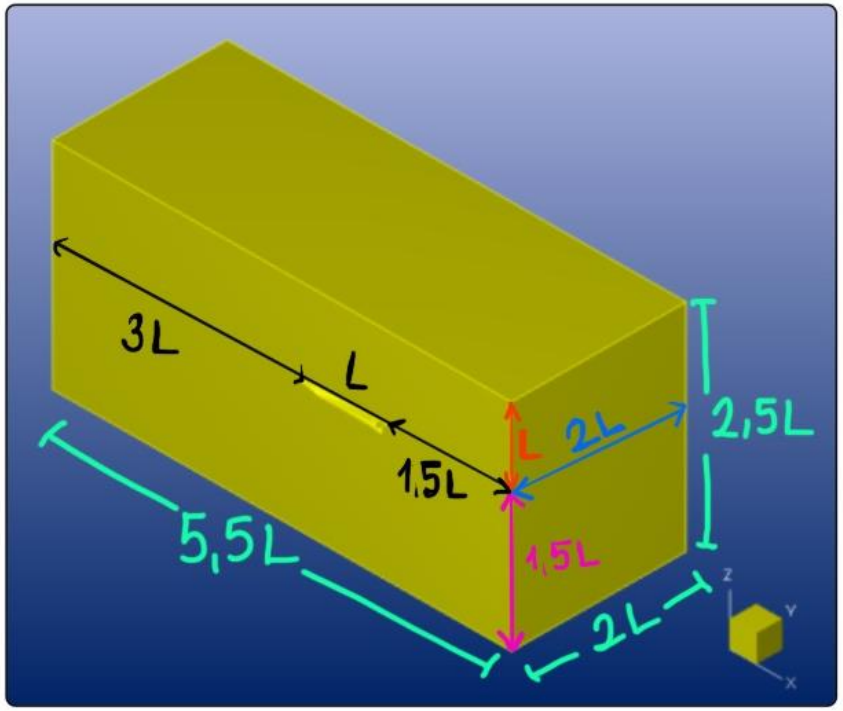


Box creation

The computational domain is usually defined as a box around the body. Standard domain sizes in terms of **LOA** are:

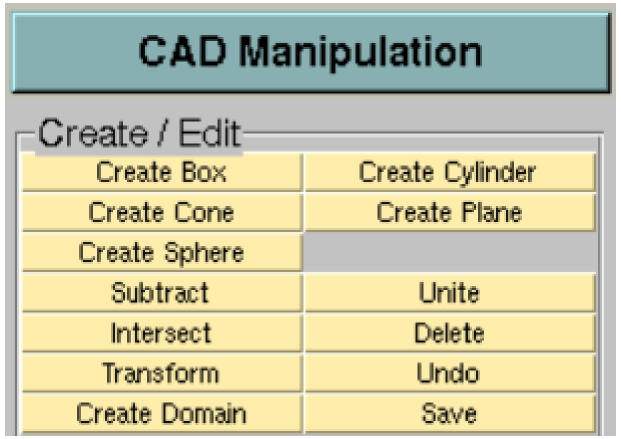
- Longitudinal (commonly X-axis): 5.5 LOA.
- Lateral (commonly Y-axis): 2 LOA.
- Normal (commonly Z-axis): 2.5 LOA.

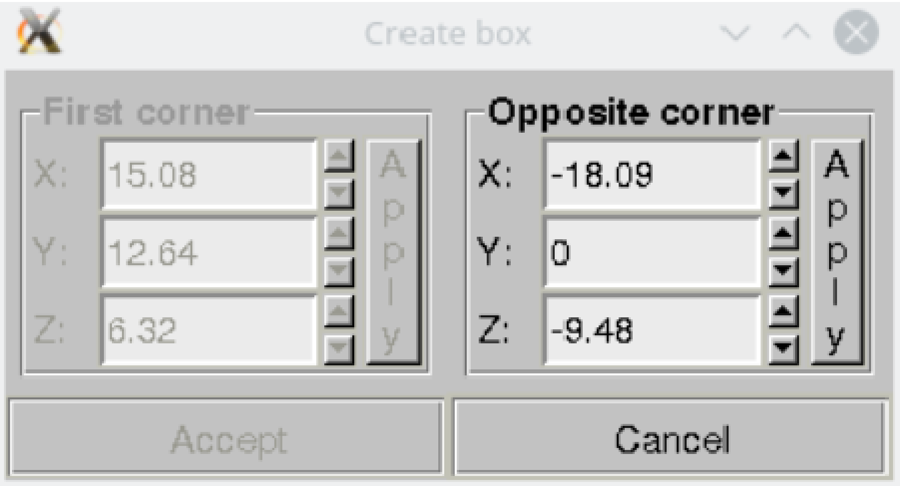




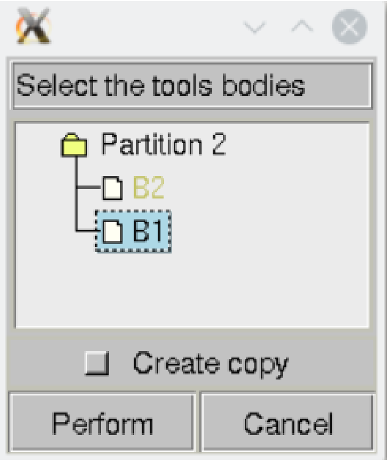
CAD Manipulation Menu

- For the 1st corner enter **(15.08, 12.64, 6.32)**.
- Click on **Apply**.
- For the 2nd corner enter **(-18.09, 0, -9.48)**.
- Click on **Apply**.
- Click on **Accept**.
- A box called **B2** is created and added to the list.
- The future meshed domain is the bounding box **B2** minus the ship's geometry **B1**. Therefore, a **Subtract** boolean operator is used during the next step.

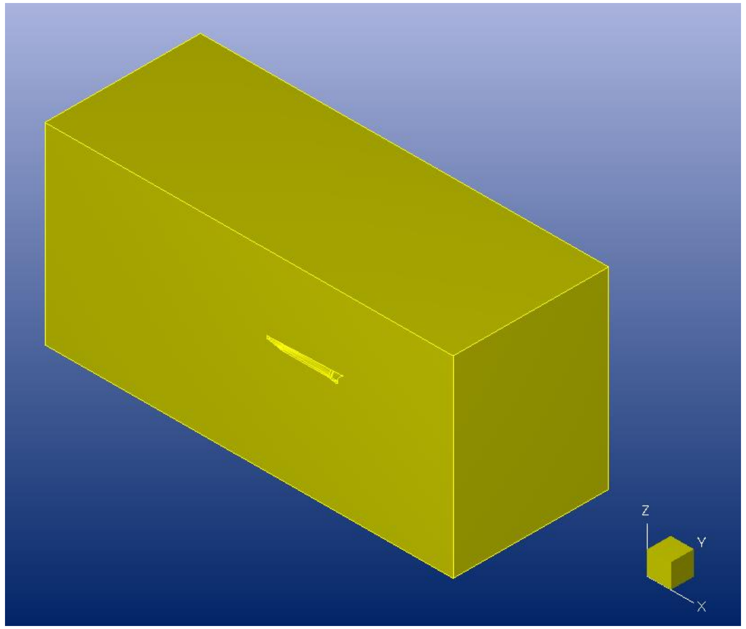




Geometry subtract



- Click on **Subtract** under the ***CAD Manipulation*** menu.
- Select the target body **B2** (the body from which you will subtract).
- Press **Accept**.
- Select the tool body: the ship **B1**.
- Press **Perform**.

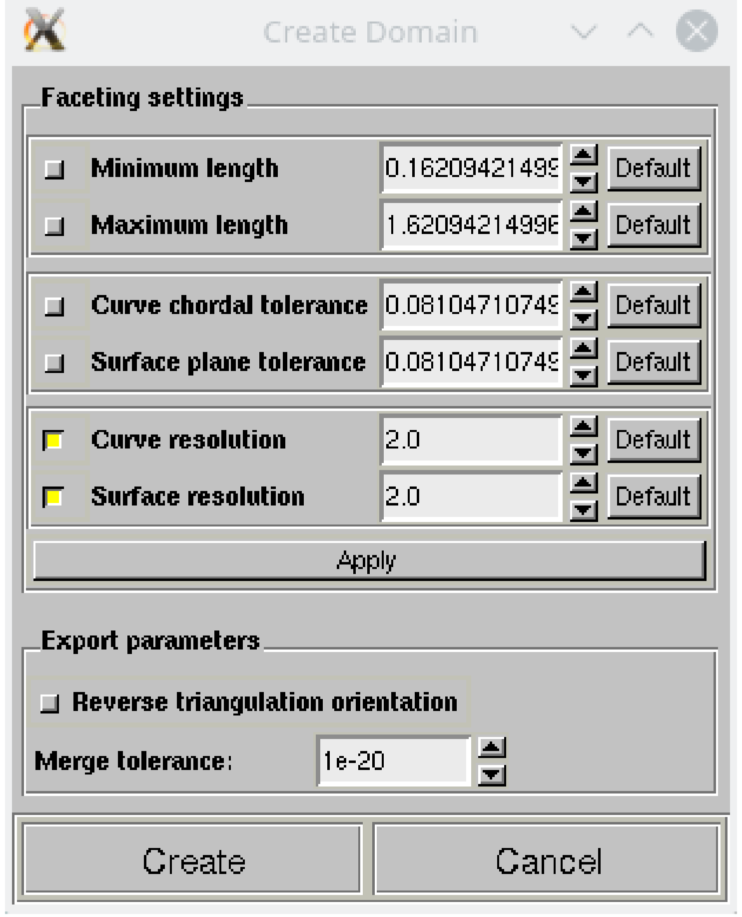


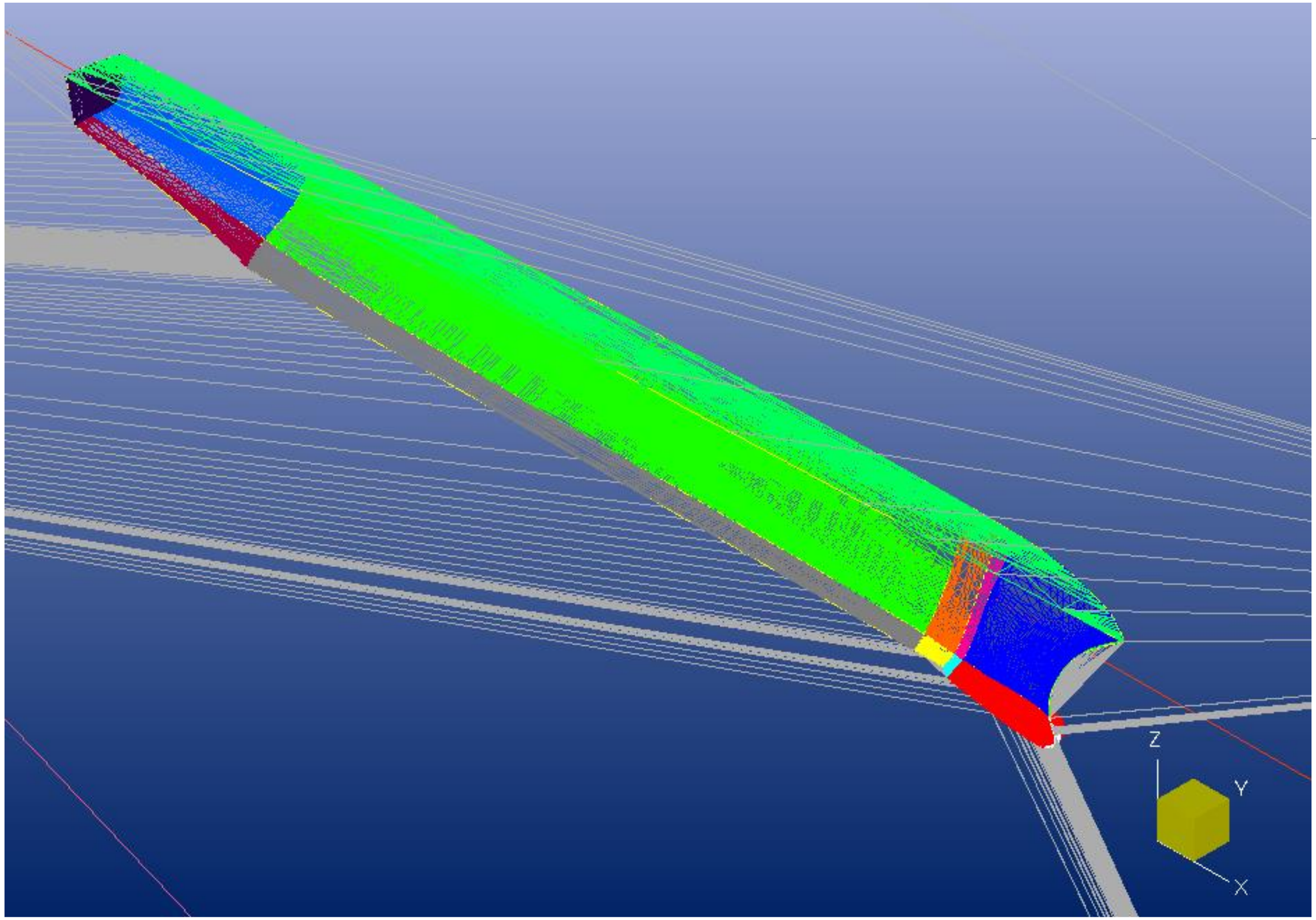
All actions performed until now could be undone using the **Undo** button.

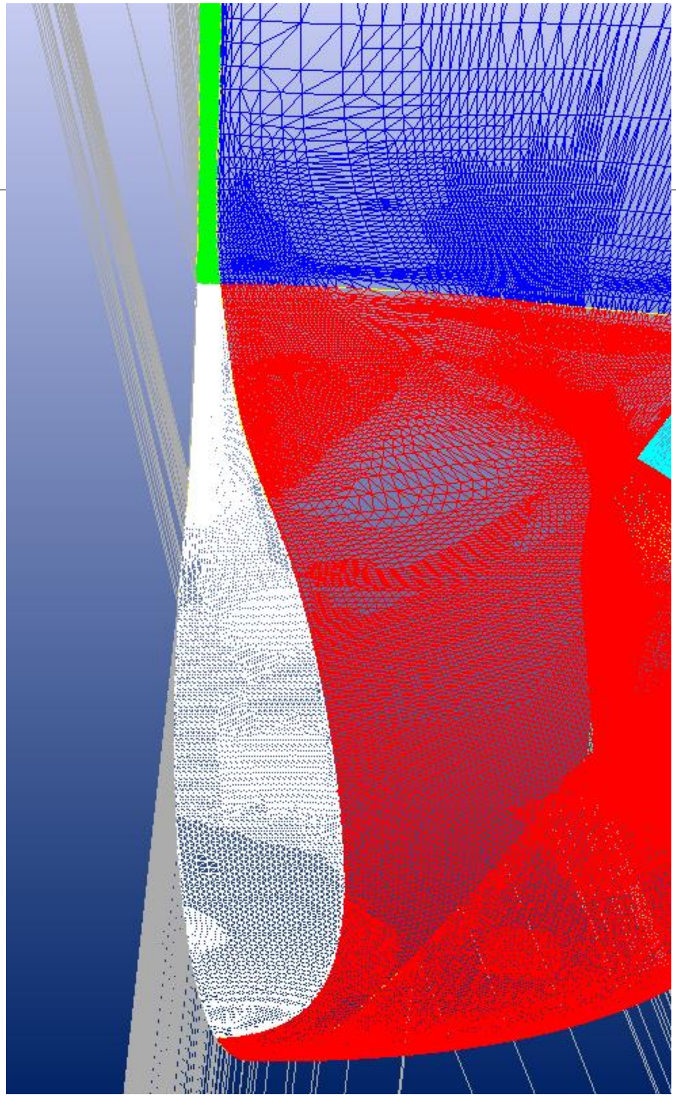
To verify that the resulting domain is indeed as expected, select the **B2** body in the list, right-click and release on **Show as solid**.

Domain triangulation and creation

- Click on **Create Domain**.
- Set the faceting settings as in the following figure:
- Click on **Apply** to see the effect of the faceting settings in the graphics window.
- These settings are important as the triangulation is the support for the future mesh. A visual check of the triangulation quality, especially in the high curvature areas, is highly recommended before saving the domain. If the geometry is not accurately respected, the faceting settings parameters should be improved.
- When satisfied, click on **Create** to save the domain file under the name **DTMB_domain** in the ***/_mesh*** subfolder of the project directory.




Z
Y
X



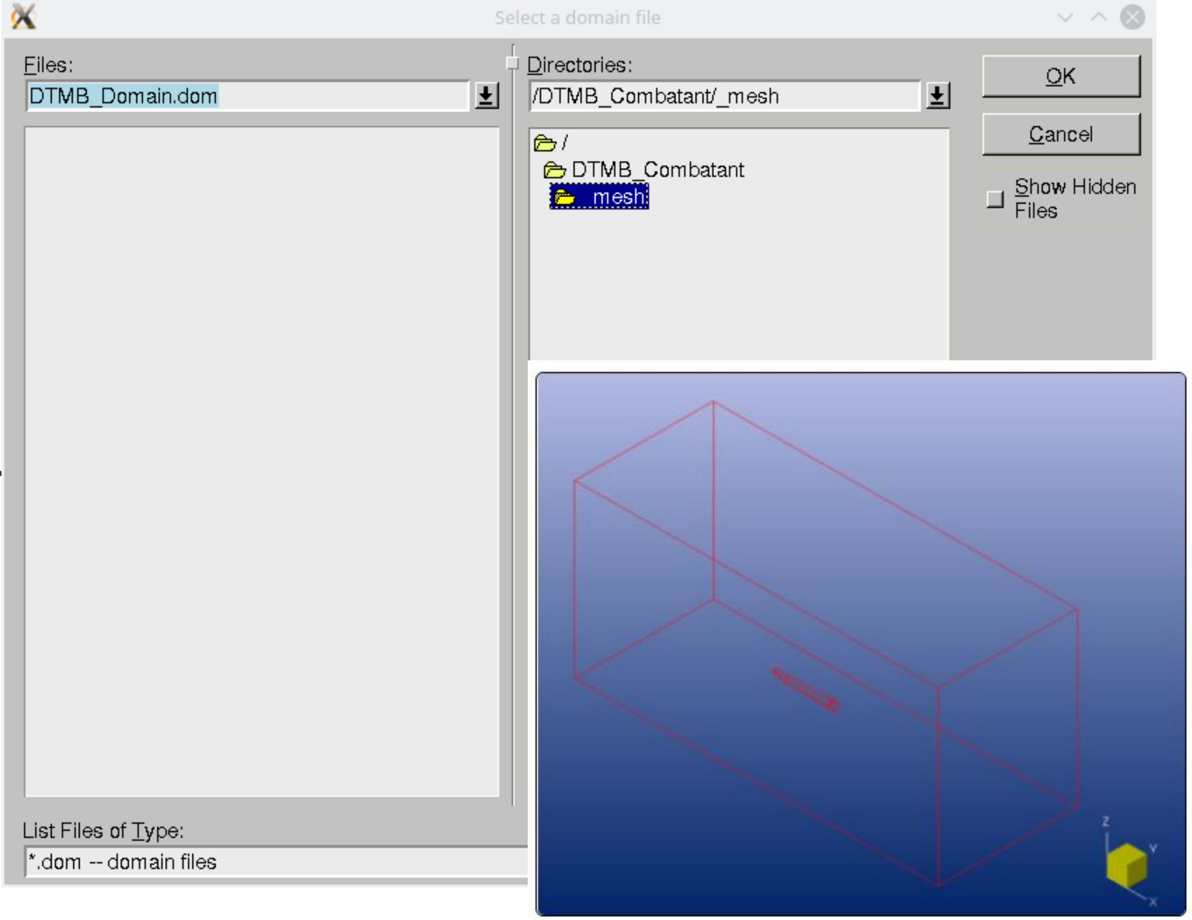
Domain backup file

Press **Yes** in order to import the successfully generated domain.

The resulting imported domain is presented in the following figure:

After the triangulation step, it is highly recommended to save a copy of the domain file under another name **DTMB_Domain_SAVE.igg**.

In case of wrong manipulations during the next step, this file will be useful for a backup of the generated domain.



Renaming the ship faces

All boat parts are already named by patch. However, during the pre-processing of the simulation, some physical values will be required. These variables will be computed using the **Domhydro** tool. As an input, this tool requires a special naming of ship parts: all body faces have to include the suffix **__b1** (for *body 1*).

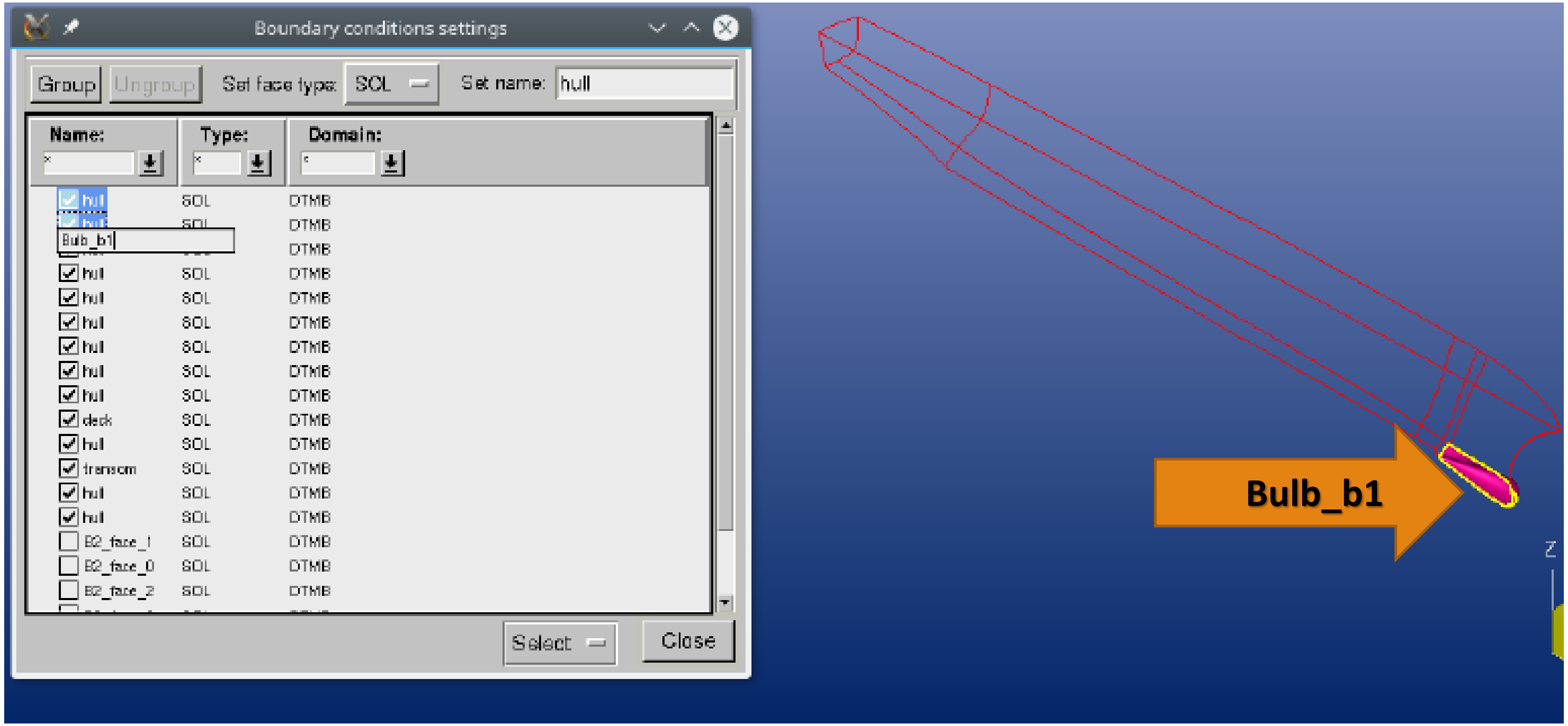
Click on the icon or select **Grid/ Boundary Conditions...** menu.

As in the following picture, select the first two surfaces of the hull.

Rename these boundaries **Bulb_b1** either by:

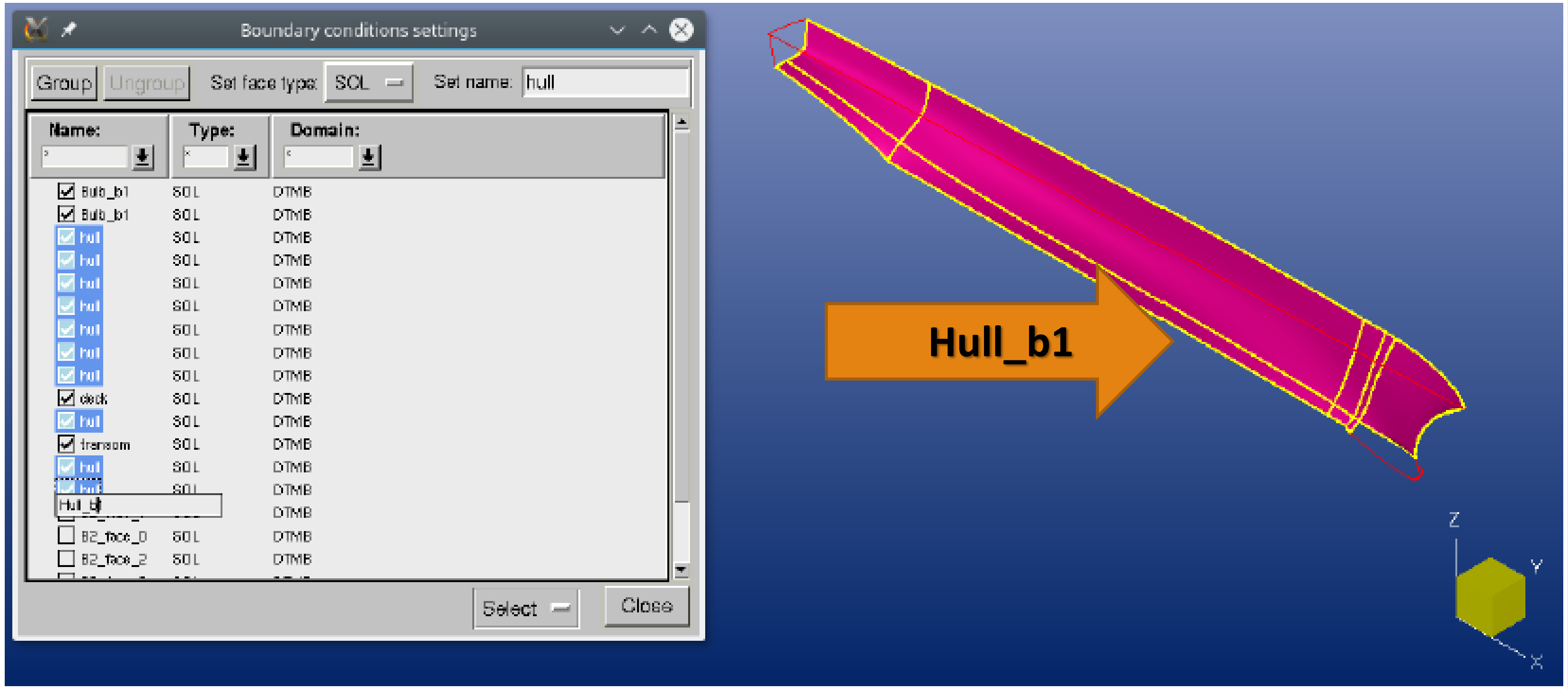
- Pressing **n** on the keyboard;
- Right clicking and selecting **Set name**;
- Using the **Set name** area at the upper right corner of the window.

Renaming the faces for domhydro tool



Renaming the hull patches

Rename the hull, deck and transom patches by selecting them and adding the **_b1** suffix, as presented in the images below:



Renaming the domain boundaries

Using the same window, the renaming of the domain boundaries is also advised in order to ease the pre-processing and the post-processing of the computation. To do so, rename each boundary using the following corresponding names:

Default	New names
B2_face_0	zmax
B2_face_1	ymin
B2_face_2	xmin
B2_face_3	ymax
B2_face_4	zmin
B2_face_5	xmax

Merging the ship faces

Meshing actions in HEXPRESS™ are mainly done at the level of edge or face entities. To reduce the engineering time spent during the mesh set-up, the number of edges and faces should be kept to a minimum (process called **domain simplification**). To reduce the initial number of faces and edges, some of them have to be **merged** together.

By default, HEXPRESS™ captures all vertices and edges in the domain. This means, during the merging procedure:

- Edges which represent a real geometrical feature or of great interest (possible cavitation, separation, etc.) should be kept;
- Edges not representing a feature can be merged together;

Merging the hull patches

Vertices with a connectivity equal or larger than 4 (connected to 4 edges) should be avoided.

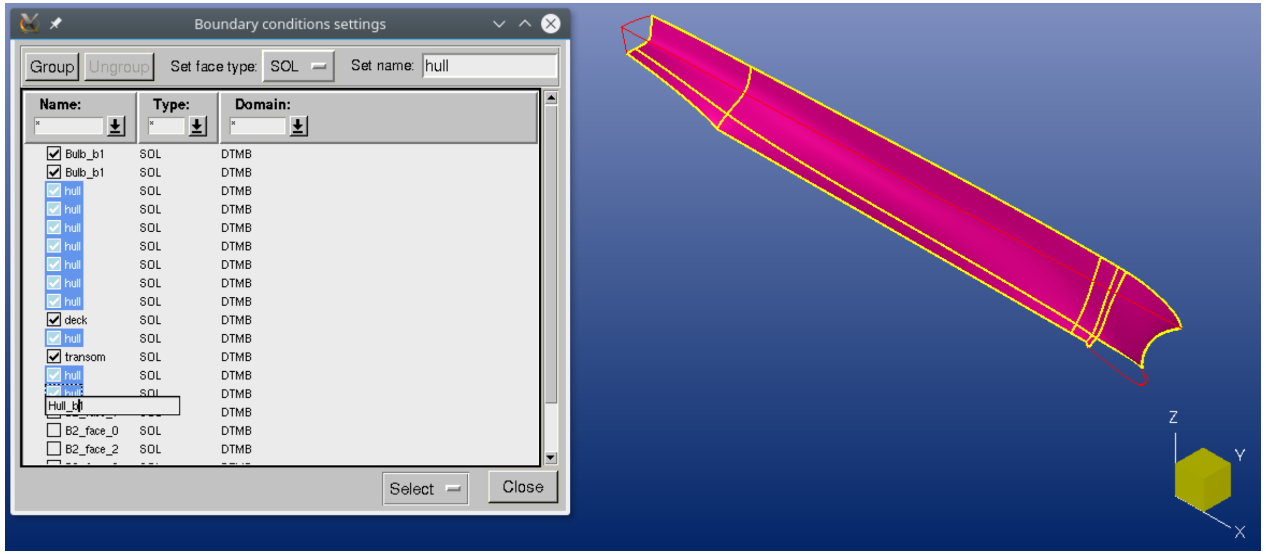
According to these guidelines, some DTMB faces can be merged together:

Click on **Domain Manipulation** to expand options.

Press the **Merge Face Selection** button .

Select the **Bulb_b1** faces and press **Merge**.

The following result should be obtained:

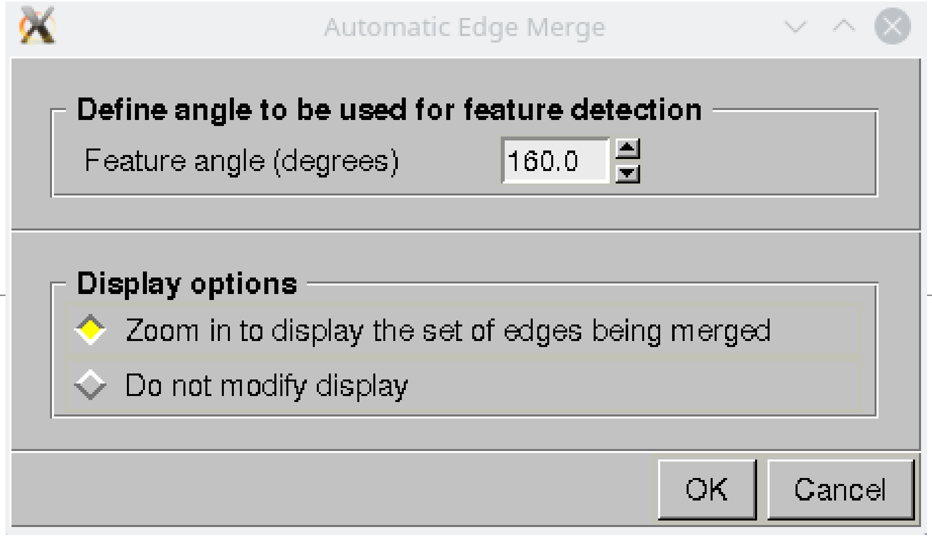


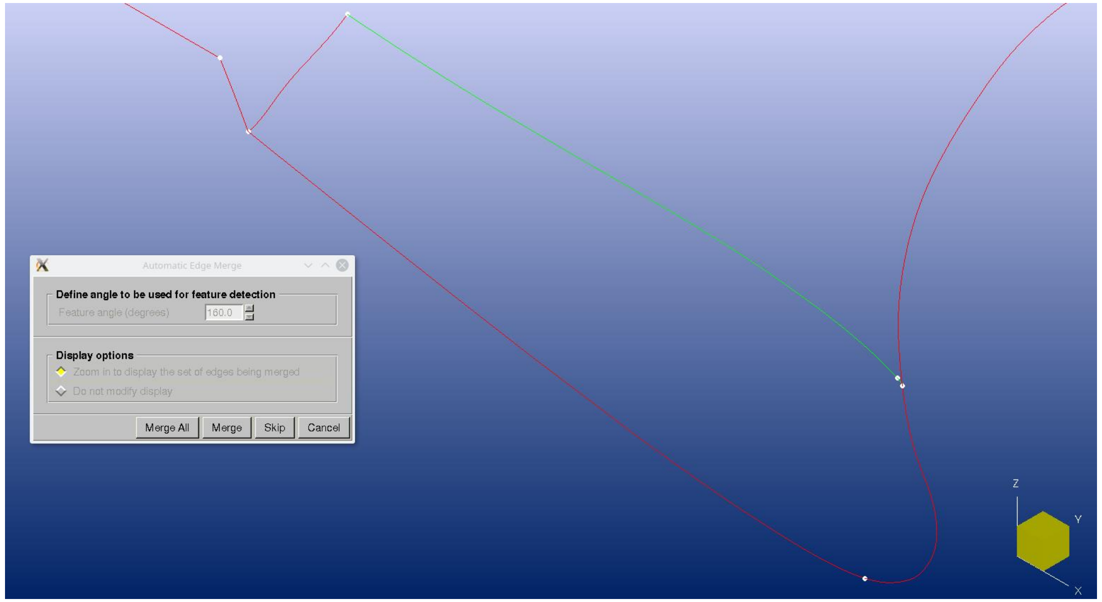
Merge edges

The **Merge Edges Automatically** feature is used so as to ease this step, it will allow to verify each edge merging before performing it:

Press the **Merge Edges Automatically** button .

Define the **Feature angle** to 160 degrees as in the picture below. Every pair of edges making an angle above 160 degrees will then be a candidate for the merging.





Boundary conditions

The boundary type of each surface is more a physical parameter than a mesh parameter.

However, HEXPRESS™ will take into account these conditions to compute a mesh in accordance with the future flow. For instance, a boundary layer mesh will be computed in our case to properly capture the flow next to the wall thanks to a turbulence model. This is why it is important to define the **boundary conditions** during the mesh setup.

By default, each physical boundary is defined as a **SOLID**. So, every boat surface is correctly described. Only the box faces have to be changed.

Since the flow is symmetric and only one half of the geometry is meshed, the **ymin** face has to be defined as a **Mirror plane**. To do so:

- Click on the icon or select **Grid/ Boundary Conditions...** menu.
- Select the **ymin** boundary.
- Left click on the **Set face type** button and release it on **MIR**.
- Follow the same steps to define **zmax**, **xmin**, **ymax**, **zmin** and **xmax** as external boundaries.

Mesh wizard

Five steps are necessary to build an unstructured hexahedral mesh of the domain:

1. Initial mesh: create a very coarse to define the cell size in the far field.

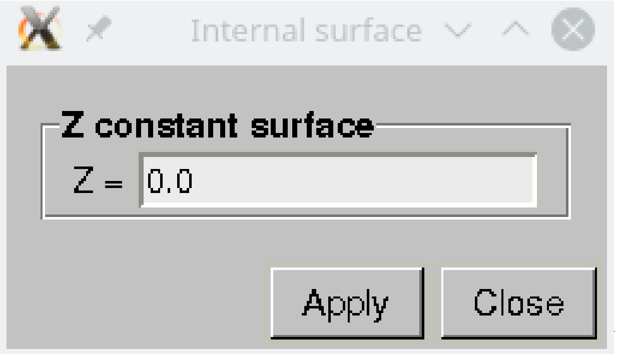
2. Adapt to geometry: refine the initial cell size a certain number of times to better capture the geometry.

3. Snap to geometry: project the refined cells onto the geometry.

4. Optimize: improve the mesh quality to remove all invalid cells.

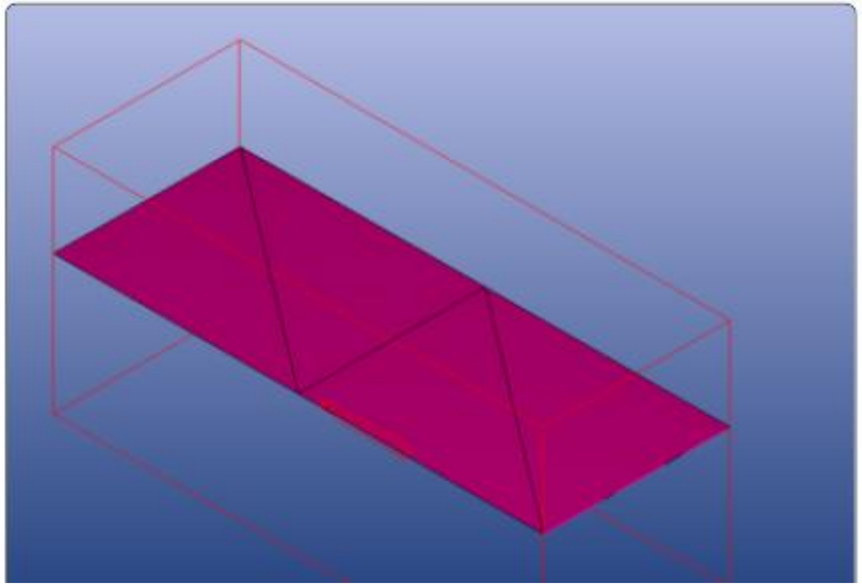
5. Viscous layer: insert viscous layers to capture the turbulence.

Internal surface creation



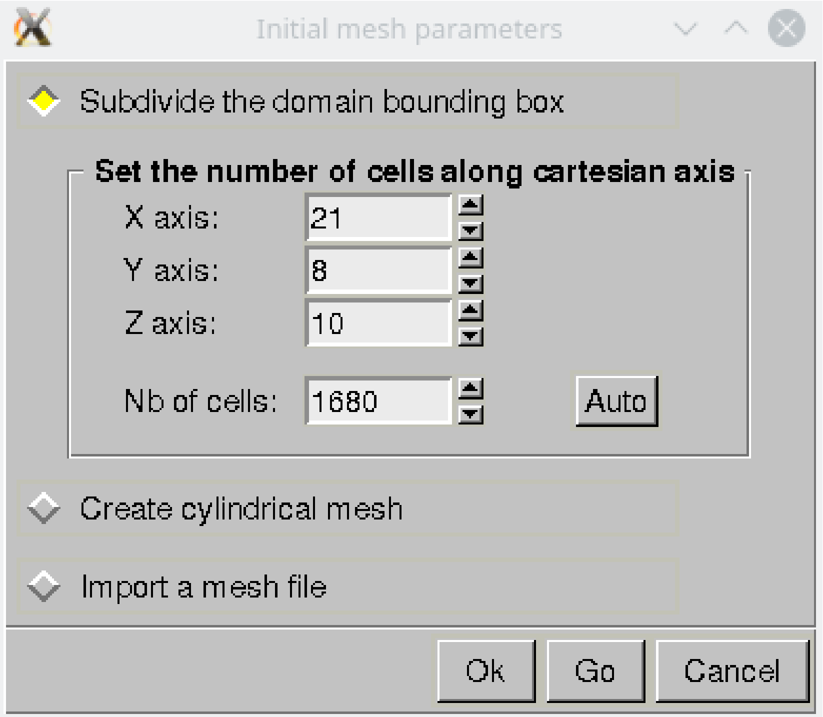
Also, since the free surface will be the interface between two fluids, it has to be meshed more accurately. For this purpose, it is convenient to define a geometrical surface at the location of the initial free surface position. To do so:

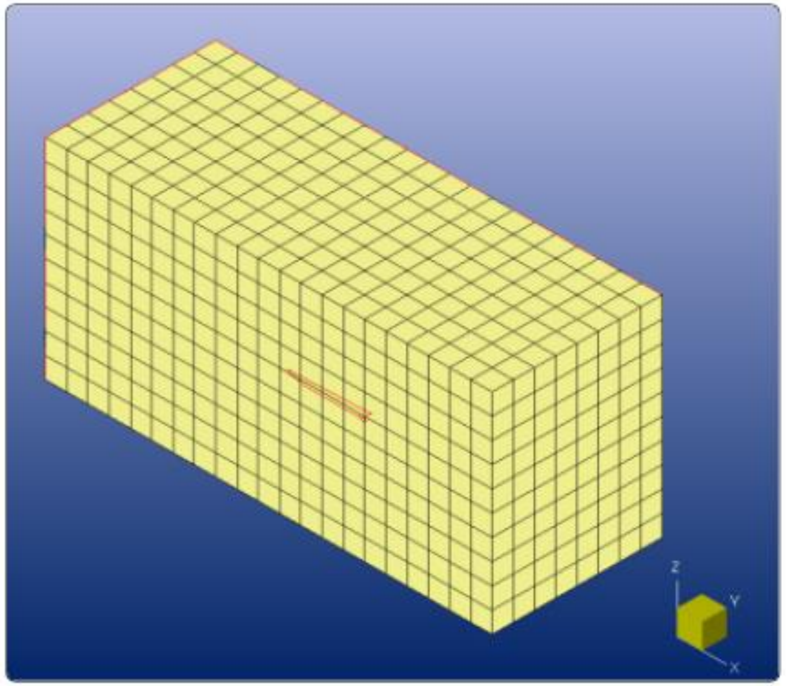
- Under the **Plugins/Marine/** menu, launch the **Internal surface creation** plugin.
- Define the initial position of the free surface:
- Press **Apply**.
- The tool will create a surface called **ISurface_Z=0.0** :



Initial mesh

After turning on the **Initial Mesh** checkbox of the **Mesh Wizard**, set the following values in the **Initial mesh parameters** window:



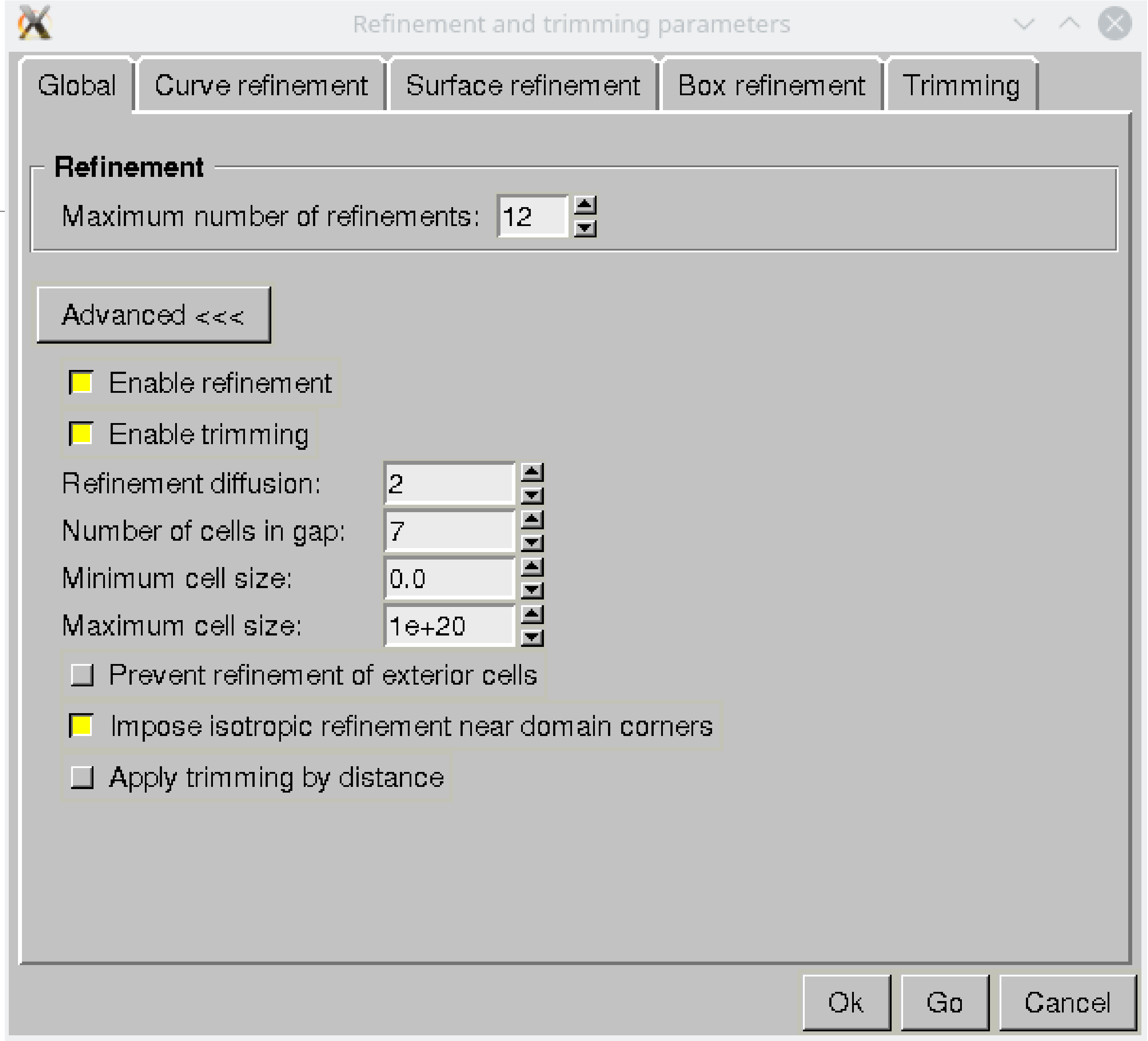


Adapt to geometry

After turning on the **Adapt to geometry** checkbox of the **Mesh Wizard**, set the following values in the **Global** tab:

It is really important to change the **Maximum number of refinements** value from 1 to 12.

As this is a **Global** parameter, a value of 1 would block every other refinement done in the other tabs.

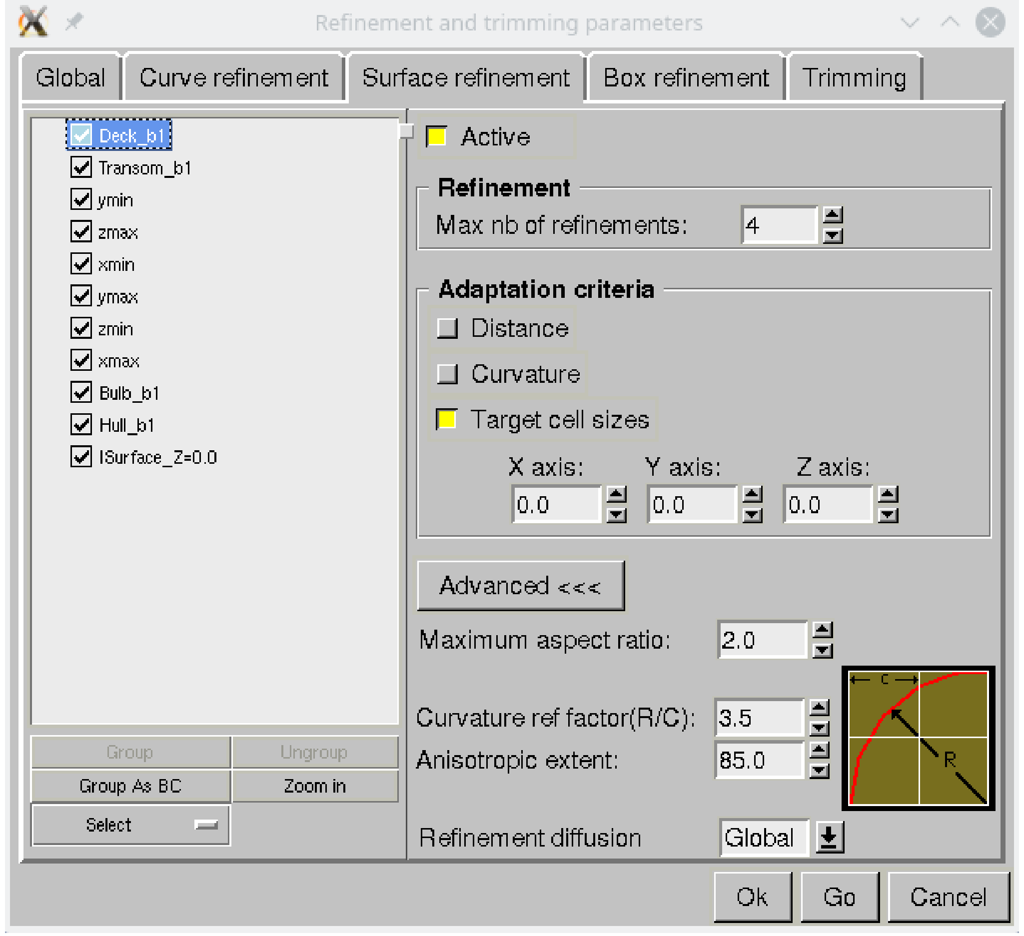


Patch surface refinement

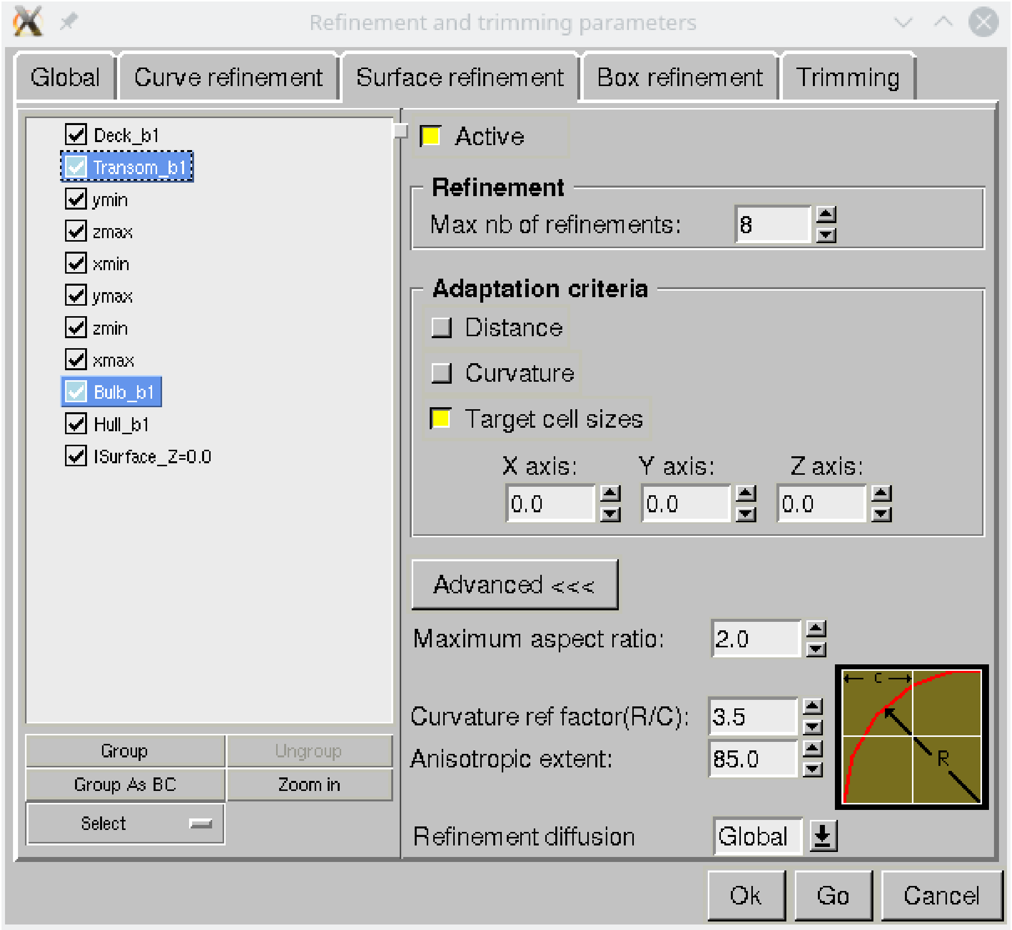
Then, the boat surfaces must be refined in order to properly capture the flow next to the wall. To do so, the **Surface refinement** tab is used.

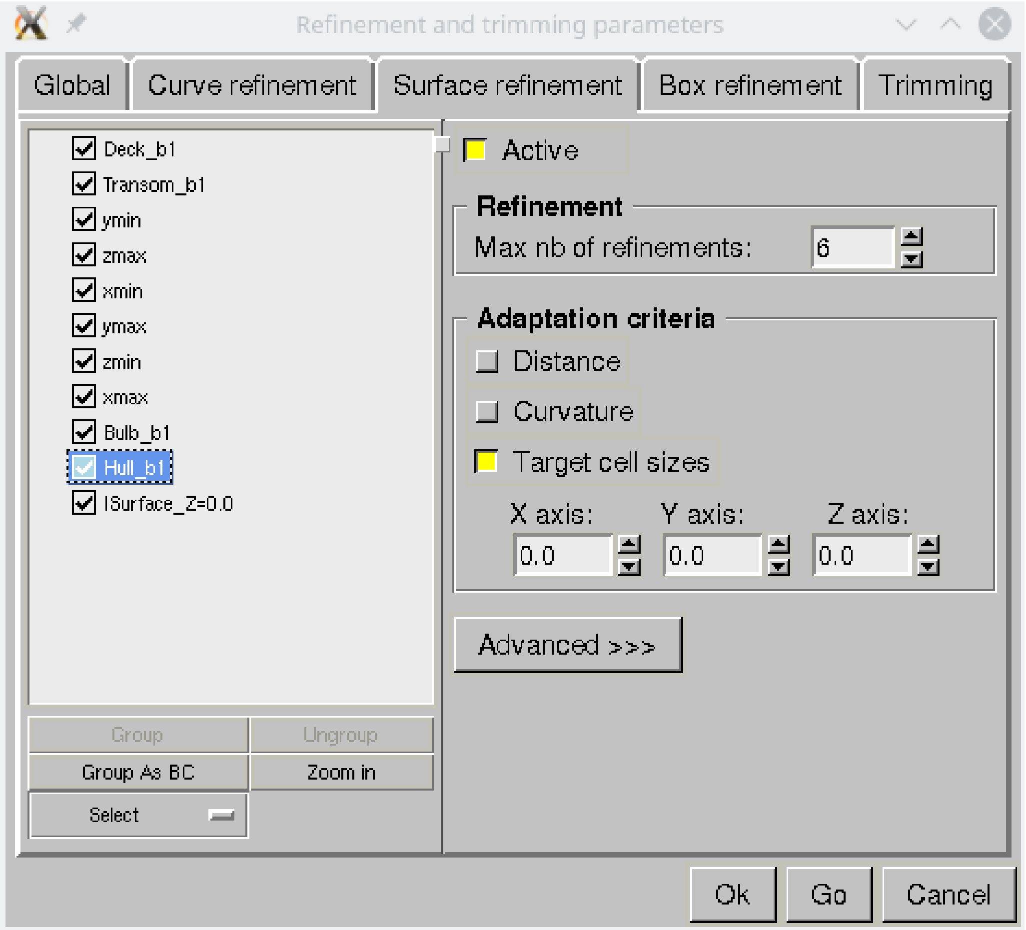
Here, the strategy adopted for surface refinements is to set the **Target cell sizes** to the minimum and to allow HEXPRESS™ to compute a maximum number of refinements.

Define the **Surface refinement** parameters for the **Deck**, the **Transom**, the **Bulb** and the **Hull** according to the following images:



Refinement values
Left: Transom & Bulb Right: Hull





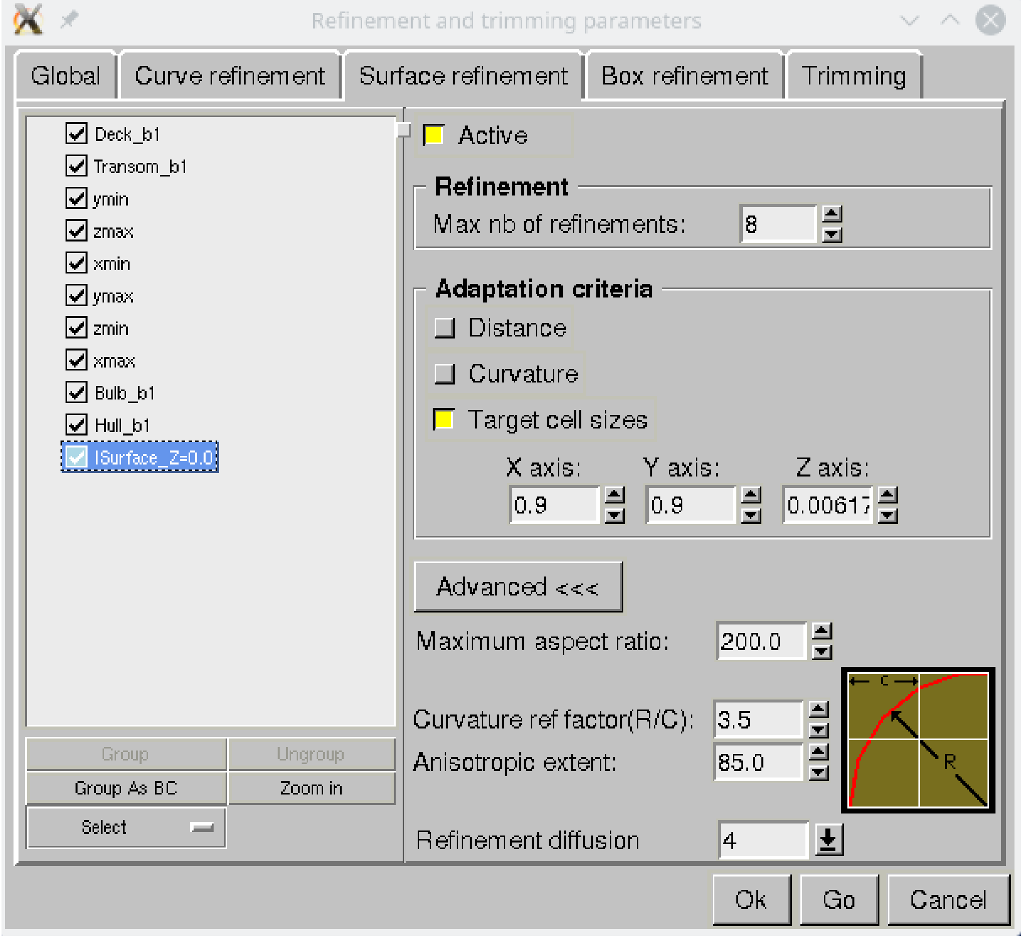
Free surface refinements

The strategy adopted for the free surface is different since **8** refinements have to be achieved along the **Z** direction.

However, the mesh must not be too refined along **X** and **Y** directions as we do not want to create to much cells.

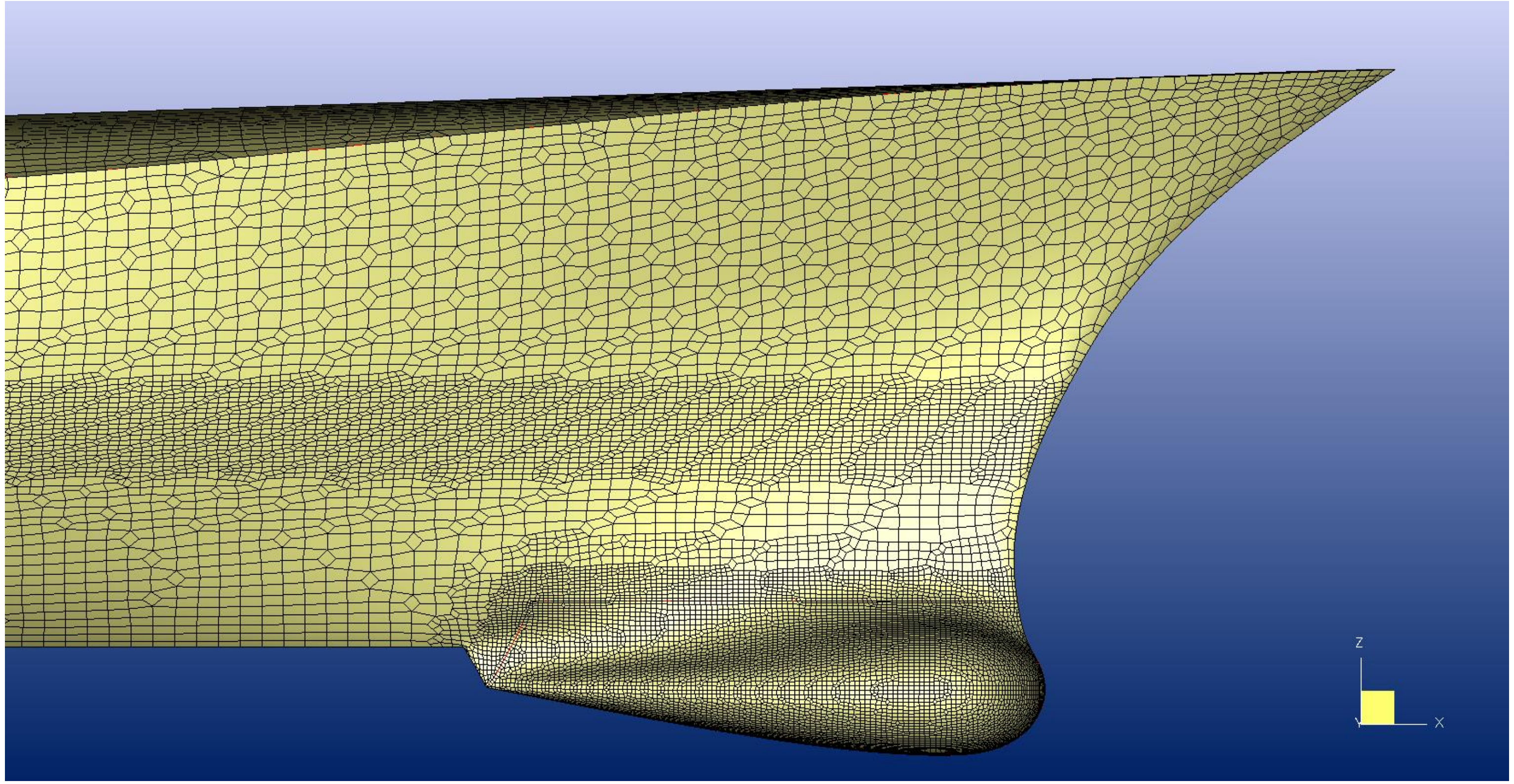
This is why the **Maximum aspect ratio** need to be defined to **200.0** in order to allow the mesh anisotropy close to the free surface.

The **Refinement diffusion** in the advanced parameters is set to 4 in order to improve the free surface capturing.



Snap to geometry & Optimize

- Turn on the **Snap to geometry** and the **Optimize** checkboxes and press **Step** for each one of them.
- Display the resulting mesh on the ship by hiding the boundaries using the **Face viewer** button and by clicking on
- Usually, when the snapping step fails, it is mainly due to the previous adaptation step.

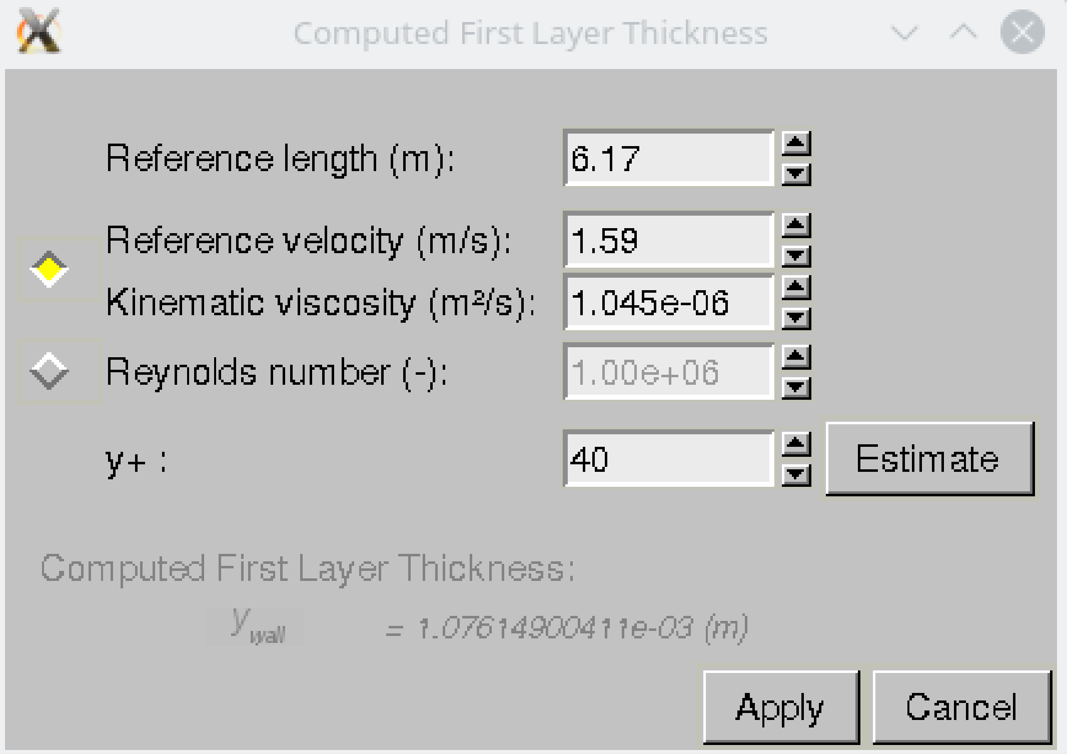


Viscous layer insertion

After turning on the **Viscous Layer** checkbox of the **Mesh Wizard**, open the **Compute** tool so as to estimate the **First Layer Thickness** for a **y+** of **40**. To do so, enter the flow and geometry properties into the following tool:

- Click on **Apply**.
- Switch to the **Surface** tab and turn on the **Active** option for the following patches:
- Transom_b1
- Bulb_b1
- Hull_b1

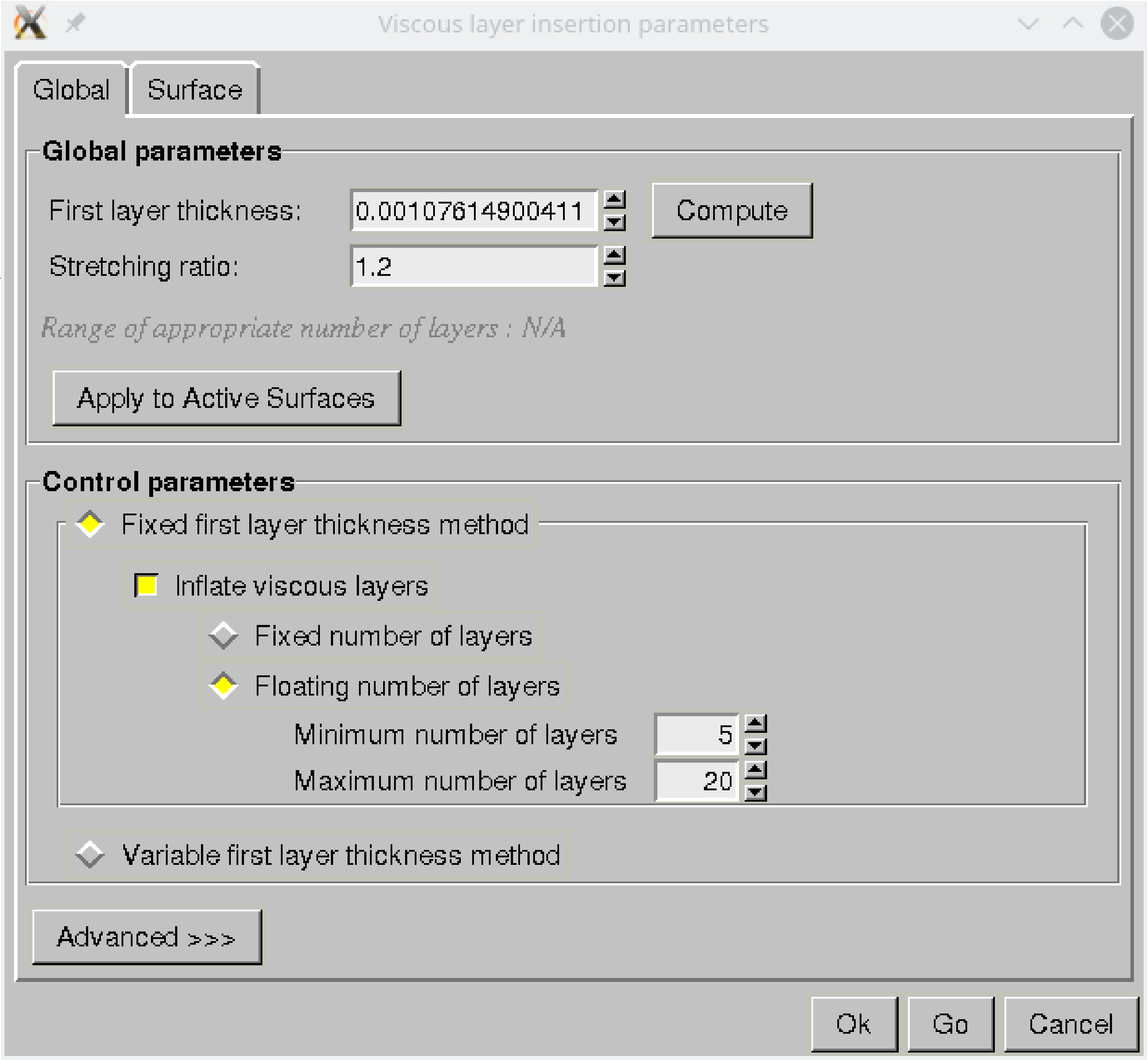
Go back to the **Global** tab and set the following **Control parameters**:



Press **Apply to Active Surfaces**.

The viscous layer parameters can now be controlled using the **Surface** tab. **Transom_b1** and

Bulb_b1 patches should have the same settings:

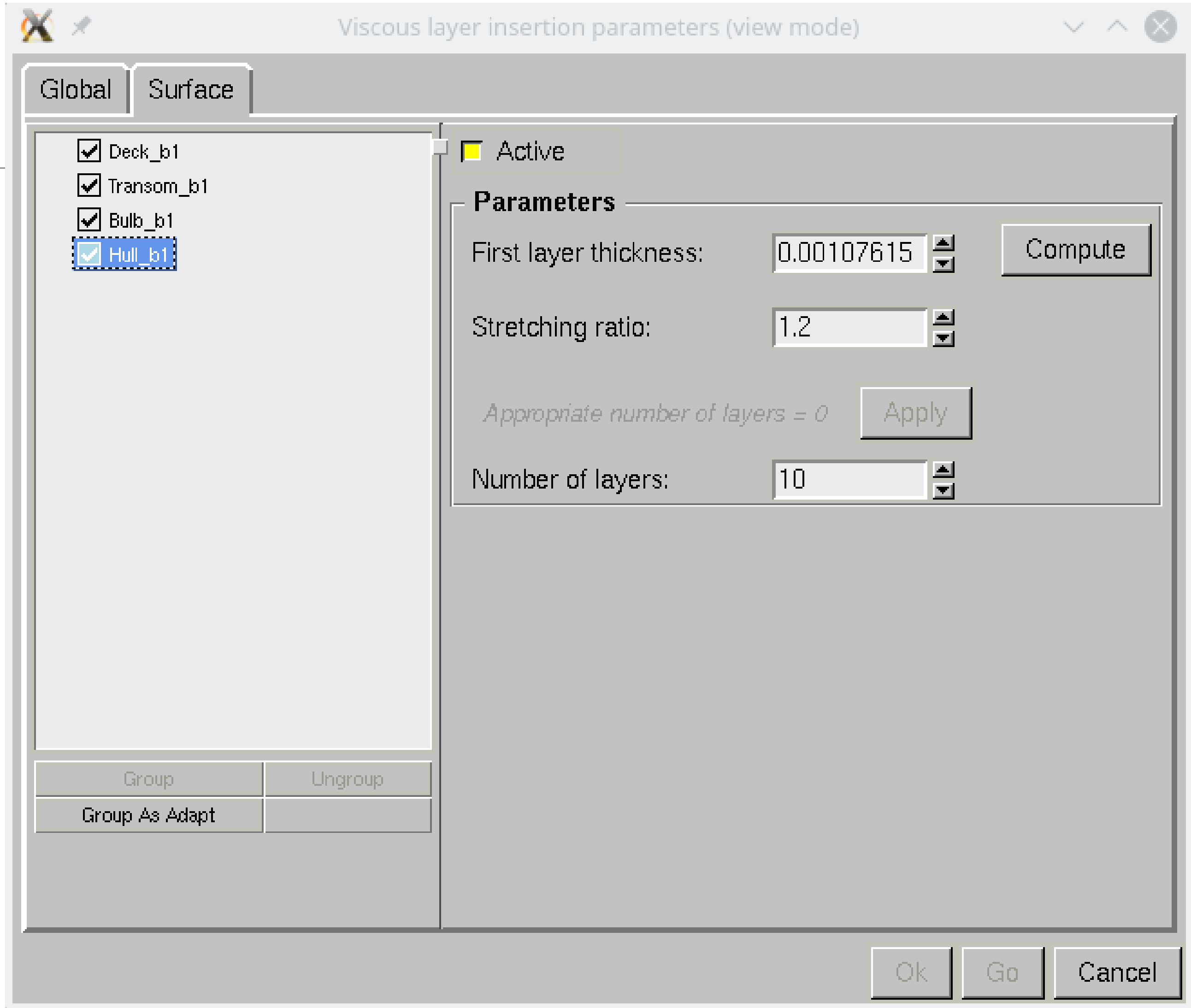


The **Number of layers** for each surface is defined by the tool according to the resulting mesh after the **Optimization** step.

As the viscous stresses due to the air flow are negligible on the **Deck**, the boundary layer of this surface will not be meshed.

After clicking on **Ok**, press **Step** in order to complete the last step of the meshing process.

Save the mesh at the end of the **Viscous layer** insertion thanks to the **Save** icon .



Mesh quality

The last step of the meshing process is to control the grid quality. To do so:

- Click on the mesh quality icon .
- Verify that there are no **Negative**, **Concave** or **Twisted** cells.
- Select the **Orthogonality** criterion. The cells can be visualized through the HEXPRESS™ GUI, by clicking on the considered column of the bar chart:

The **Orthogonality** must be always checked, the following empirical rules should be kept in mind:

- **Minimal orthogonality > 5 deg**: computation should pass without problems;
- **1 deg < Minimal orthogonality < 5 deg**: if number of cells is limited (<1%), solver should pass;
- **Minimal orthogonality < 1 deg**: try to increase the orthogonality (check the domain validity and the refinement level at first).

Mesh quality check!

Observe the **Expansion ratio** criterion as showed in the following picture. This criterio should be kept under 50.

The **Show markers** option might be activated so as to locate the cells more easily.

For a quick quality check, under the **Plugins/Marine/** menu, the **Mesh_quality_check** tool can be used. A report will be generated with NUMECA advice.

The mesh generation is now finished, go back to the FINE™/Marine interface by clicking on the **Go back to project set-up** button.

The **Mesh properties** menu appears. Check that the information is correct (**Grid units** set t **Meters**) and click on **Ok**.

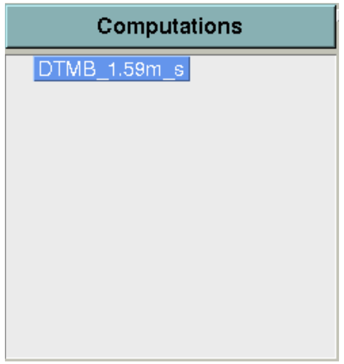
Flow settings

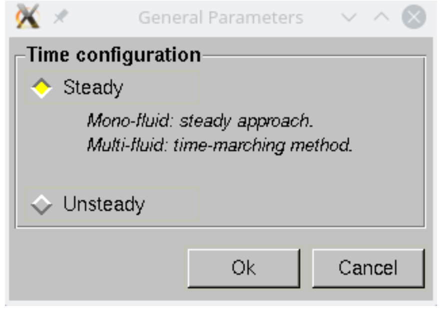
The **Physical configuration** step in which all the flow and motion settings will be specified;

The **Computation control** where the numerical parameters and the computation outputs will be defined.

Physical configuration

Double-click on General Parameters menu, keep **Steady** active and click on **Ok**:

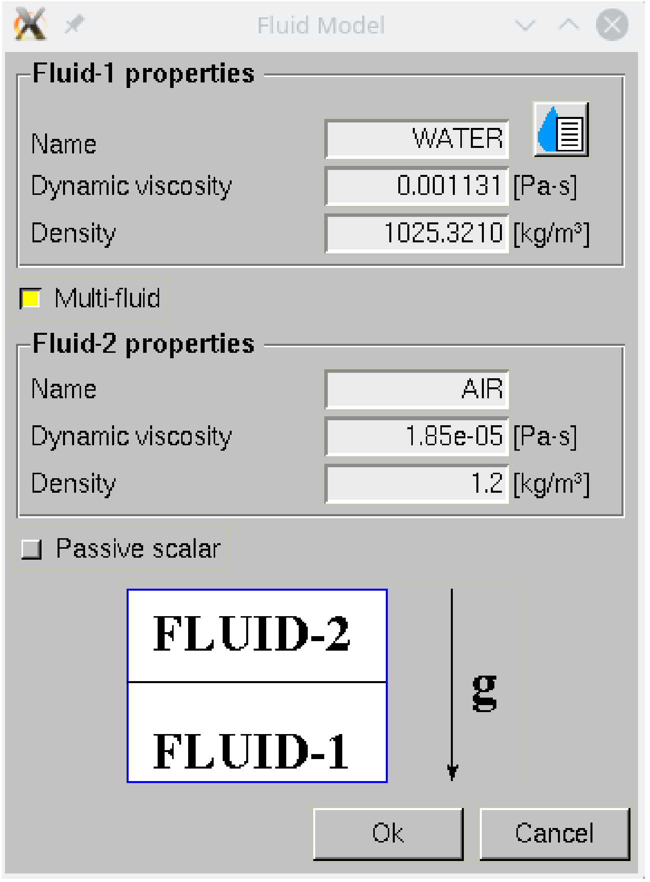


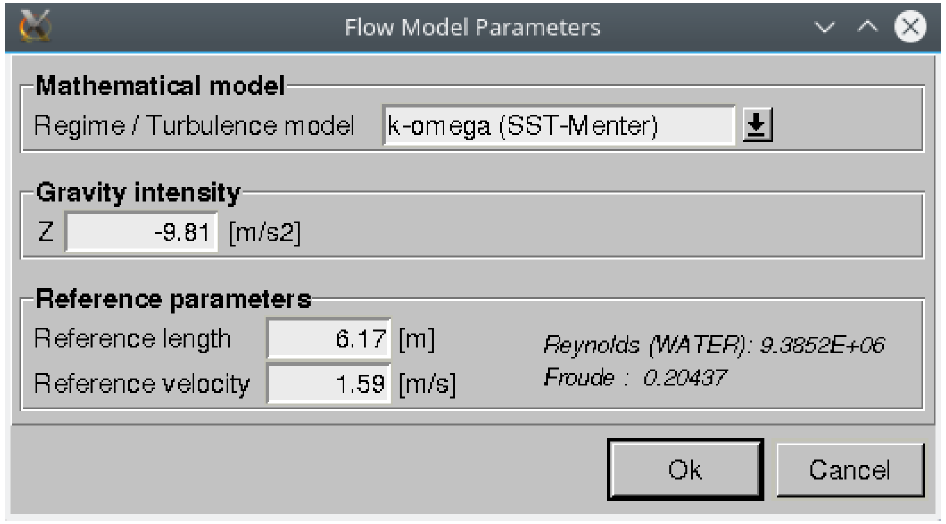


Fluid and Flow model

In the **Fluid Model** window, press the **Water properties** button . Then, in the **Salt Water** tab, select the **18.0°C** line and press **Ok**. Click on **Ok** to exit from this menu.

In the **Flow Model Parameters** menu, set the reference values for the **length** and the fluid **velocity** to respectively **6.17 m** and **1.59 m/s**:





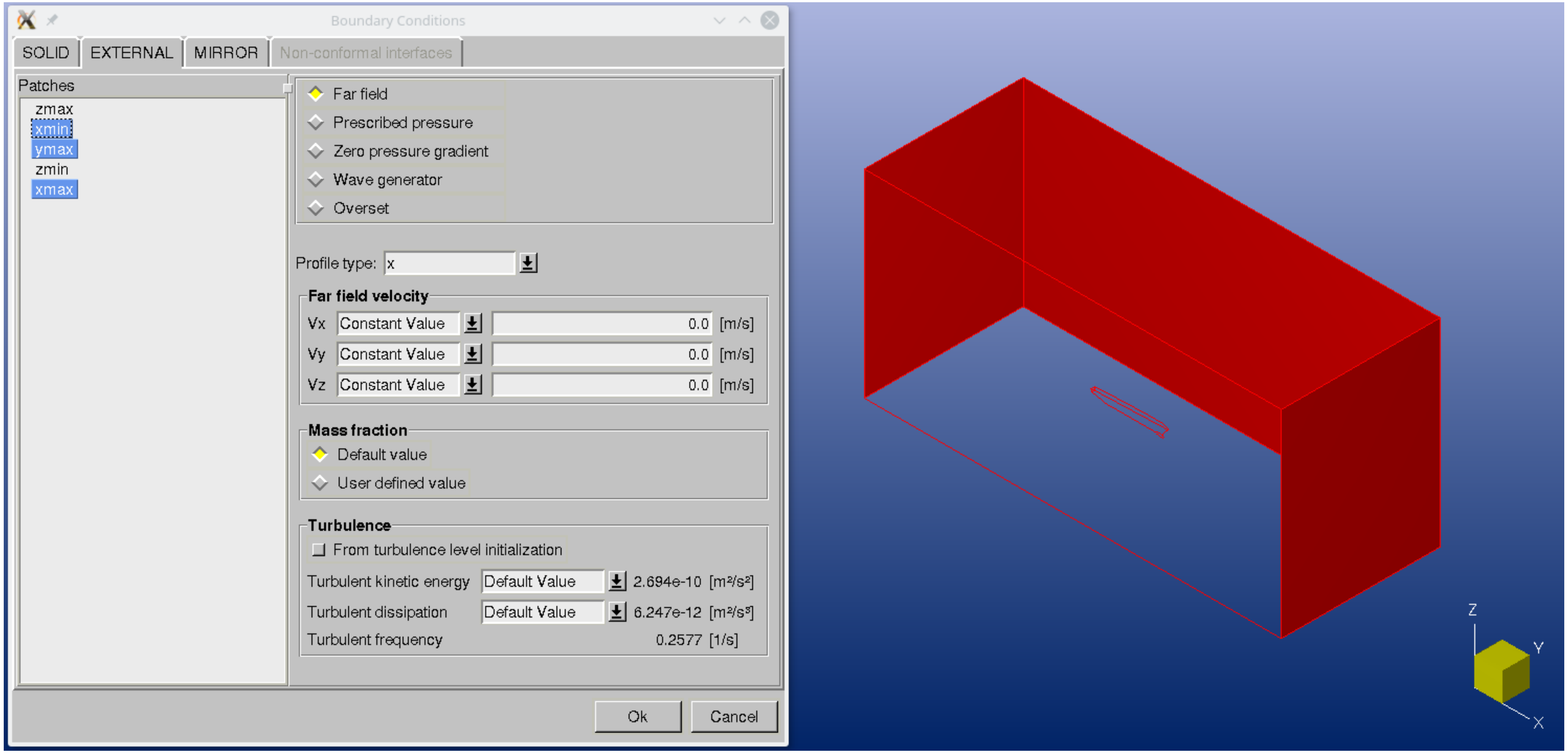
Boundary conditions

In the **Boundary Conditions** menu, three tabs corresponding to the boundary conditions defined in HEXPRESS™ are active. For each one of them, some boundary conditions are available for the patches.

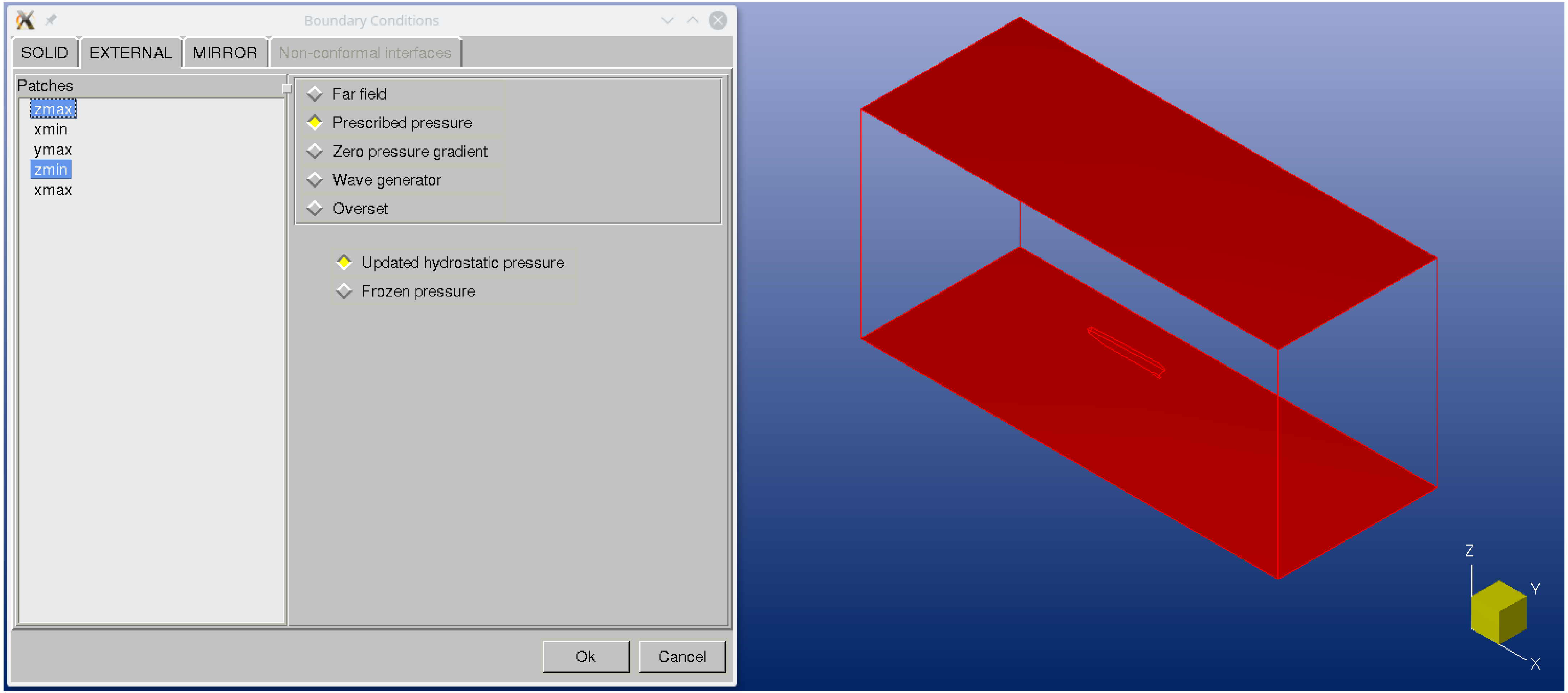
Define the **SOLID** boundaries as in the following pictures:

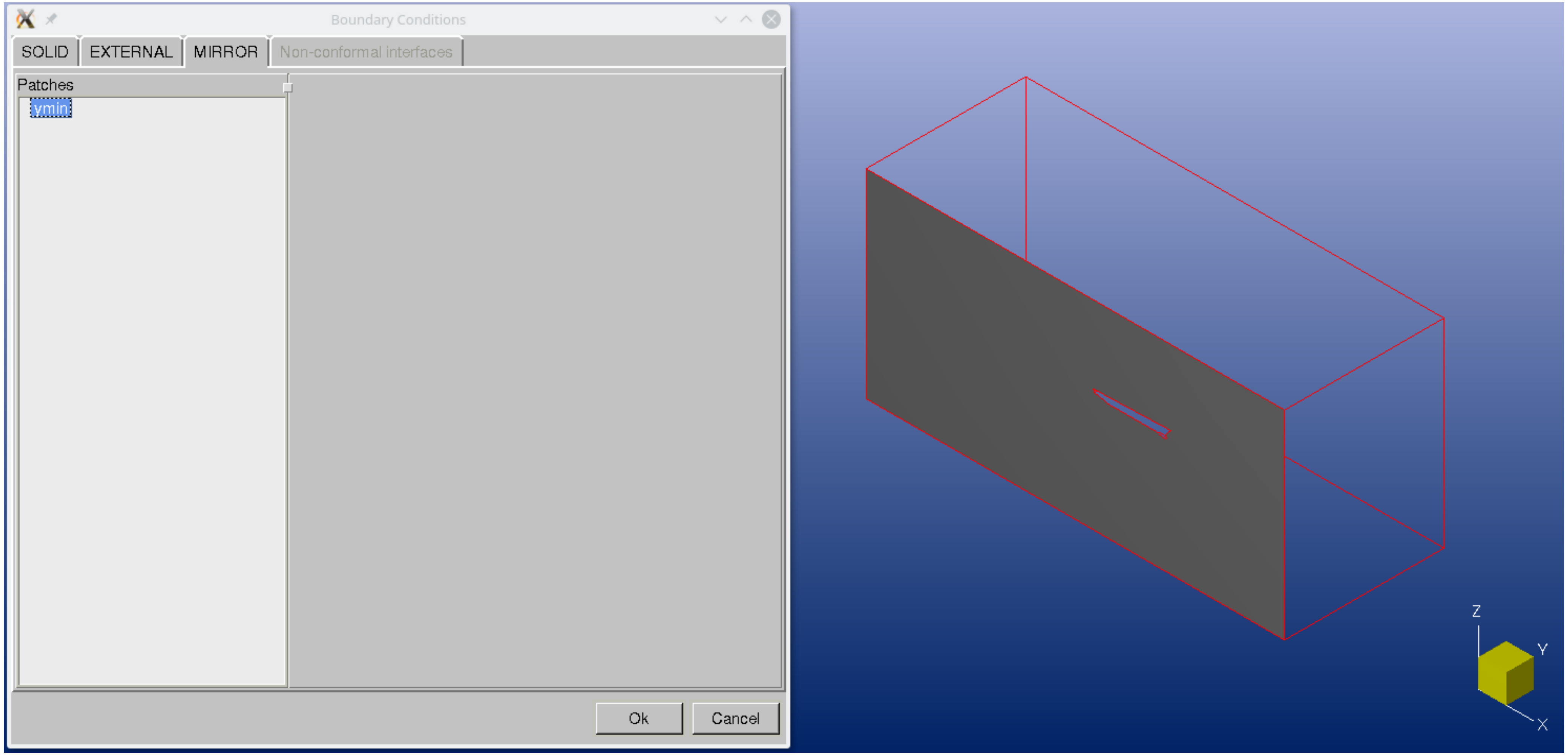
Note that the **Deck_b1** patch is defined as a **Slip** wall in order to correspond to the choice of not inserting a viscous layer mesh on this surface.

Define the **ETERNAL** boundaries as in the following pictures:



Boundary Conditions
SOLID
EXTERNAL
MIRROR
Non-conformal interfaces
Patches
zmax
xmin
ymax
zmin
xmax
Far field
Prescribed pressure
Zero pressure gradient
Wave generator
Overset
Profile type: x
Far field velocity
Vx Constant Value 0.0 [m/s]
Vy Constant Value 0.0 [m/s]
Vz Constant Value 0.0 [m/s]
Mass fraction
Default value
User defined value
Turbulence
From turbulence level initialization
Turbulent kinetic energy Default Value 2.694e-10 [m²/s²]
Turbulent dissipation Default Value 6.247e-12 [m²/s³]
Turbulent frequency 0.2577 [1/s]
Ok
Cancel
Z
Y
X

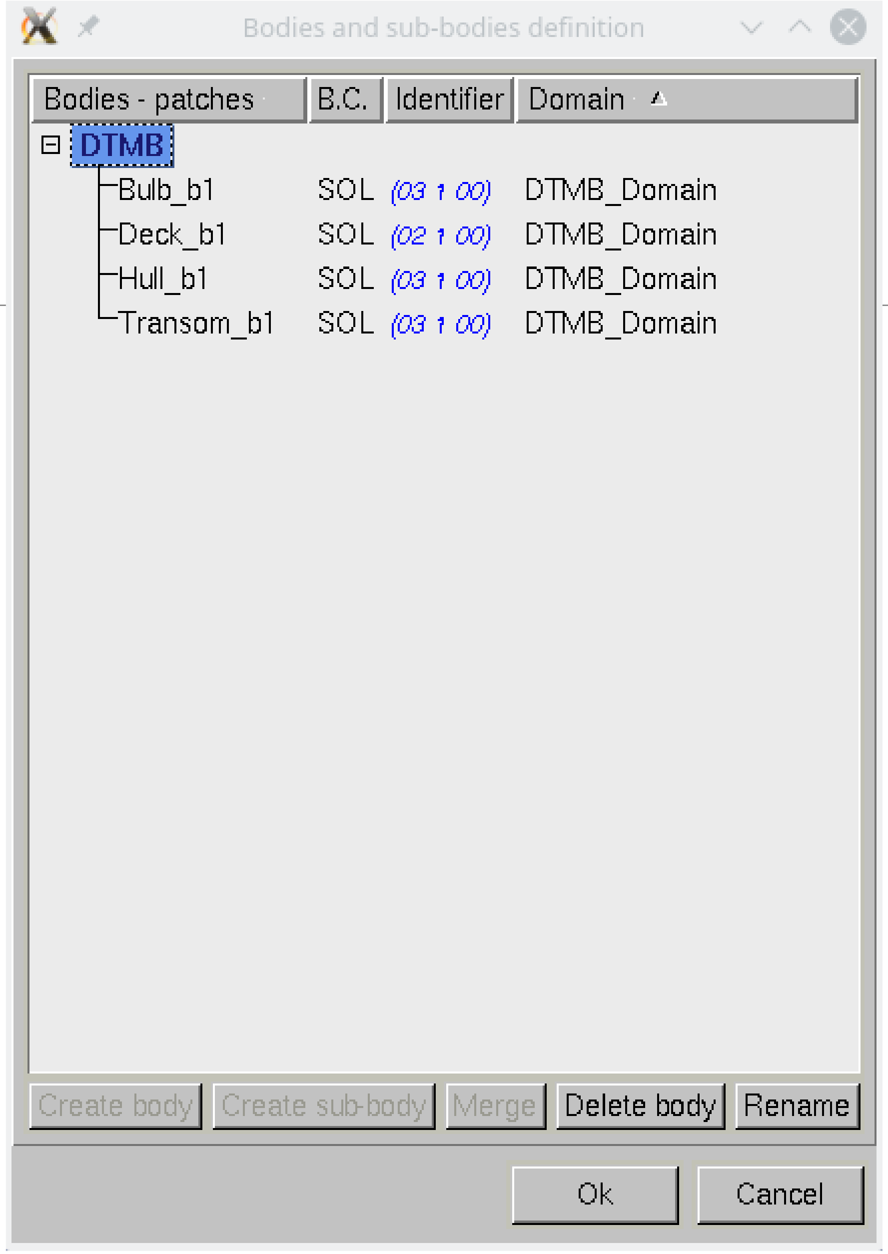

Boundary Conditions
SOLID
EXTERNAL
MIRROR
Non-conformal interfaces
Patches
zmax
xmin
ymax
zmin
xmax
Far field
Prescribed pressure
Zero pressure gradient
Wave generator
Overset
Updated hydrostatic pressure
Frozen pressure
Ok
Cancel
Z
Y
X


Boundary Conditions
SOLID
EXTERNAL
MIRROR
Non-conformal interfaces
Patches
ymin
Ok
Cancel
Z
Y
X

Body definition

In order to solve the entire body motion, it is now important to group the boat patches under a single body. To do so:

- Open the **Body definition** window.
- Select all the **_b1** patches.
- Click on **Create body**.
- Set the body name to **DTMB**. The result is shown on the picture below:



Domhydro tool

Now, in the **Body Motion** window, the DTMB motion can be defined. However, some physical values are required in order to properly describe the dynamic parameters of the DTMB:

- Center of gravity coordinates.
- Mass in kg.

In this case, the **h2p** (Hydrostatic Position => Parameters) mode is executed in order to determine the mass and the boat center of gravity. To do so:

Go back to HEXPRESS™ .

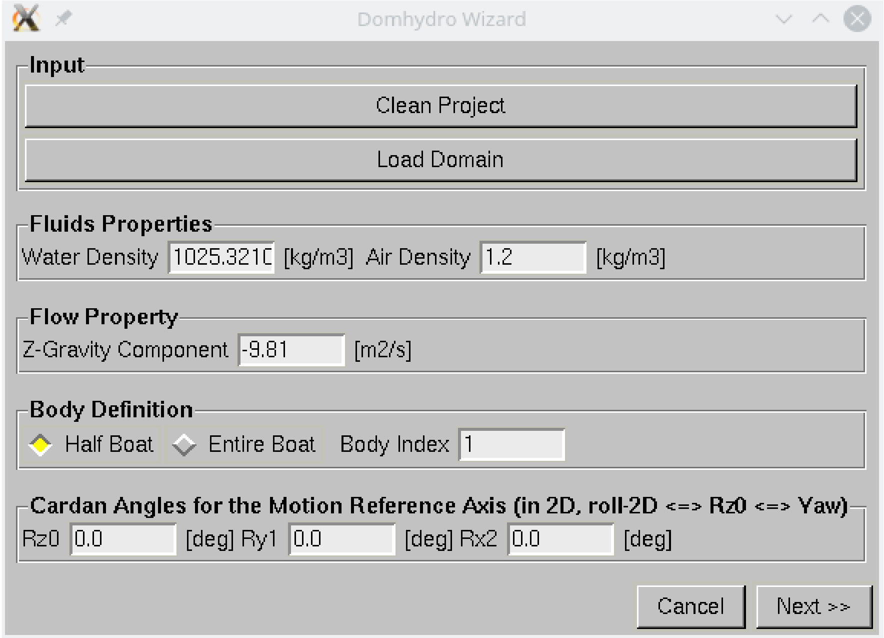
Launch the **Domhydro** tool under **Plugins/Marine/** menu.

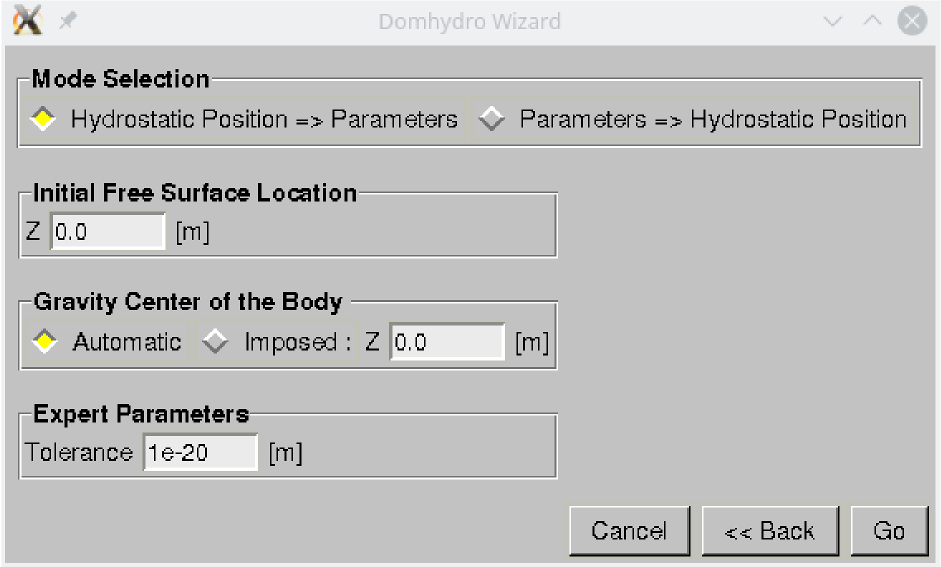
Enter the **Water Density** and precise the **Half Boat** definition of the body as shown below:

CG and mass via domhydro

Click on **Next**.

The default **Initial Free Surface Location** is well defined for this geometry (**Z = 0.0 m**).

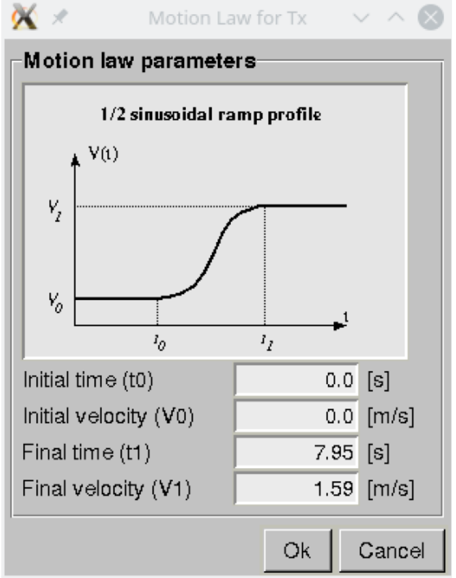


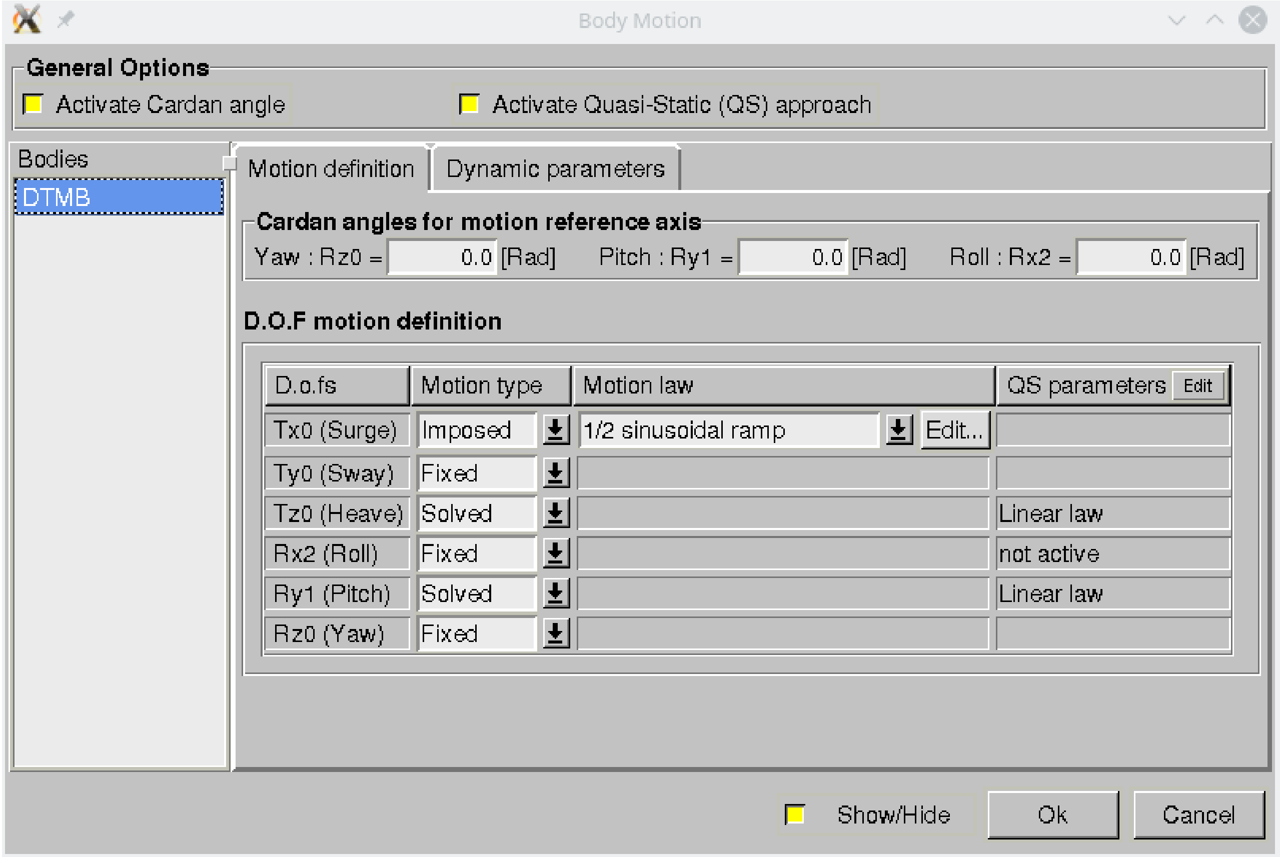


Press **Go** to execute the tool.

Body motion

Edit the **1/2 sinusoidal ramp** as following:





The final time value **t1** has been defined in order to reach an acceleration of during the velocity ramp. This acceleration has been determined to be the optimum value between speed and stability of the computation.

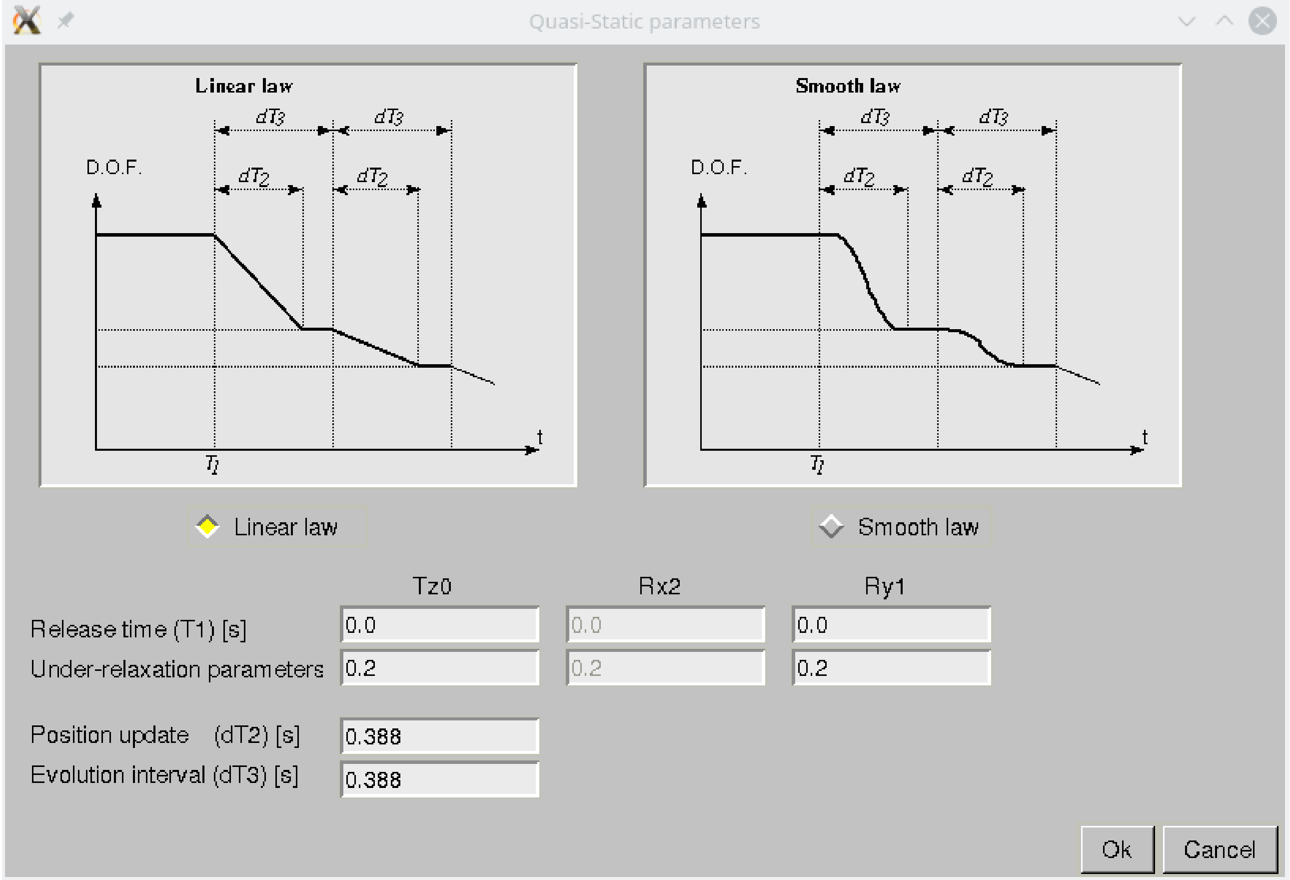
Quasi-static parameters

The **dT2 = dT3** choice comes from the NUMECA experience of resistance computation. It is a general guideline to follow for this kind of computations using the **Quasi-Static approach**.

Click on **Ok**.

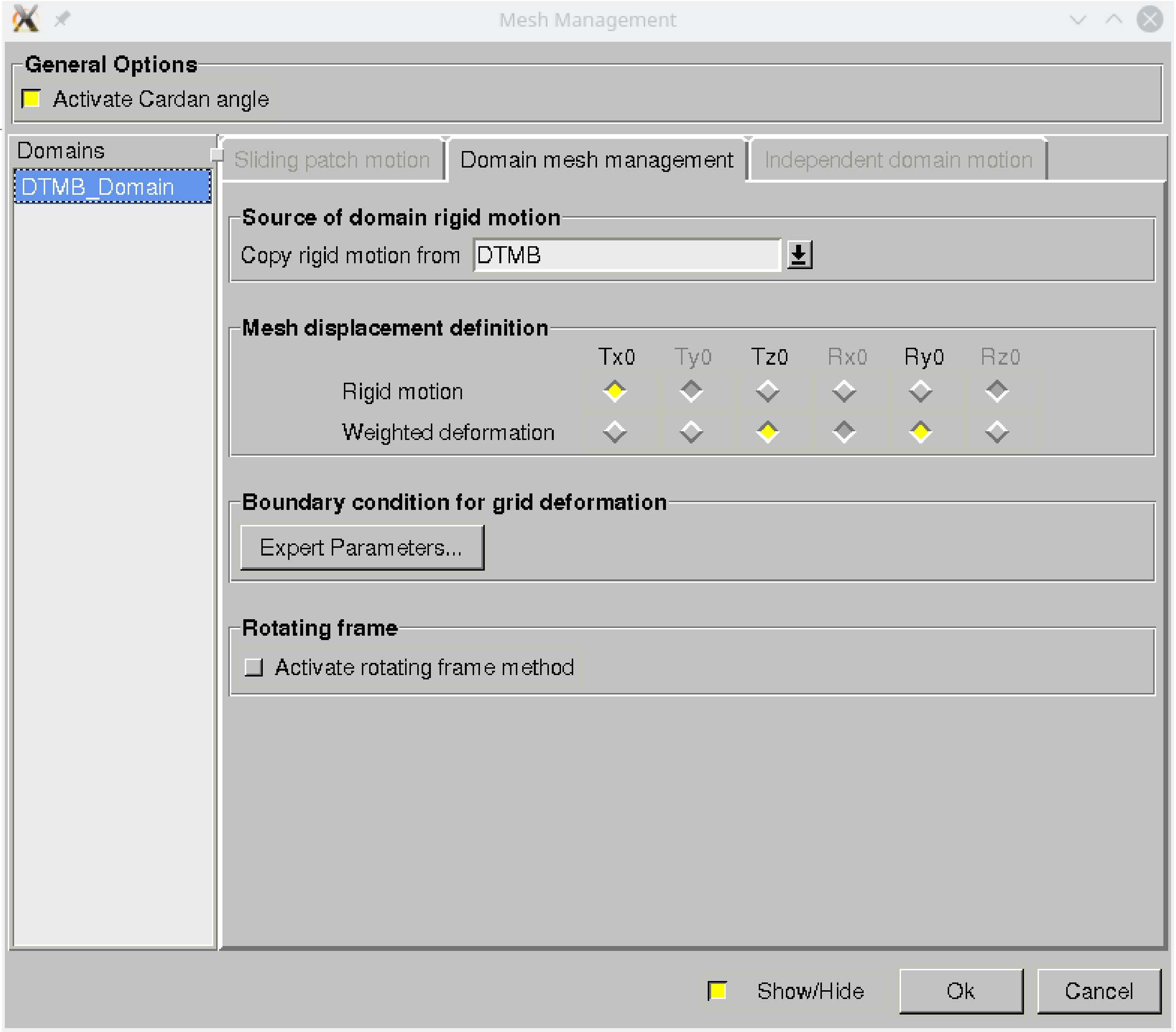
Under the **Dynamic parameters** tab, set the previously determined physical values into the **Inertial data** tab.

The **Quasi-Static approach** is based on a succession of predicted body attitudes, this is why the inertia tensor of the boat is not required (see Quasi-Static Approach for further details). However, not using this method would impose to collect these values from the previous Domhydro analysis.



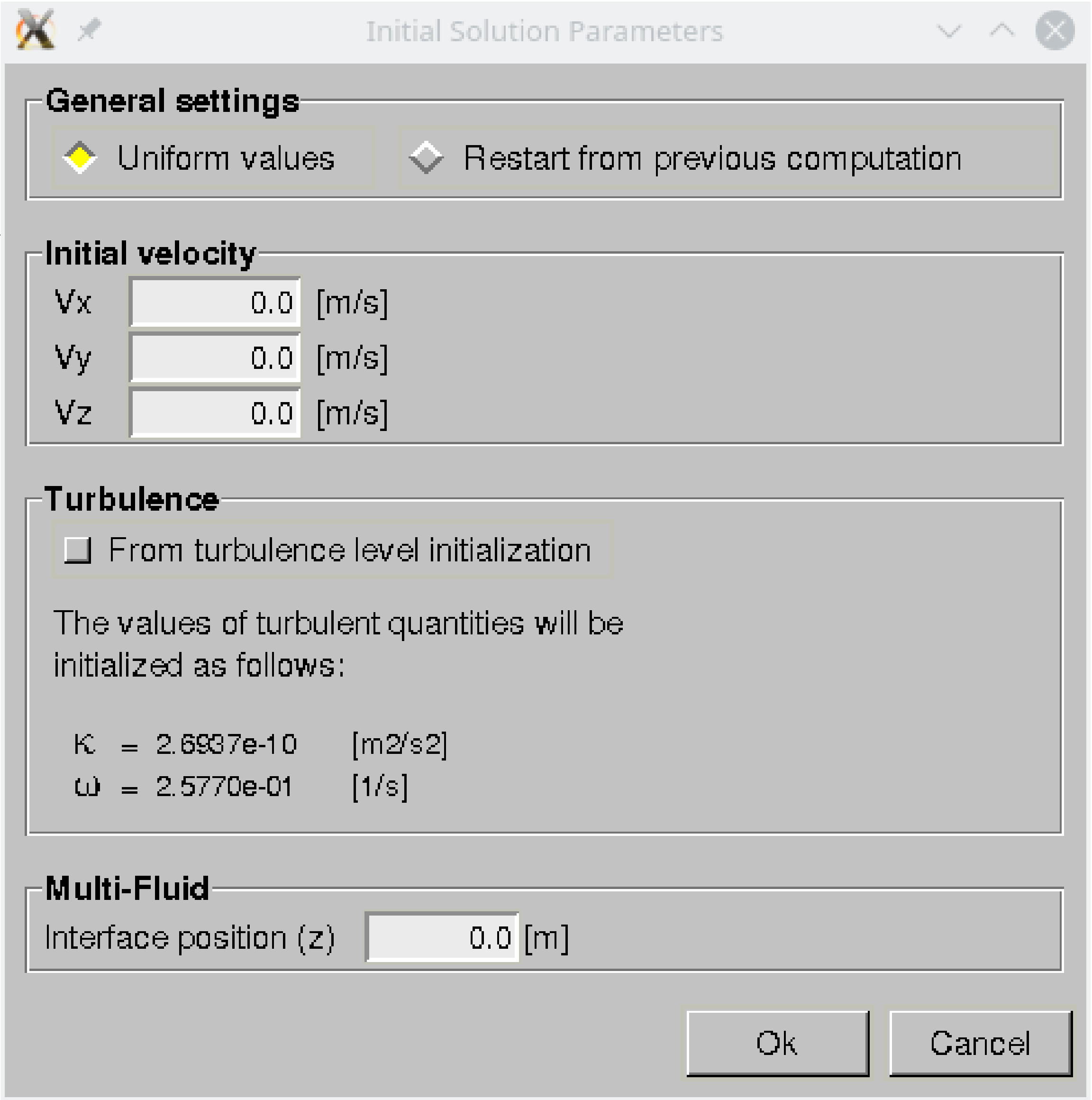
Mesh management

Verify the following parameters in the **Mesh Management** menu:



Initial solution

Once again, the default initial **Interface position (z = 0.0 m)** is well defined in the **Initial solution** window



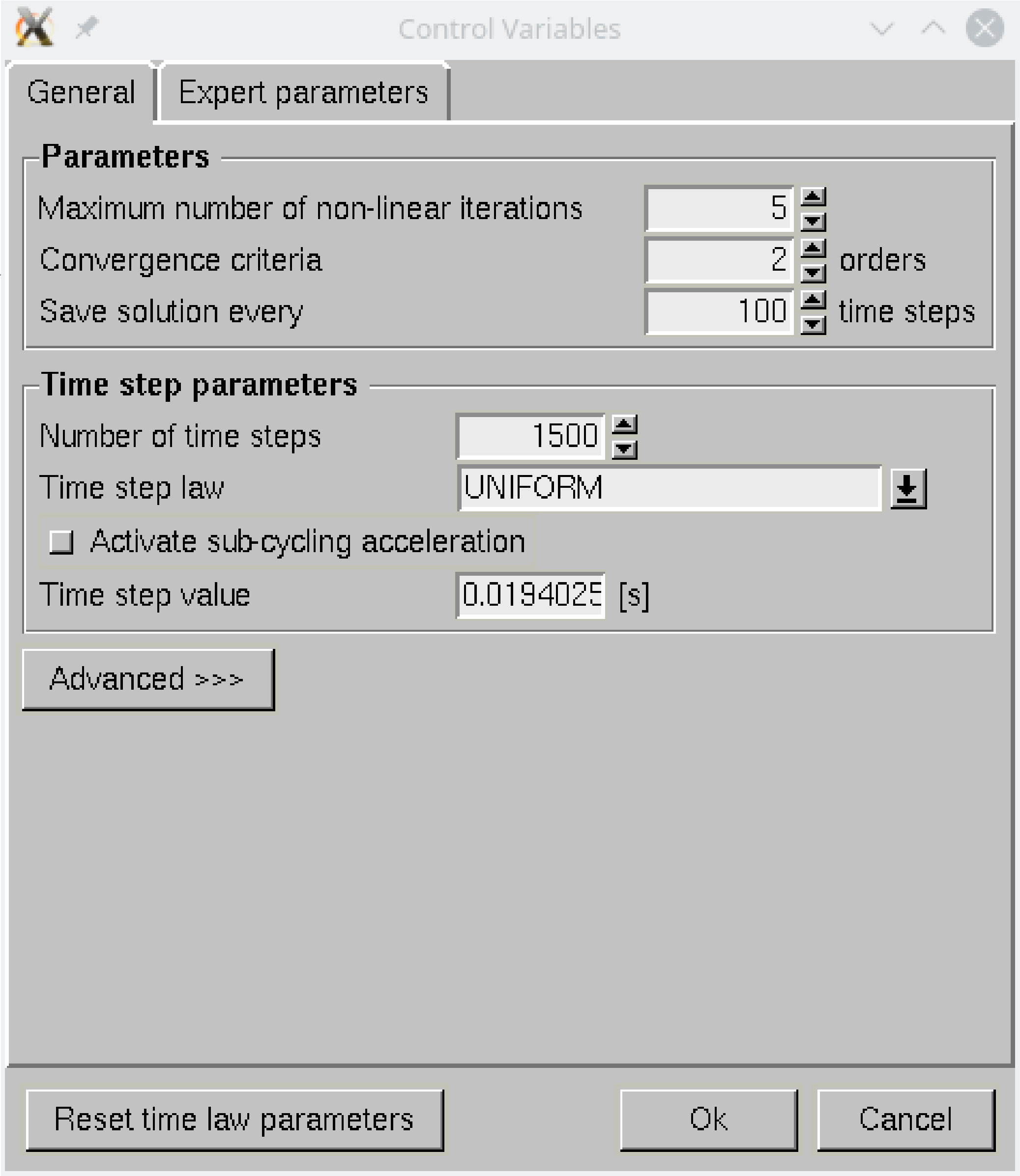
Computation control

In the **Control Variables** menu, set the following parameters and press **Ok**:

The **Time step value** has been automatically computed from the reference values of the flow, according to the following empirical formula:

$$dT = 0.005 \times \frac{L_{ref}}{V_{ref}}$$

Combining this **Number of time steps** and this **Time step value** will enable the computation to reach approximately **29 seconds**.

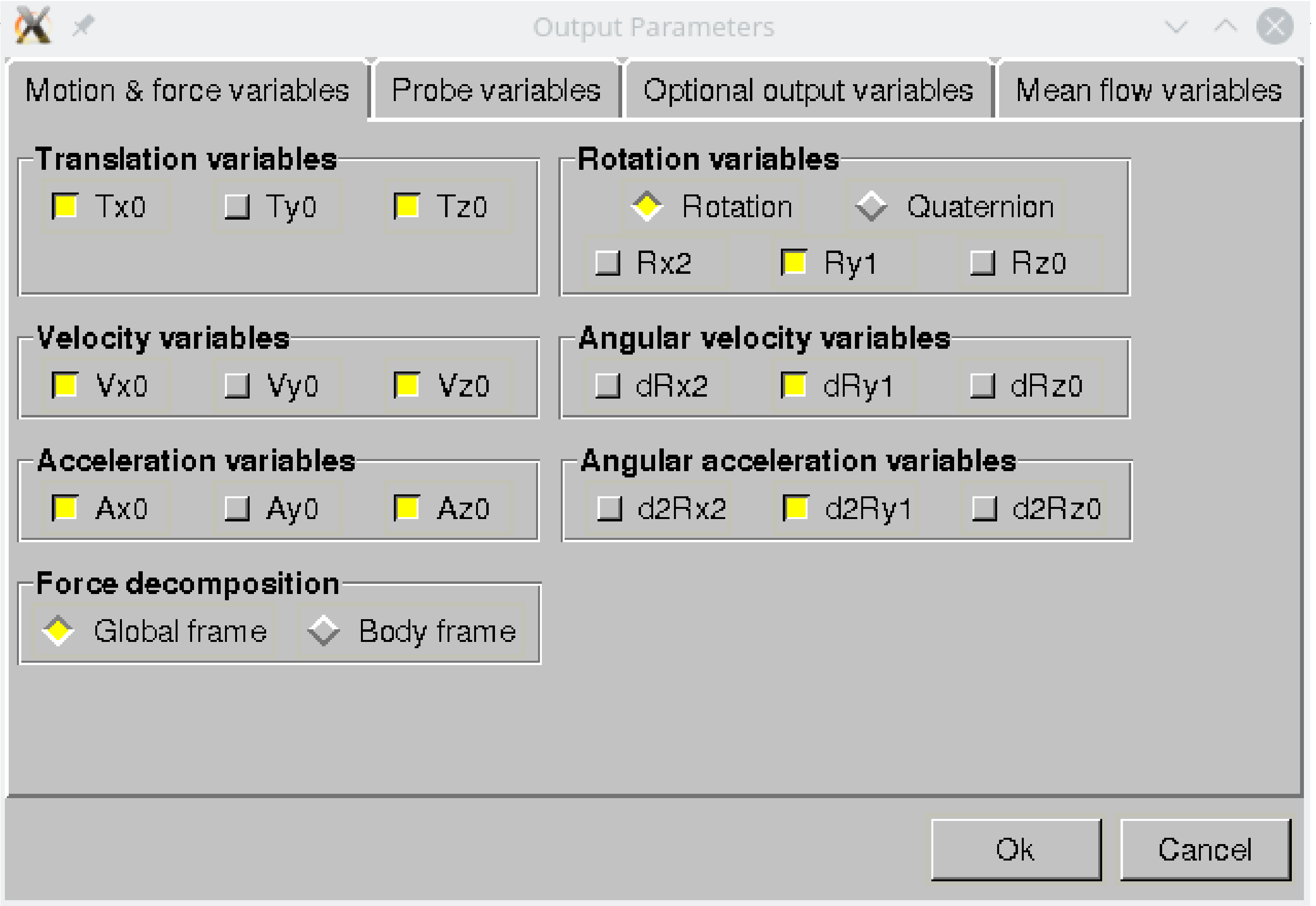


Outputs

After opening the **Outputs** menu, verify all parameters are set as below:

These variables will be recorded during the computation **every 100 time steps** according to the **Save solution every** value defined in

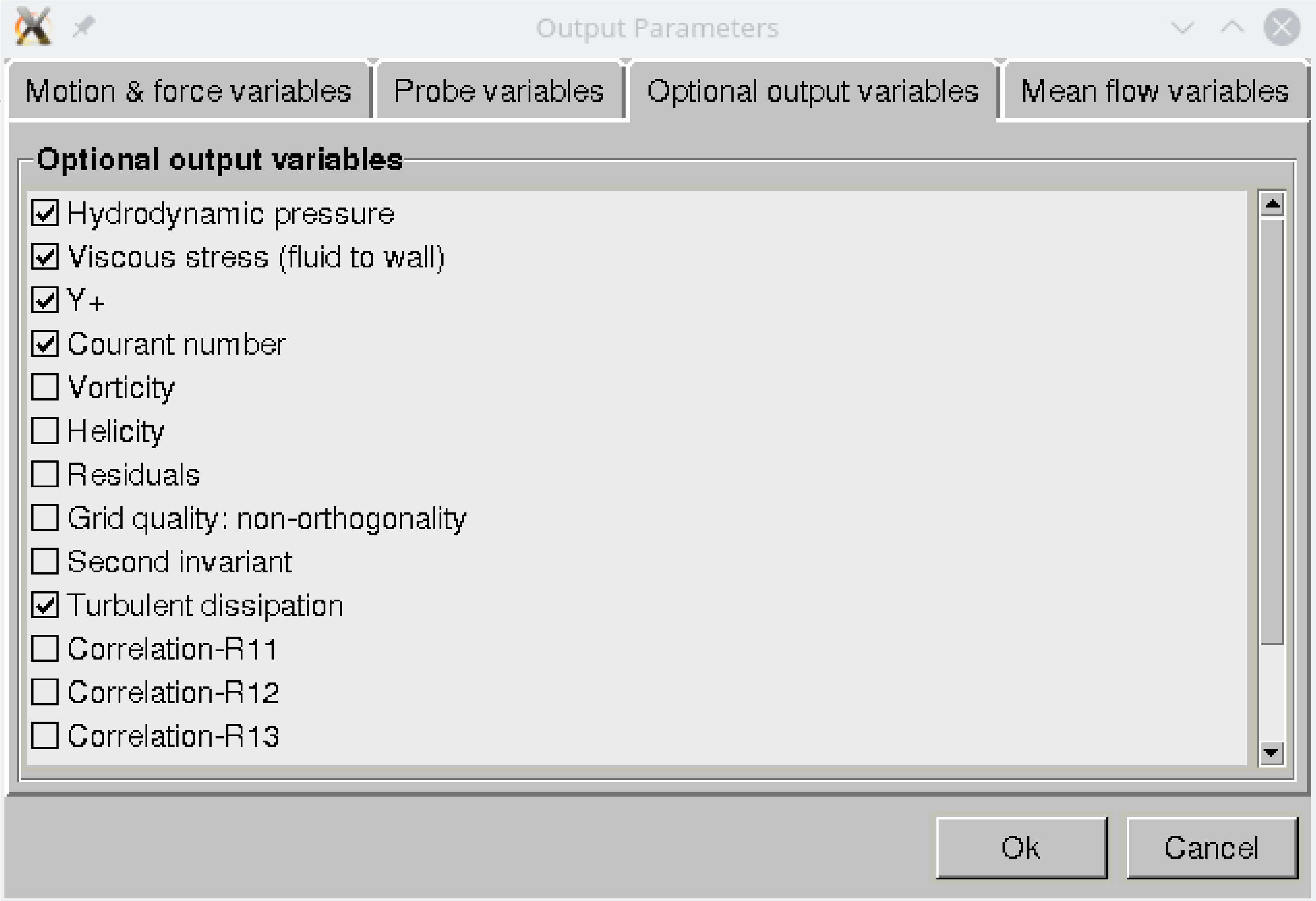
the **Control Variables** menu. Moreover, a record will be performed at the last time step of the computation.



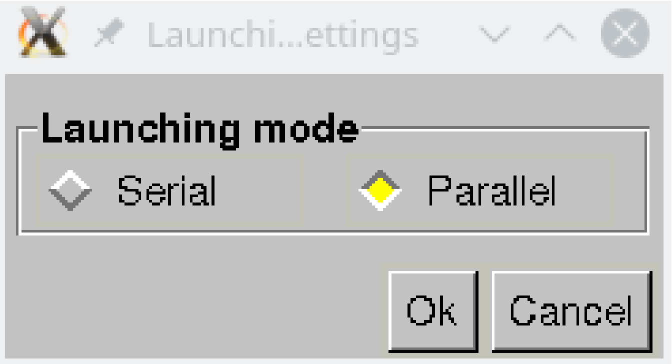
These variables will be recorded during the computation **every 100 time steps** according to the

Save solution every value defined in the **Control Variables** menu. Moreover, a record will be

performed at the last time step of the computation.



Launch & control the computation



Parallel computations are launched through the Task manager.

To do so, follow the steps below:

Start the computation by clicking on .

Select **Parallel** and click on **Ok**.

The **Task manager** is loaded, a task is created and the corresponding simulation file is automatically selected:

Enter the number of partitions

It is recommended to assign a maximum of 300 000 cells per partition, for each GByte of RAM available.).

Select the machines on which to run the computation by clicking on **Machines selection & balancing**.

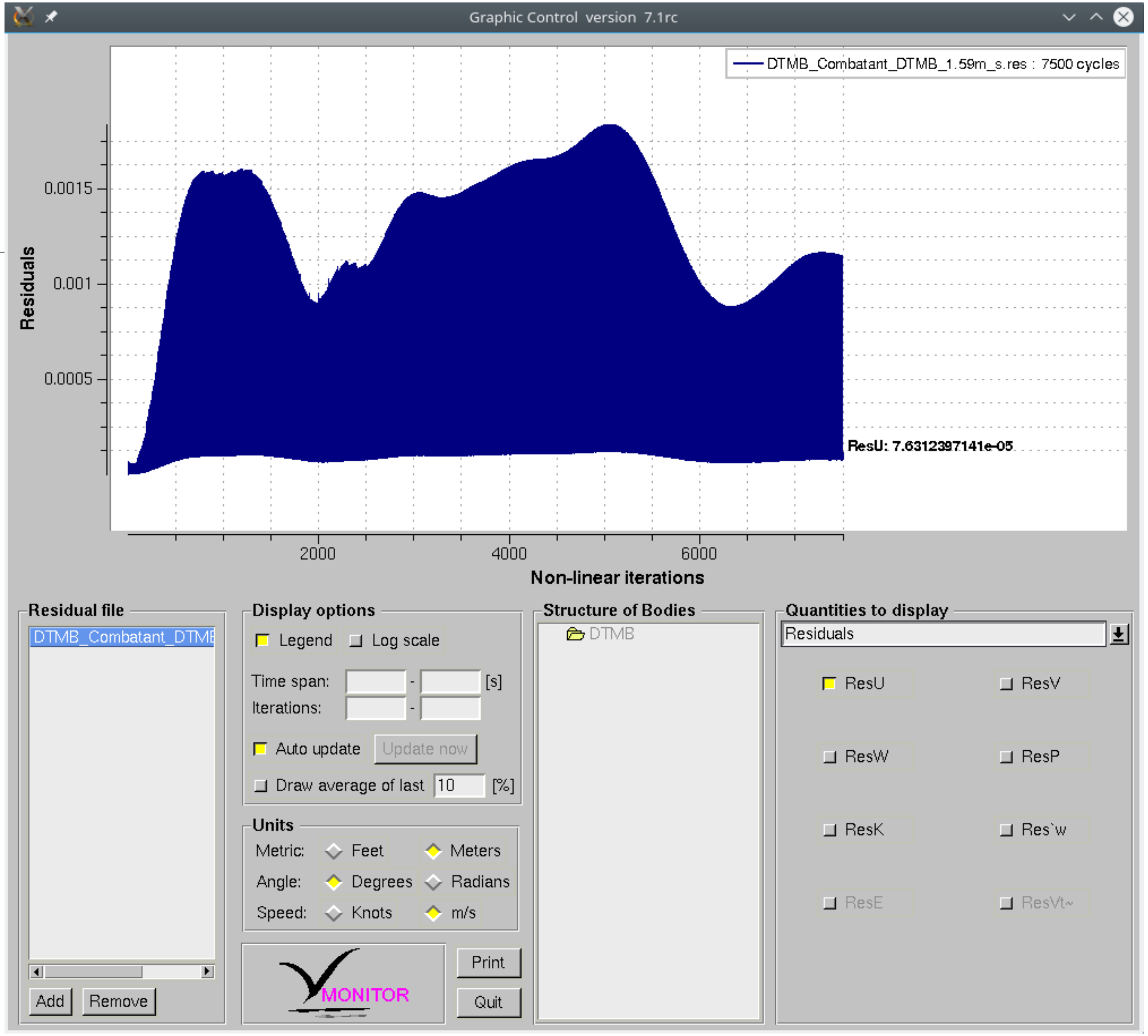
Click on the **Start** button to run the selected computation.

Monitoring

Click on the **Close** button to go back to the FINE™/Marine GUI.

Press the **Start Monitor** button in the FINE™/Marine GUI and on **Ok** to open the monitor.

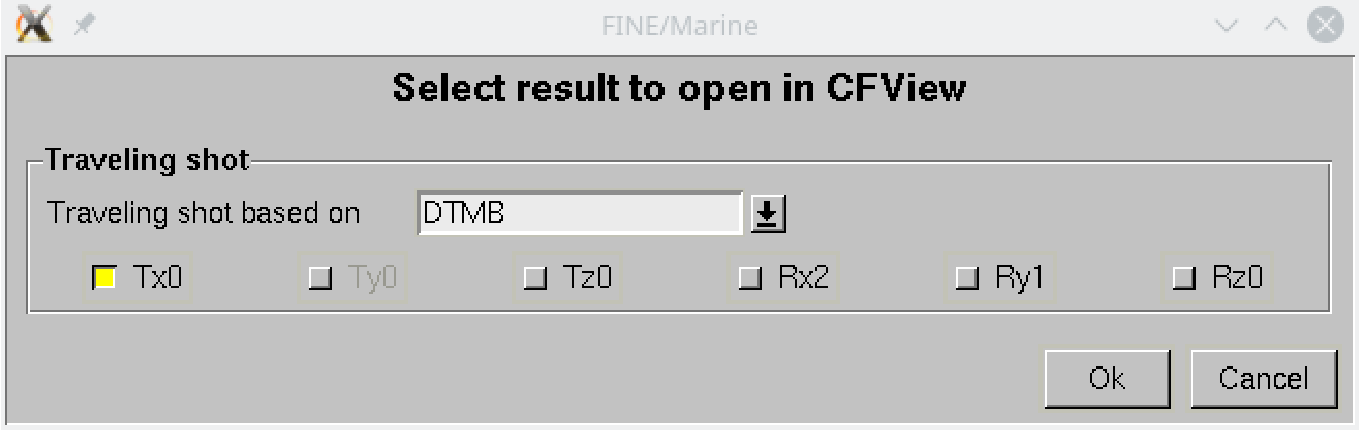
In the **Quantities to display** menu, it is possible to follow the convergence history for a variety of quantities (residuals, motions, forces and moments). Multiple components or quantities from different computations can be displayed together as presented below.



Post processing via CFView

Press the CFView™ button in order to open it.

Press **Ok** on the following window:



Visualize solid patches

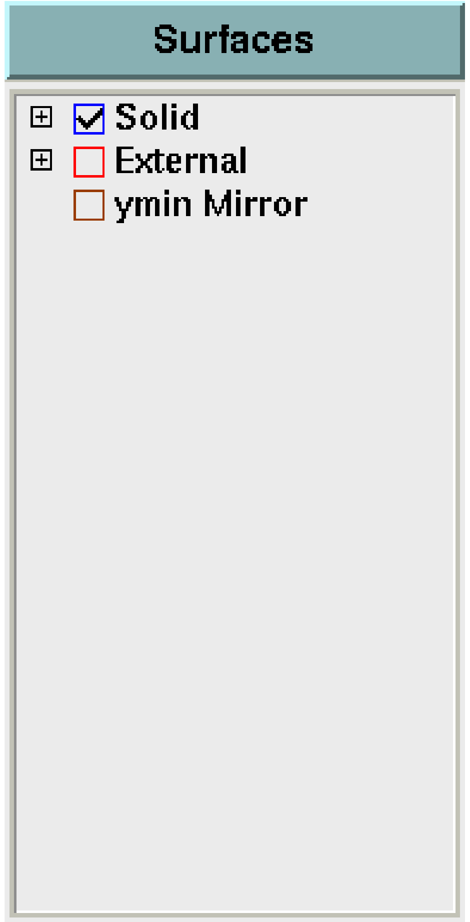
By default, the solid patches are selected.

However, it is always more convenient to group patches by boundary condition. There is two ways of doing this in order to obtain the Picture below:

- Select the patches by type, right-click on it and release on **Make Group**
- Under the **Macro** menu, the **Group_ patches_ by_ type** plugin performs the grouping automatically.

A half DTMB geometry has been input. Therefore, it would be more appropriate to represent the entire body: under the **Geometry** menu, click on **Repetition on/off**:

Click on the **Render Gouraud** icon to view the shaded body.



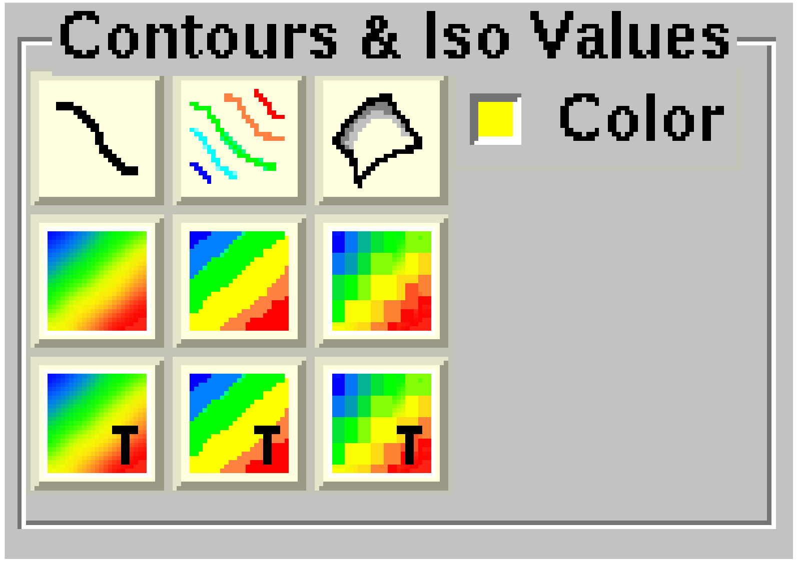
Observe wave surface elevation

Double click on **Mass Fraction** from the **Quantities** to select it (the quantity will be marked by a green tick when active)

In the **Representations** area, click on the **Contours & Iso-Values** section to expand it.

Click on the **Iso-Surface** icon and enter the value **0.5** in the keyboard input area, press **Enter**, then click again on the **Iso-surface** icon to add the iso-surface in the list of available surfaces.

The value of **0.5** of the mass fraction represents the contact surface between the two fluids: water and air.



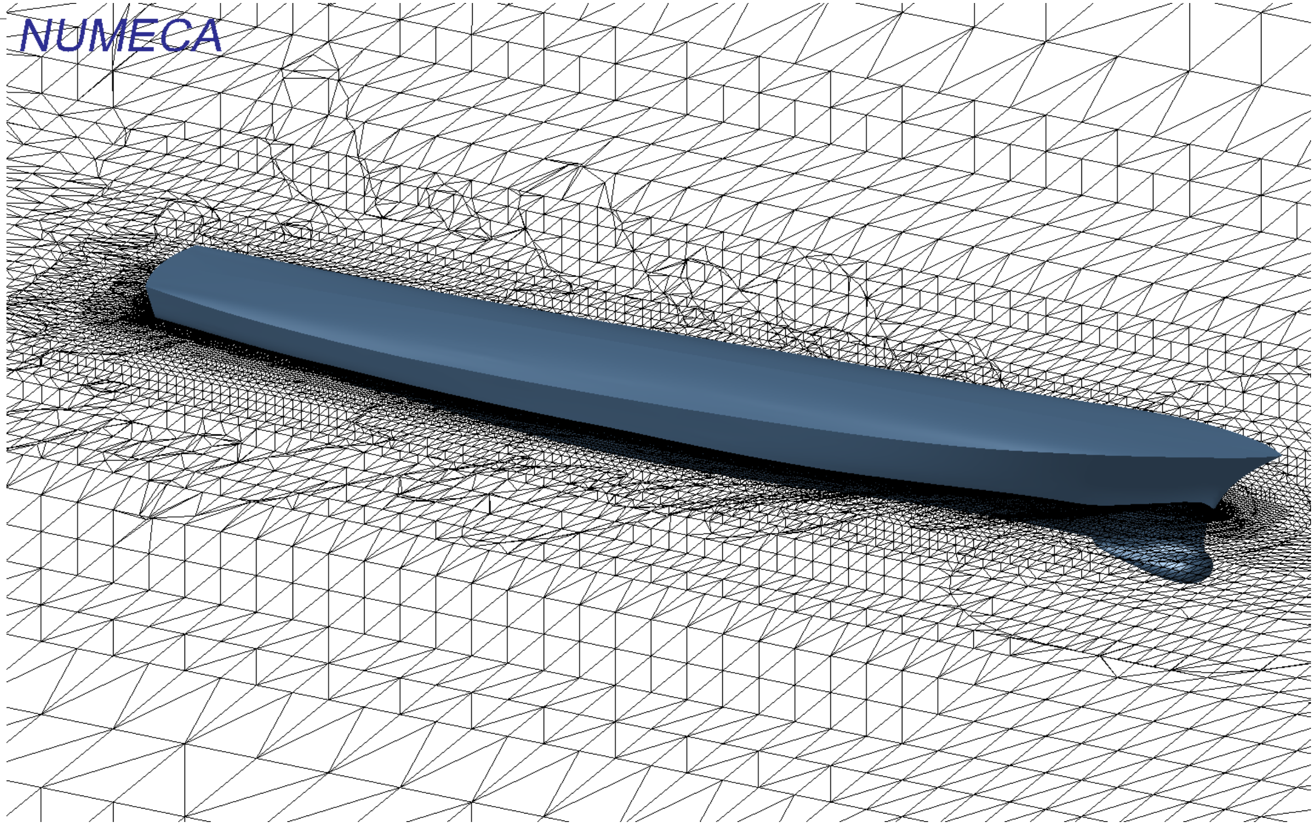
A new surface called **ISO Mass Fraction=0.5.D1** has been added in the **Surfaces** list and is displayed as a grid as presented below:

Click on **ISO Mass Fraction=0.5.D1** in the list, then right-click and release **Select**.

Press the **New** button to create a new quantity, as presented below:

Set **Name** as **Elevation**.

Set **Definition** as **z** (vertical position).



Wave elevation

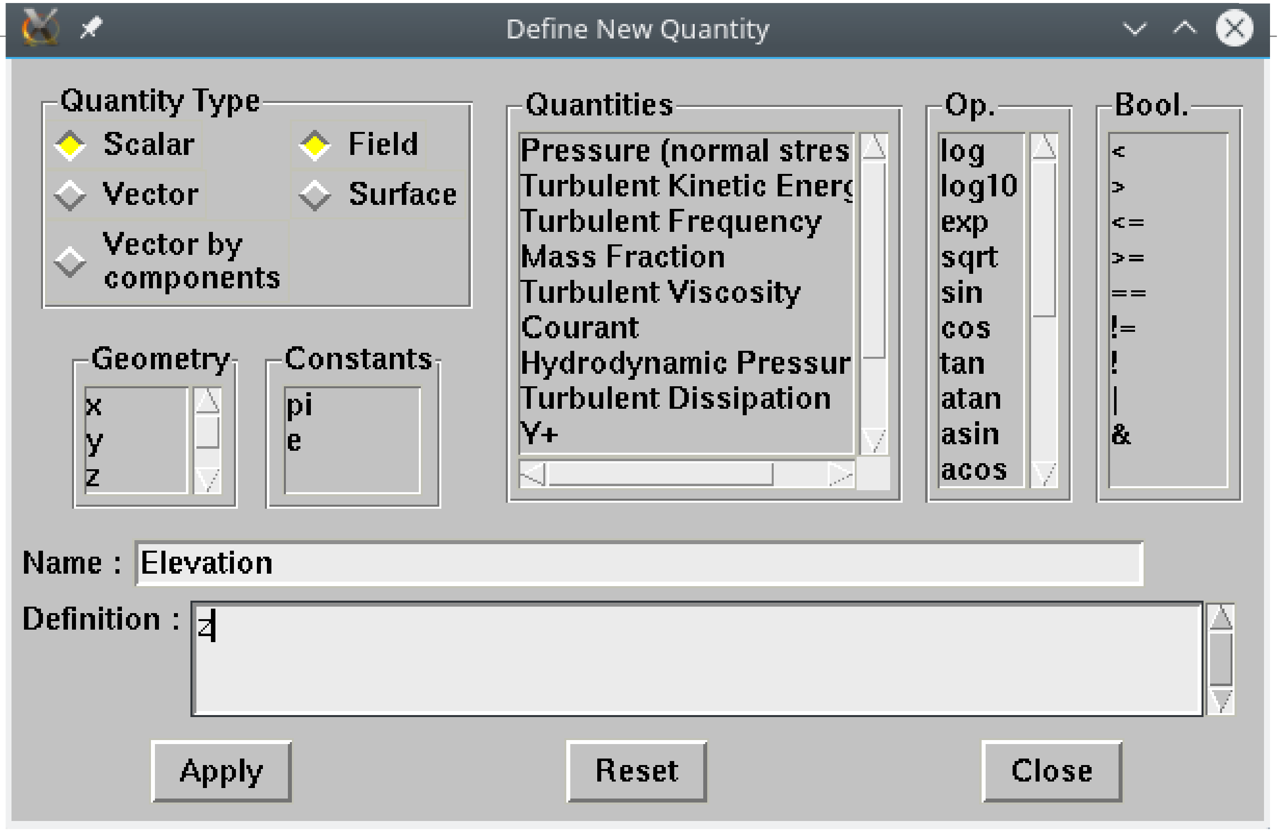
Select **Apply** and **Close**. By default the new quantity **Elevation** will be active.

Click on the **Smooth Contour** icon in **Representations/Contours & Iso Values** to visualize the colormap.

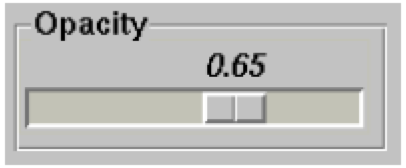
The pallet of colors can be optimized to the selected surface by clicking on the **Colormap Optimum Range** icon in the toolbar.

Switch off the grid by clicking on the **Toggle Grid** icon in **Representations/Grid**.

Click on **Opacity** in **Representations** and decrease the value to about **0.65** by dragging the cursor (by default set to 1):



Wave elevation



Select **Apply** and **Close**. By default the new quantity **Elevation** will be active.

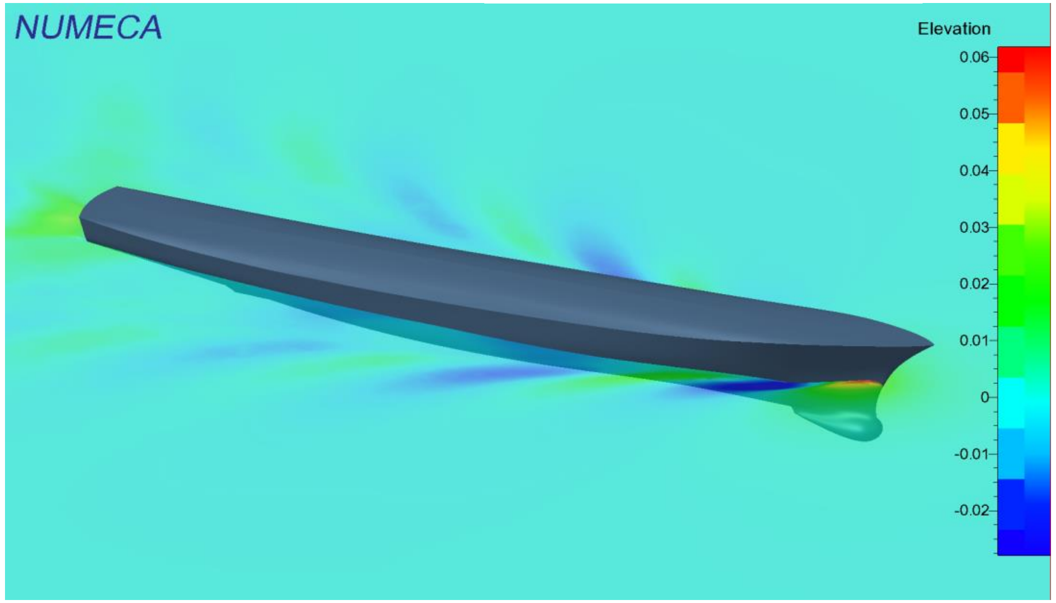
Click on the **Smooth Contour** icon in **Representations/Contours & Iso Values** to visualize the colormap.

The pallet of colors can be optimized to the selected surface by clicking on the **Colormap Optimum Range**

icon in the toolbar.

Switch off the grid by clicking on the **Toggle Grid** icon in **Representations/Grid**.

Click on **Opacity** in **Representations** and decrease the value to about **0.65** by dragging the cursor (by default set to 1):



Generate streamlines on free surface

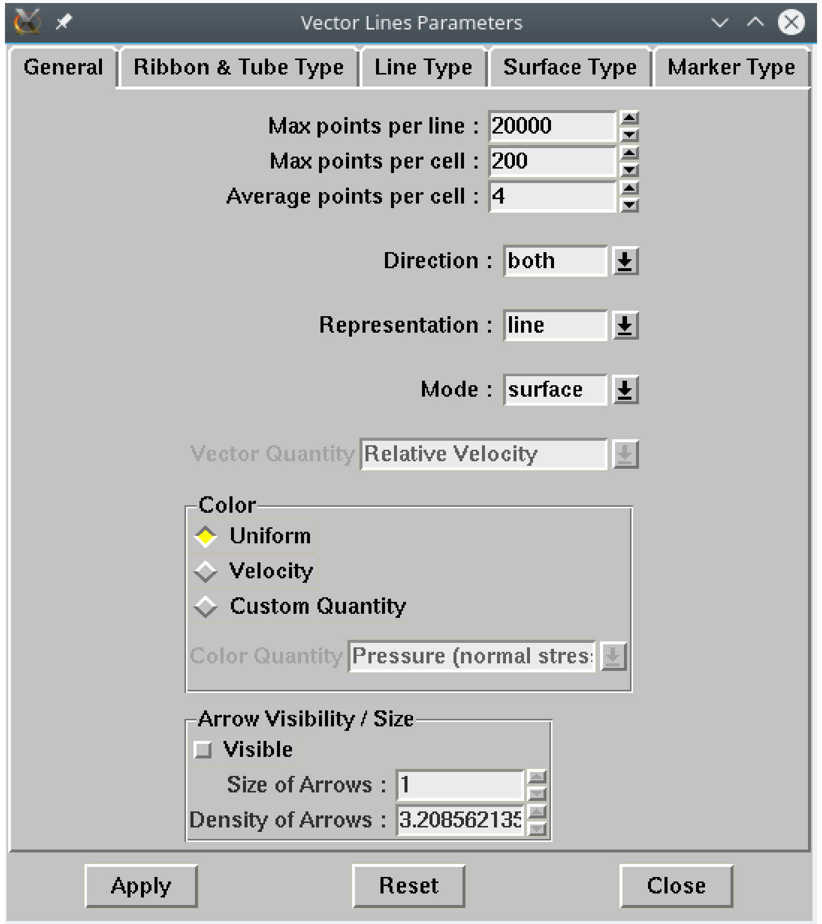
The above representation of color contour and isolines on free surface can also be obtained by clicking on the **Macros** menu and selecting **Represent_Free_Surface** option. This will automatically perform all the operations mentioned in the above steps and will give the representation of color contour and isolines as shown in the figure above.

Delete the **Isolines** by selecting **Update/Delete/Isoline**.

Select the **Relative Velocity** (double-click) in the **Quantities** menu and go to **Representation/Vector**

line/Parameters... or click on **Vector Lines Parameters** under **Representations/Vector Lines**.

In **General** tab select **both** as **Direction** and **surface** as **Mode**.



In **Line Type** tab set **black** color for the streamlines.

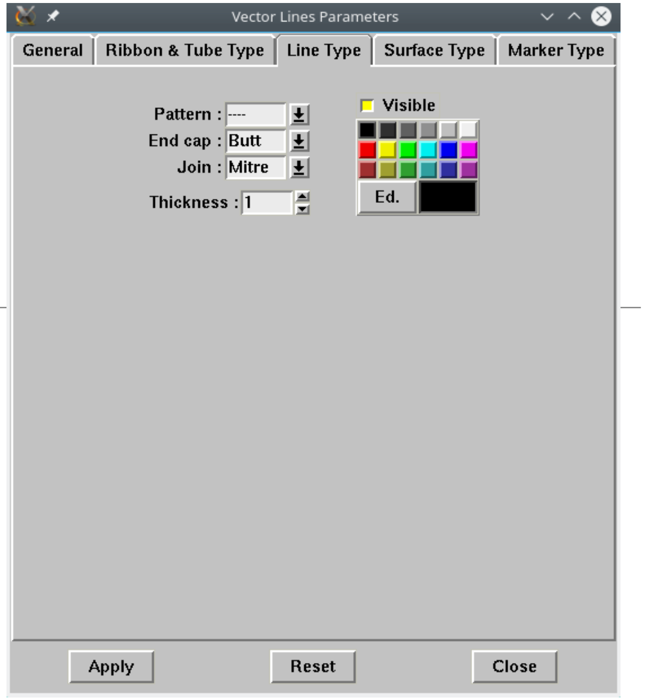
Click on **Apply** and **Close** the window.

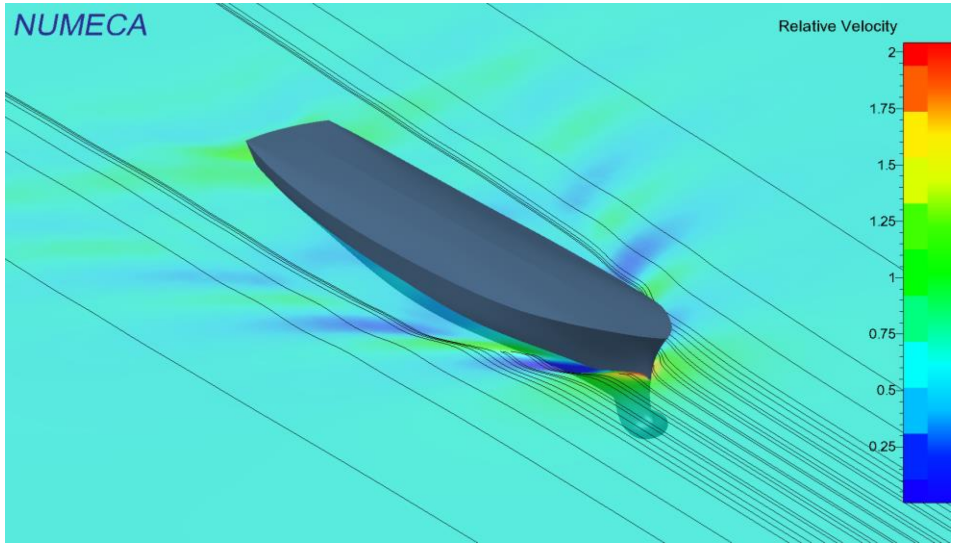
In the **Surfaces** list, check that only the **ISO Mass Fraction=0.5.D1** is selected.

Select **Representation/Vector Line/Local** menu or click on under **Representations/Vector Lines** and

draw some streamlines by clicking on the free surface (not on the mirrored side, otherwise nothing will be

displayed):





Calculating wetted surface area

Update the view by deleting everything: go to **Update/Delete/All**.

Check that all the solid patches are selected in the **Surfaces** list.

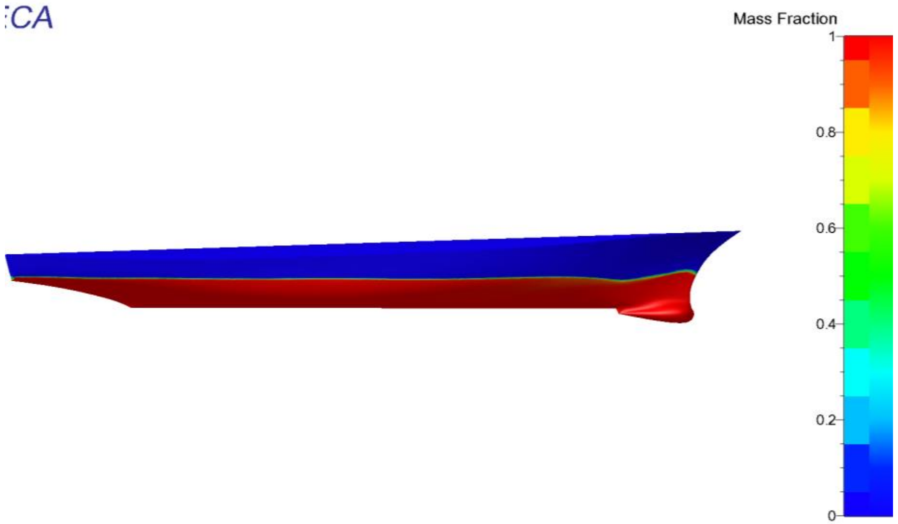
Click on the **Render Gouraud** icon to view the shaded body.

Select the **Mass fraction** in the **Quantities** area.

Compute the wetted surface value of the quantity by clicking on **Scalar Integral** icon under **Representations/Integrals**.

Read the result in the information bar:

Visualize it by pressing on the **Smooth Contour** icon in **Representations/Contours & Iso Values** to activate the colormap:



The above representation and calculation of the wetted area can also be obtained by clicking on the **Macros** and selecting **Computed_Wetted_Area**. This will automatically perform all the operations mentioned in the above steps.

Volume streamlines

Check all solid patches are selected in the **Surfaces** list, press the **Render Gouraud** icon to visualize the shaded body.

From the **Quantities** list, select **Relative Velocity**.

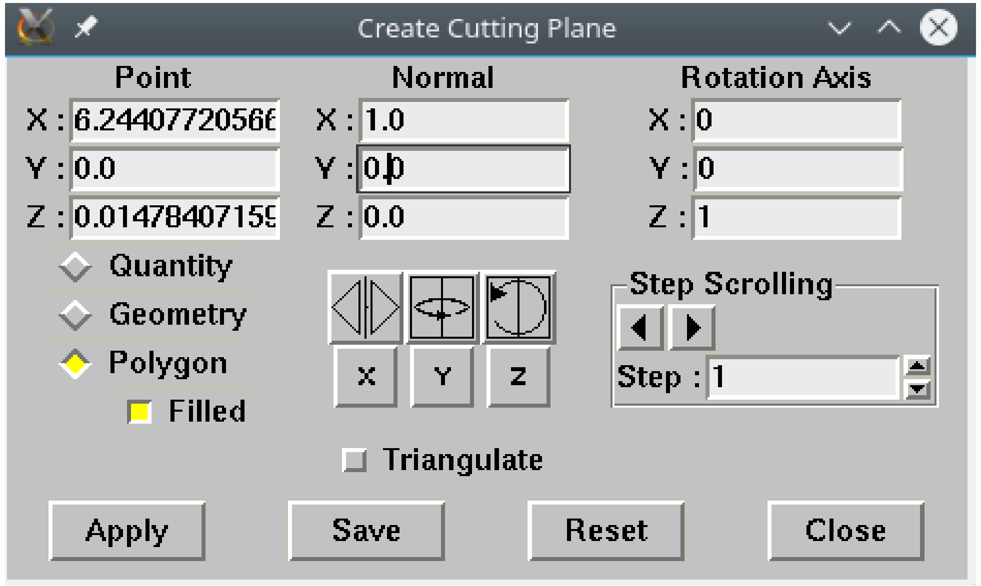
Press on the **Y Projection** button to see the ship from aside.

Go to **Geometry/Create Cutting Plane...**

Press on the **X** button from the **Create Cutting Plane** window to generate a **Y-Z** plane.

Use the left-right arrows from the **Step Scrolling** to move the cutting plane just before the ship's bow. Decrease the **Step** value for more accurate translations, if needed.

Press **Save** and **Close** when the cutting plane is set as in the following figure.



The cutting plane has been added in the **Surfaces** list as **CUT1**.

Press on **X Projection** button for a frontal view and zoom in.

Click on the **Vector Lines Parameters** icon under **Representations/Vector Lines**. Under the **General** tab, set **both** as **Direction**, **tube** as **Representation**, **volume** as **Mode** and **velocity** as **Color**.

Click on **Apply**.

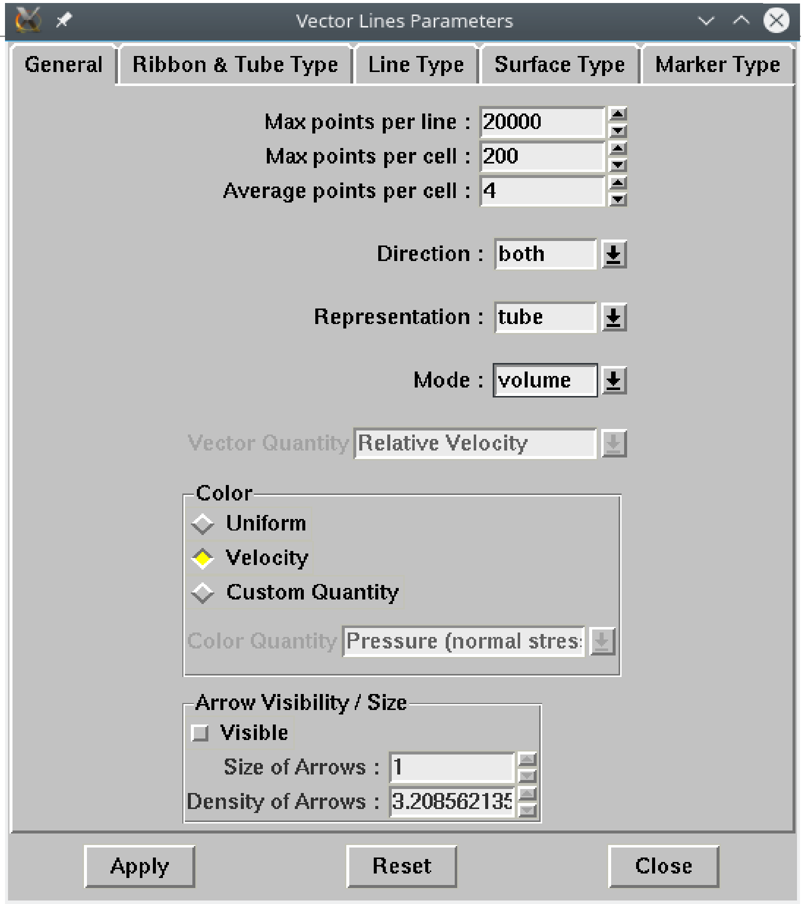
Go to the **Ribbon & Tube Type** tab and decrease the tube radius **r0** to **0.05**.

Click **Apply** and **Close**.

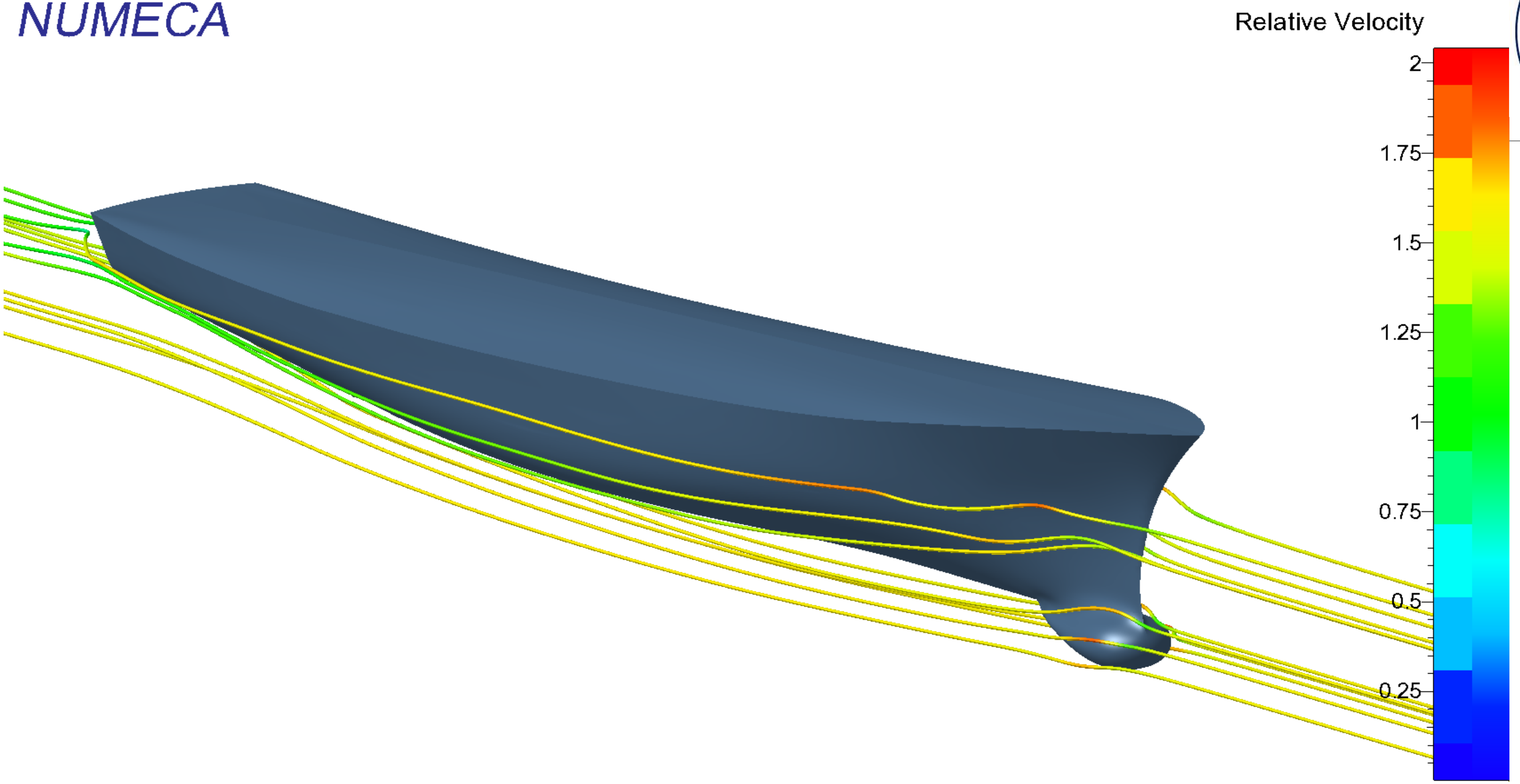
Select only the cutting plane **CUT1** in the **Surfaces** list.

Press the **Local Vector Line** icon under **Representations/Vector Lines** and then click on the cutting plane

where you want to draw the 3D streamlines from. The resulting streamlines are presented in the following figure:



NUMECA



References

Jurij SODJA , Turbulence models in CFD, Seminar, 2007.

Thombare A. S., Computational fluid Dynamics lecture notes.

Arvind Deshpande, Introduction to CFD lecture notes.

H K Versteeg and W Malalasekera, An Introduction to Computational Fluid Dynamics, The Finite Volume Method, Pearson Education, 2008.

S V Patankar , Numerical Heat Transfer and Fluid Flow –, Taylor & Francis, 1980.

John.D.Anderson, Computational Fluid Dynamics, The basics with applications, JR.,Mcgraw-Hill International edition, 1995.

K.Muralidhar and T.Sundararajan, Computational Fluid Flow and Heat TransferNarosa, 2007.

Ferziger and Peric, Computational methods for fluid Dynamics Springer, 2004.

A.W. Date , Introduction to Computational Fluid Dynamics, Cambridge, 2005.

www.cfd-online.com

Hi-Tech engineering services

Pointwise

ANSYS Fluent