TUTORIALS

FINETM/Marine 10.2

cādence®



www.numeca.com

CONTENTS

TUTORIAL 1. C-WIZARD RESISTANCE

1.1	Introdu	ction	3
	1.1.1	Problem Description	3
	1.1.2	Objectives	3
	1.1.3	CPU Prerequisites	4
	1.1.4	Estimated Engineering and Computing Time	4
	1.1.5	Preparation	4
1.2	Part I -	Project Setup	5
	1.2.1	Start C-Wizard	5
	1.2.2	C-Wizard Part 1: Create & Set up Project	6
		A. Create Project	6
		B. Set up project: Body configuration	8
		C. Set up project: Flow definition	10
		D. Set up project: Additional inputs	12
1.3	Part I -	Domain & Mesh Setup	16
	1.3.1	Mesh Setup	16
	1.3.2	Mesh Generation	24
1.4	Part II	- Flow Settings & Post Processing	26
	1.4.1	C-Wizard Part II: Flow Settings	26
		A. Estimate Hydrostatic Values	26
		B. Flow Settings	27
	1.4.2	Launch & Control Computation	34
	1.4.3	Post Processing	38
		A. Results analysis tool	38
		B. CFView TM	47

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 FINETM/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/m³;
- Kinematic viscosity: 1.19e-6m²/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 FINETM/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 FINETM/Marine v10.2 graphical user interface with the following command line:

finemarine102 -print

• For Windows systems (Windows 7 and older), you can access the FINETM /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 FINETM /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 FINETM/Marine interface.

1.1. Open FINETM/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.

Create a new project							
◇ Importing a mesh							
♦ Creating a mesh							
Using the C-Wizard							

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

X 💿	C-Wizard		\sim \sim \times			
⊢War	ing					
	An ongoing project has been detected. Unsaved data will be lost. Do you want to proceed with the C-Wiza					
		No.	Na			
		Yes	No			

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 FINETM/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.

		C-Wizard 🔶 🗆	×
	9	WIZARD	
Project Managemen		DN FLOW ADDITIONAL MESH DEFINITION ADDITIONAL SET-UP	
Project man	agement		
		Create project	1
Application	ce 🔷 Seake	eping 🔷 Open water 🔷 Planing Regim	e
-Fluid model	l 🔷 Mono-flu	uid double body 🔷 Mono-fluid underwater	
-Wizard units	;		
Angle:	🔶 deg	🔷 rad	
Length:	🔶 m	♦ ft	
Speed:	🔶 kt	♦ m/s	
		Cancel Next >	>

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.

X	C-Wizard			\sim \sim \otimes
		TIONAL ME		
Input geometry				
 Parasolid/CATPart STL/Domain Loaded file: None 	Existing mesh	Import Para	asolid/CATF	Part file
-Body orientation-				
CoG to bow: 🔷 Positive X-axis 💊 Negat	ive X-axis			
CoG to side: 🔶 Positive Y-axis 🗇 Negat	ive Y-axis			
Is the body aligned with the Cartesian at Yes Ves No	xis?			
Body reference length Automatic (= LOA)	♦ User-defined			
Initial free surface position				
\diamond Automatic (based on body mass)	Oser-defined	Z-coordinate	0.0	[m]
Body mass				
 Automatic (based on initial free surface) 	📀 User-defined			
Center of gravity				
 Automatic (based on initial free surface) 	♦ User-defined	Specify z0	CoG	
Adjust pitch for hydrostatic equilibrium Trim: 🔶 Frozen 💠 Free	?			
Body motion(s) to solve during simulation Trim Sinkage Surge	on			
		Cancel	<< Back	Next >>

7

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).

Input geometry							
Parasolid/CATPart	💠 STL/Domain	♦ Existing mesh	Import Parasolid/CATPart file				
Loaded file: None							

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.

K	C	-Wizard			~ ^ 🛛			
Input geometry					A			
Loaded file: KCS_hull_SVA_cf_w	thnames cf2.	x t						
The loaded file contains: 🔶 Hal	_	_						
-Body orientation								
CoG to bow: <> Positive X-axis	Negative	X-axis						
CoG to side: Positive Y-axis								
	Vivegauve	T GAIS						
Is the body aligned with the C	artesian axis	?						
🔶 Yes 🔷 No								
-Body reference length								
Automatic (= LOA)	<	User-defined						
 Initial free surface position— Automatic (based on body ma 	ss)	User-defined	Z-coordinate	90	[m]			
	,	oser denned	2 0001 an fact	, 10:0				
Body mass								
Automatic (based on initial free	e surface) 🔍	User-defined						
Center of gravity								
Automatic (based on initial free	e surface) 🔇	User-defined	Specify z	CoG				
-Adjust pitch for hydrostatic ed	uilibrium?							
Trim: \diamond Frozen \diamond Free								
-Body motion(s) to solve durin	g simulation							
Trim Sinkage Surge					V			
			Cancel	<< Back	Next >>			

Click on Next>>.

C. Set up project: Flow definition

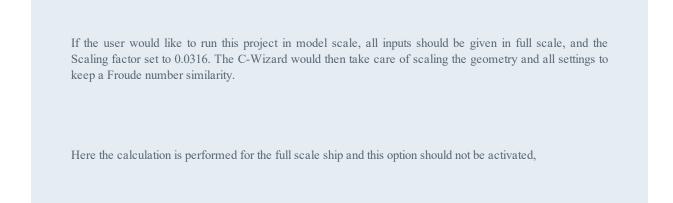
2.3.1 In the section **Speed definition**: activate **Resistance curve**. Define $V_{min} = 12kt$, $V_{max} = 24kt$, **Speed Increment** = 6kt and press *Enter*.

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

C-Wizard									
PROJECT BODY FLOW ADDITIONAL MESH MANAGEMENT CONFIGURATION DEFINITION INPUTS SET-UP									
Speed definition (positive value(s))									
Single speed									
Resistance curve									
Vmin 12 [kt] Vmax 24.0 [kt]									
Speed increment 6 [kt]									
Number of computations: 3									
List of speeds 12.0 18.0 24.0	[kt]								
Successive restarts									
-Scale input data (Froude number similarity)									
Fluid properties									
Water Air									
	Pa-s]								
Density 1026.021 [kg/m3] Density 1.2 [kg/m3]	(g/m3]								
Water properties database									
Shallow water (positive value)									
☐ Activate									
Cancel << Back N	lext >>								

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).



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.

Fluid properties					
Water			Air		
Dynamic viscosity Density	0.00122 1026.021	[Pa-s] [kg/m3]	Dynamic viscosity Density	1.85 0 -05 1.2	[Pa-s] [kg/m3]
Water properti	es database				

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:

C-Wizard	× 🗆 4
PROJECT BODY FLOW ADDITIONAL MESH MANAGEMENT CONFIGURATION DEFINITION INPUTS SET-UP	
Actuator disk	
External force	
Adaptive grid refinement (AGR) on free surface during the simulation-	
Activate	
♦ Initial base refinement + AGR ♦ AGR only	
Wall roughness	
Cartivate	
Type of coating Anti-fouling coating Value descrip	tion
Convergence	
Convergence checker	
Computation resources	
Activate Number of cores per computation 8	
Cancel << Back Ne	ext >>

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, Yand Z,

- Set the normal to <-1.0 0 0> [m] for Dir_X, Dir_Yand 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.

Actuator disk							
Activate							
Geometric definition							
Thickness 1.5 [m] Ir	nner radius	0.5	[m]	Outer radius	2.5	[m]	
Center coordinates							
× 5.11 [m]	Y	0.0	[m]	Z	4.0	[m]	
Shaft direction (pointed to the	wake)						
Dir_X -1.0	Dir_Y	0.0	[Dir_Z	0.0		
Force definition							
Thrust 100.5 [N]							
Activate tangential force							
C Activate body self update	Frequency	5	Time	steps			
🔶 Body drag	🔷 Open v	vater data					

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

Adaptive grid refinement (AGR) on free surface during the simulation							
Activate							
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.

-Wall roughness	
- Activate	
Type of coating Anti-fouling coating	Value description

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.

Convergence		
Convergence checker	Convergence booster	

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

-Computation resources-		
Activate	Number of cores per computation	8

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 HEXPRESSTM 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.

CAD Manipulation	Project Internal Surface Gro View STL Tools Plagins
Create C (cit) Create C (cit) Create C (cit) Create C (cit) Create C (cit) Create C (cit) Create C (cit) C (cit)	Conservation ander 2 3

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 <u>refinement</u> 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_l 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.

17

This option is exactly the same as for the HEXPRESSTM (**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.

C-Wizard 🕈 🗆 🗙
PROJECT BODY FLOW ADDITIONAL MESH MANAGEMENT CONFIGURATION DEFINITION ADDITIONAL MESH SET-UP
Mesh density Extra refinement of wave field
♦ Coarse ♦ Medium ♦ Fine ♦ Yes ♦ No
Merge faces with the same name? — Merge tangential faces? —
♦ Yes ♦ No
Overset
Place body in overset domain
Advanced <<<
Simplify sharp angles
Activate
Domain size
Automatic
Triangulation density
🛇 Coarse 🛇 Medium 🔷 Fine
Y+ value
♦ Automatic ♦ User-defined ♦ Y+ 220.0
Refinement dictionary file (*.csv)
Oefault (installation directory)
Cancel << Back Start mesh set-up

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

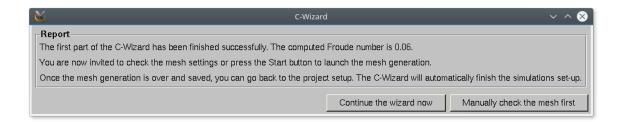
When not active, the default domain is set to $5Loa \ x \ 2Loa \ x \ 2Loa$. This domain size is recommended for resistance computations with Froude number under 0.5. For higher Froude numbers the downstream boundary will be placed further from the hull to accommodate the longer wake.

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.

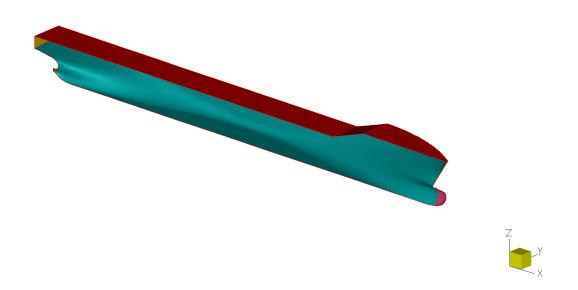
An information window reporting that the first part of the C-wizard has been successfully finished will appear. It also gives information about the computed Froude number and suggests the following actions. Click Manually check the mesh first to activate the HEXPRESSTM interface without immediately starting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

× 🖸	Во	undary conditions set	tings	$\odot \odot \otimes$
Group Ungrou	p Set face	type: MIR 🖃	Set name: ymin_SM	М
Name: *	Type:	Domain: * <u>±</u>		
Shaft_end_b1	SOL	KCS_hull_SVA_cf		
🖌 Deck1_b1	SOL	KCS_hull_SVA_cf		
🖌 Deck2_b1	SOL	KCS_hull_SVA_cf		
✓ Deck3_b1	SOL	KCS_hull_SVA_cf		
✓ ymin_SVM	MIB	KCS_hull_SVA_cf		
🛃 zmax	EXT	KCS_hull_SVA_cf		
🗹 xmini	EXT	KCS_hull_SVA_cf		
🛃 ymax	EXT	KCS_hull_SVA_cf		
🗹 zmin	EXT	KCS_hull_SVA_cf		
🗹 xmax	EXT	KCS_hull_SVA_cf		
🖌 Shaft_b1	SOL	KCS_hull_SVA_cf		
Hull_b1	SOL	KCS_hull_SVA_cf		
🖌 Transom_b1	SOL	KCS_hull_SVA_cf		
Bow_b1	SOL	KCS_hull_SVA_cf		
			Select 💻	Close

• The mesh setup is done automatically in accordance to the previously imposed settings (Medium density mesh). The parameters can be checked through the HEXPRESSTM 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

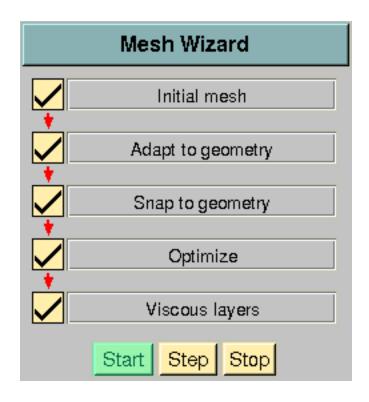
1.3.2 Mesh Generation

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

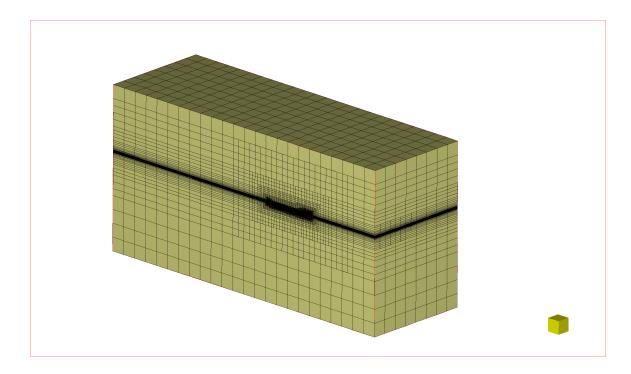
To activate it, in Menu bar of HEXPRESSTM: Project > Preference > Mesh generation > Multithreading

Please refer to the <u>HEXPRESSTM</u> 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 FINETM/Marine GUI.



2.3 Click on *Yes* to save the generated mesh.Click <u>here</u> 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.

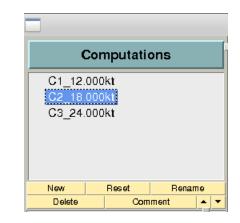
This part of the **C-Wizard** will provide the **Estimation of hydrostatic values** for the present study: the **Displacement** and **Coordinates of the Center of Gravity**. This part refers to the *domhydro* calculations (see the FINETM/Marine user guide for more explanations on the *domhydro* tool).

	C-Wizard	+ = ×
Estimation of the hydr	ostatic parameters	
Displacement: 4.26311E	E+07 [kg]	
Coordinates of the center	er of gravity:	
X: 1.13342E+02 [m]	Y: 0.00000E+00 [m]	Z: 1.11513E+01 [m]
		Naut
		INEXT

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 FINETM/Marine interface.

Time configuration:

Steady

Fluid model:

Salt water at 15°C/Air

Flow model:

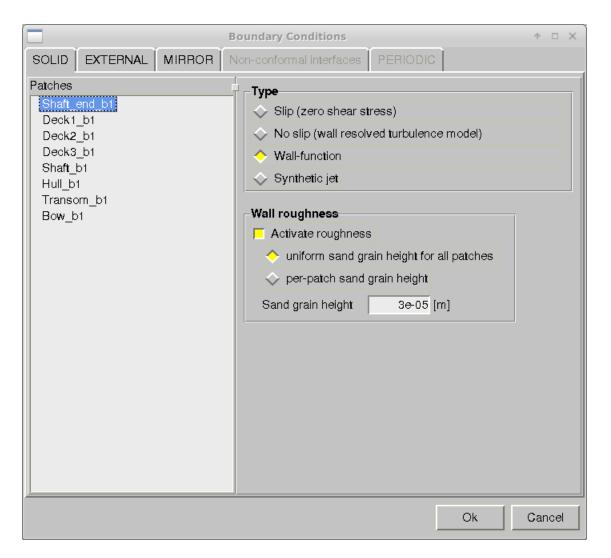
Reference length: 243.844m

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 ¹/₂ 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.

General Options Activate Cardan CActivate Quasi-8	0	ch 🔷	Hull mode 🔷 Foil mod	le			Motion Law fo		+ = :
Bodies Vessel	Motion definition Cardan angles Yaw : Rz0 = D.O.F motion d	o [F		0 [Rad]	Roll : Rx2 =	0 [Rad]	-Motion law parameter		
	D.o.fs Tx0 (Surge) Ty0 (Sway) Tz0 (Heave) Rx2 (Roll) Ry1 (Pitch) Rz0 (Yaw)	Fixed Solved Solved	Motion law		QS parameters Edit		Vo dration (dt0) Initial velocity (V0) Ramp duration (dt1) Final velocity (V1)		[s] [m/s] [s]
				F Show/	Hide Ok	Cancel		Ok C	Cance

Dynamic parameters:

X	Body and frame body motion	\sim \sim \otimes
General Options		
🗖 Activate Cardan a	ingle	
📕 Activate Quasi-Sta	atic (QS) approach 🛛 🔷 Hull mode 🔷 Foil mode	
Bodies Vessel	Motion definition Dynamic parameters	
	Inertial data Coupling parameters External forces Initial conditions	
	Geometry Half body Entire body	
	Estimate inertial data	
	Center of gravity X_CG= 113.3424 [m] Y_CG= 0 [m] Z_CG= 11.15126 [m]	
	Mass and Inertia (entire body) Mass: 41493630 [kg]	
	Inertia matrix in body primary frame	
	[A - F - E] A = 699994100 [kg·m ²] D = 0 [kg·m ²]	
	[-F B -D] B = 192735800 [kg·m ²] E = 235894300 [kg·m ²]	
	$[-E - D C]$ $C = 195092500$ $[kg \cdot m^2]$ $F = 0$ $[kg \cdot m^2]$	
1	Show/Hide Ok	Cancel

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

X Adaptive C	Grid Refinement V 🔿 😣
Activate grid refinement	
Criterion Grid quality Box Control	
Criterion Refinement criterion type	Free surface (tensor)
Similar to the Free surface criterion, but generates less refinement around foam or breaking waves. To be prefered for unsteady flow.	
Refinement target values	
Target number of cells	500000
Target grid spacing normal to free surface Minimum size limit for refined cells	0.3169972 [m] 0 [m]
	Ok Cancel

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:

	Co	ntrol Variables		(+ = ×
Time step	Convergence	Expert parameter	s	
Time step	parameters —			
Number of	time steps	1200 🚔		
Time step	law	UNIFORM	Ŧ	
🔲 Activat	te sub-cycling ac	celeration		
Time step	value	0.1316658 [s]		
		1		(
Reset tim	ie law parameters	3	Ok	Cancel

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.

		Contro	l Variabl	es			↑ □ ×
Time step Convergence Expert parameters							
General parameters							
Maximum number of non-linear iterations 5							
Convergence	Convergence criteria 2 orders						
Save solution every					100 🚔	time ste	ps
Convergence	Convergence parameters						
Activate convergence booster							
Activate	convergenc	e checł	ker				
Stability inte	rval	30.0	%				
Average las	t 🔽	10.0	%				
Tolerance		1.0	%				
Check after	k after 200			time steps			
Check quantities:							
□ Tx □ 1	Ty 🗖 Tz	🗌 Vx	🔲 Vy	🔲 Vz	🔲 Ax	🗌 Ау	🔲 Az
Fx 🗌	Fy 🔲 Fz	∐ Mx	🗌 Му	🔲 Mz	🔲 Rx	🗖 Ry	🔲 Rz
					Ok		Cancel

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

× 0	Outpu	t Parameters			\odot \odot \otimes	
Motion & force variables Probe variab		Optional outp	Mean flow	variables		
Translation variables ■ Tx0 ■ Ty0 ■ Tz0		otation variabl	🔷 Quat	ernion I Rz0		
Velocity variables		ngular velocity ⊐ dRx2 <mark>□</mark>		⊔ dRz0		
Acceleration variables Ax0 Ay0 Az0		ngular acceler ⊒ d2Rx2	ation variab d2Ry1]	
Force decomposition Global frame Sody frame						
				Ok _	Cancel	

1.4.2 Launch & Control Computation

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

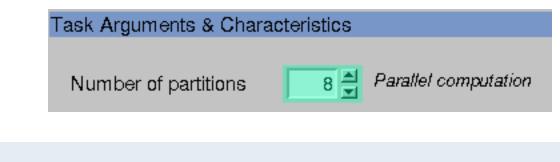
2.2. Go to the **Task Manager** by clicking on the icon

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:

	Task Manager	File					
	🗁 Task Manager	Go back to project set-up					
	Tasks definition	Task List					
		Task Name Task Sta	atus Task delay				
		test_C1_12.000kt.sim pending	none				
		test_C2_18.000kt.sim pending	of after test_C1_12.000kt.sim				
		test_C3_24.000kt.sim pending	of after test_C2_18.000kt.sim				
		New Task Remove Task Rename Task Move Up	Move Down Save Batch File Start Stop Delay				
		Task Arguments & Characteristics					
		Number of partitions 8 A Parallel computation	∐ Use Fuliflex license key				
		Simulation file (.sim) /marketing/home/anna/Tasks/geom/test/test_C	C1_12.000kt/test_C1_12.000kt.sim				
		MPI library Intel MPI					

2.4 The number of partitions is automatically set to the earlier specified value of $\langle 8 \rangle$, or the maximum number of cores available in the machine. Change this value if needed.

34



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.

New Task R	emove Task	Rename Task	MoveUp	Move Down	Save Batch File	Start	Stop	Delay

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.

TASK MANAGER INFO	×
COMPUTATION INFO Project path: /marketing/home/clero/Tutorials/C-Wizard_Resistance Computation name: C-Wizard_Resistance_computation_3.0ms	
RUN INFO:	
Pre-processing: finished Solver: finished Post-processing: finished	

2.6. While the computations are running the **Monitor** \bowtie 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.

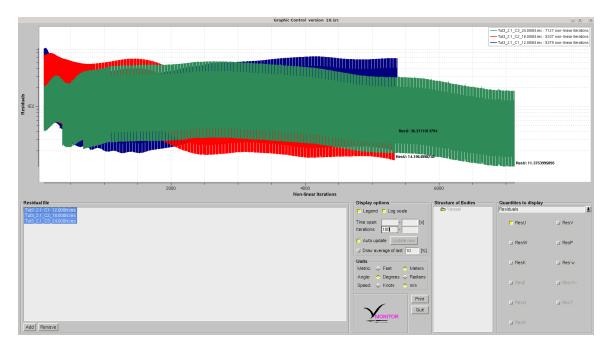


FIGURE 1.1 Residuals evolution

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.

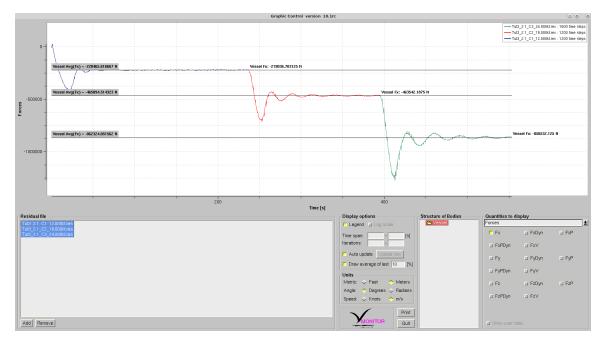


FIGURE 1.2 Drag evolution and averaged value on the last 10% of each computation

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.

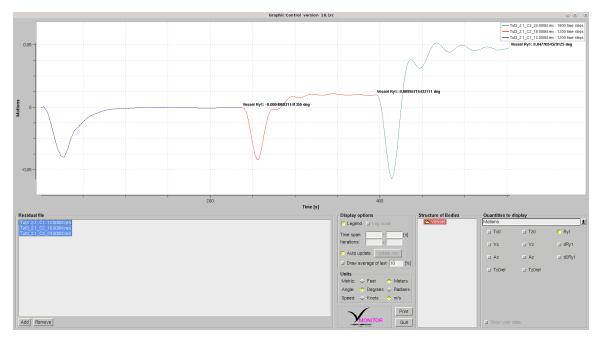


FIGURE 1.3 Trim angle (Ry) evolution in the three computations

The angle units can be switched beteen radians or degrees.

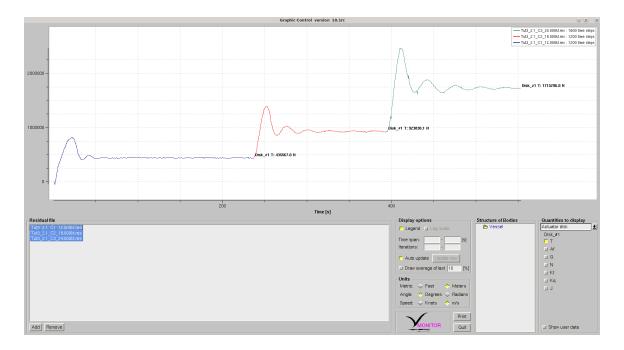


FIGURE 1.4 Actuator disk thrust

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 .

	Results analysis		\odot	\otimes
Computations list				
	C1_12.00 C2_18.00 C3_24.00 Select)Okt		
Application selection Specific Resistance curve Perform resistance curve (a	quantities Analysis op		options e number))
Self-propulsion analysis				
Open water performance curve				
		Close	Perfor	m

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	\odot \odot	\otimes
Computations list		
C1_12.000kt		1
C2_18.000kt		
C3_24.000kt		
Select >		
▲ Remove		
Application selection Specific quantities Analysis options	Plot option:	s
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.

	Results	s analysis	\odot \odot \otimes
Computations list —			
		C1_12.000kt C2_18.000kt C3_24.000kt	
Application selection	Specific quantitie	s Analysis options	Plot options
Criterion for average Average efforts Average motions Average values over t Draw average line Draw convergence	he last 10 [%		erion 1 [%]
Filter for efforts	Window width in Window width in		
Filter for motions — Median filter Moving average	Window width in Window width in		
			Close Perform

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**.

Results analysis			\odot	\otimes	
Computations list					
	Select. Remove	C1_12.00 C2_18.00 C3_24.00	Okt		
Application selection Spe	cific quantities	Analysis o	ptions	Plot optio	ns
Ymin Yn Yn Picture size	ntities on the sam putations on the	me plot			
Units Metric: ◇ Feet ◇ N Angle: ◇ Degrees ◇ R Speed: ◇ Knots ◇ m Dimensionless coefficient □ Activate	adians n/s				
			Close	Perfor	m

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.

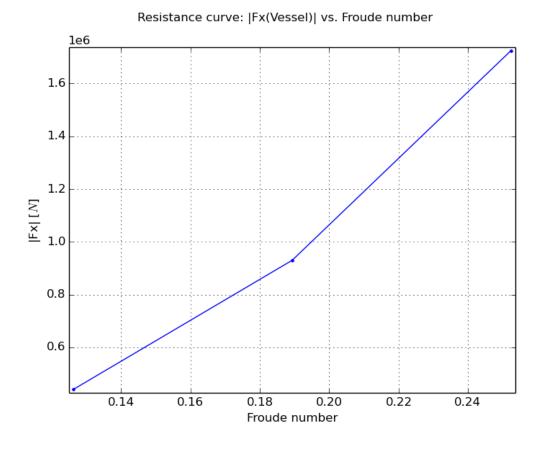


FIGURE 1.5 Drag vs Froude number plot.

Resistance curve: |Ry1(Vessel)| vs. Froude number

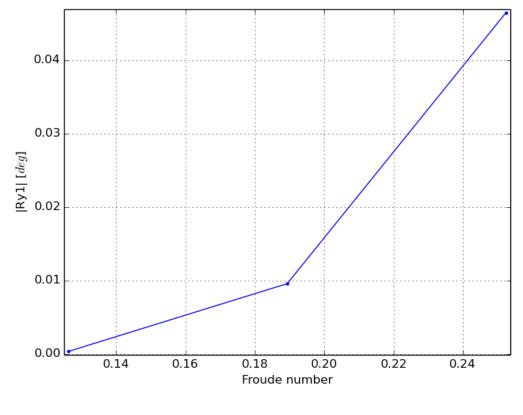


FIGURE 1.6 Trim angle vs. Froude number plot.

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.

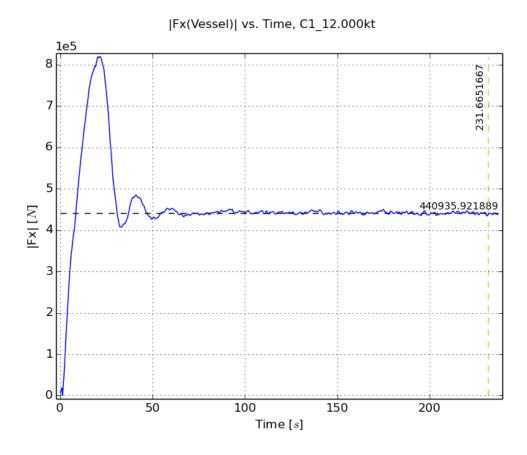


FIGURE 1.7

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).

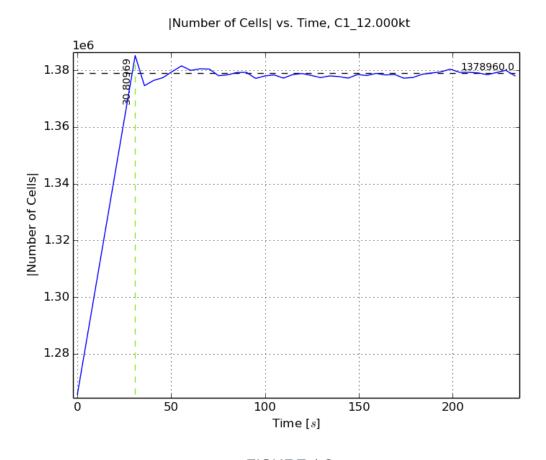


FIGURE 1.8 Evolution of the number of cells in the mesh due to Adaptive grid refinement in the 12kt computation.

B. CFView[™]

3.2.1. To start the Post-processing, click on the CFView icon in of the FINETM/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.

		FINE/Ma	rine		\odot	\otimes
	Sele	ct result to o	pen in CF	View		
Traveling shot Traveling shot ba	sed on Ves	ssel	<u>±</u>			
Mesh reference	ce frame 💠 E	lody primary frame				
Tx0	🗆 Туо	🗖 Tzo	🔲 Rx2	🖾 Ry1	🗖 Rzo	
				Ok	Canc	el

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%.

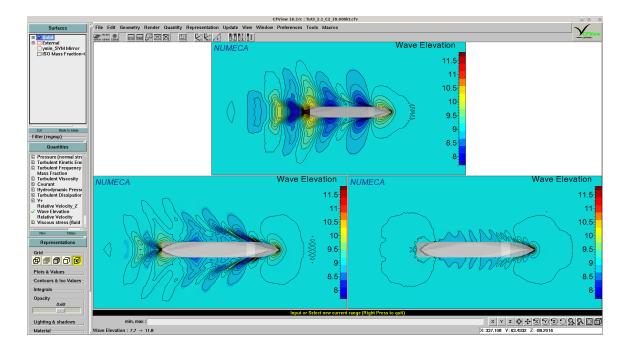
Representations
Grid
Plots & Values
Contours & Iso Values
Integrals
Opacity
Lighting & shadows Material

Surface Mate 🕑 🔿 🛛 🙁
Diffuse
Color
Ed.
-Specular Highlight-
Opacity
0.70
Deflection
Reflection

3.2.8 Free surface representation:

- Use Macros > Represent free surface
- Use the Set range button it 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.

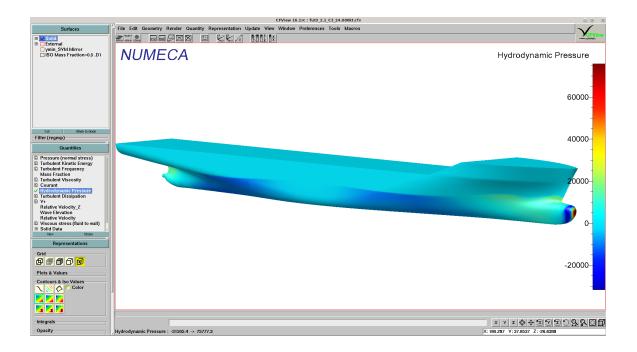
min, max : 7.7 11.8

	Hardcopy Output Set Up	\odot	×
-Formats	♦ Active View		
🔶 PNG	🔷 Graphics Window		
◇ PostScript	🗆 Banner		
Others	Banner Text and Settings		
	Date in Banner		
	Type : times		
	Size : 0 🚆 (0->default)		
	☐ Frame		
	Options		
	Page Layout		
	🔷 scaling (%) : 100 🚆		
	🔷 size (pixels) : 1920 🚔 x 108	0	
	Quality : High	Ŧ	
Ok	Reset	ose	

• 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 **Solution** 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.

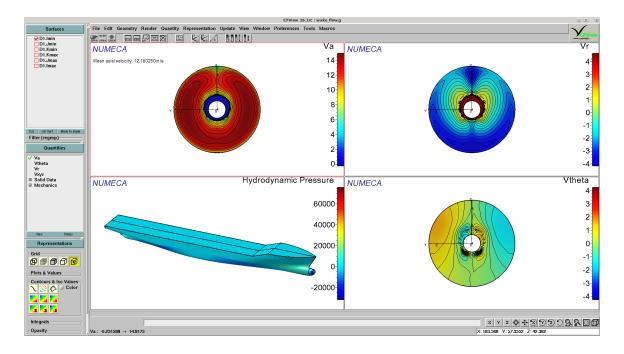
- Use or **Window** > **Close** to close all views except the 24kt result.
- Select the Macros > Wake_flow_tool.

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.

Image: Synth Mirry	Surfaces	File Edit Geometry Render Quantity Dependent View Window Preferences Tools Macros	
Actualor disk parameters Countifies Countriles Coun	External ymin SYM Mirror		
Lighting & shadows Americal American Americ American American Am American American A	Filter (regop) Cuantitie Cuantitie Pressure (normal stress) Turbulent Kinetic Energy Mark Streation Corrant Doronthic Pressure Ournathic Pressure Variation Pressure Representations Grid Grid	Actuator disk parameters Countercidowise positive Counterdidowise positive Connercidowise positive	0- -5- -10- -15-
… No quantity selected 次100-297 11:27.85.37 2:-26.5399	<u></u>		

- 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.



cādence[®]

10.2

© NUMECA International, all rights reserved EN202208311820