



JoRes1 Tanker

Aim

The aim of this test case is to obtain insight into the capability of CFD to predict full-scale trial powering performance. The target trial speed is 13.34 kn ($F_r=0.164$). The package of this test case is designed such that direct full-scale performance predictions with CFD tools can be compared and assessed. Validation metrics are the propeller revolution rate and delivered power to the propeller.

Description

This test case originates from the JoRes Joint Industry Project (<https://jores.net/>). The hull form features a (mild) bulbous bow, a single propeller with a propeller boss cap fin (PBCF), a horn rudder and an asymmetric centreline skeg. This tanker was the subject of full-scale trials, during which the powering performance was measured. Furthermore, a PIV measurement system was installed on board to measure part of the flow coming into the propeller.

The vessel was in dry dock in early 2022 where the exact geometry was obtained using a 3D laser scanner and hull and propeller roughness measurements were performed. The scanned geometry was cleaned to remove imperfections and this cleaned version of the geometry is provided for the workshop. The PIV unit (designated FlowPike) was also installed in the dry dock in the area above the propeller. Comprehensive sea trials on the JoRes1 tanker were completed after the dry docking. The collected data is aimed to be used for ship-scale CFD validation.

The ship geometry including appendages can be found on JoRes1 Tanker case on the W2025 website under <https://w2025.nl/instructions/jores1-tanker-full-scale-powering-performance>. Additionally, this page also provides a questionnaire that needs to be filled in. Guidelines for the submission of the output from the CFD predictions are given in these instructions and under <https://w2025.nl/instructions>.

The JoRes1 Tanker has been extensively studied within the JoRes JIP and these instructions are based on the instructions provided to the JIP participants, written by Peter Horn, Dmitriy Ponkratov, Themis Melissaris and Serge Toxopeus. The permission to use these instructions for the Wageningen 2025 CFD Workshop is greatly appreciated.

References

- [1] Birvalski, M., Struijk, G. D. and Ponkratov, D., "Full-Scale PIV Measurements of the Propeller Inflow", 15th International Symposium on Particle Image Velocimetry (ISPIV), San Diego, CA, June 2023. [↗](#)
- [2] Birvalski, M., Toxopeus, S. L., Struijk, G. D., and Ponkratov, D., "Comparison of Full-Scale PIV Measurements with CFD Simulations of a Ship Propeller Inflow", 21st International Symposium on Applications of Laser and Imaging Techniques to Fluid Mechanics, Lisbon, Portugal, July 2024. [↗](#)



Geometry of the Hull and Appendages

Table 1: Main particulars of the ship

Characteristic		Value	Units	Notes
Length between perpendiculars	L_{pp}	178.50	m	
Length of submerged body	L_{OS}	183.20	m	
Breadth	B	32.26	m	
Draught at aft perpendicular	T_{AP}	10.38	m	
Draught at fore perpendicular	T_{FP}	10.56	m	
Draught at midship	T_M	10.47	m	$T_M = \frac{T_{AP} + T_{FP}}{2}$
Static trim angle	θ	0.0578	deg	$\theta = \arctan\left(\frac{T_{FP} - T_{AP}}{L_{pp}}\right)$
Hydrostatics of hull with rudder, FlowPike, propeller and PBCF				
Displaced volume	∇	46650	m ³	
Wetted surface area	A	8369.1	m ²	
Displaced mass	Δ	48003	t	
Longitudinal centre of buoyancy	x_{CB}	93.021	m	
Transversal centre of buoyancy	y_{CB}	0.0	m	
Vertical centre of buoyancy	z_{CB}	5.610	m	
Hydrostatics of hull with rudder, FlowPike, dummyHub				
Displaced volume	∇	46648	m ³	
Wetted surface area	A	8337.1	m ²	
Displaced mass	Δ	48001	t	
Longitudinal centre of buoyancy	x_{CB}	93.023	m	
Transversal centre of buoyancy	y_{CB}	0.0	m	
Vertical centre of buoyancy	z_{CB}	5.610	m	
Estimation of mass properties				
Longitudinal centre of gravity	x_{CG}	x_{CB}	m	Based on actual draughts
Transversal centre of gravity	y_{CG}	0.0	m	
Vertical centre of gravity	z_{CG}	10.47	m	Assumed to be T_M
Radius of gyration	k_{xx}	11.29	m	$0.35 \cdot B$
Radius of gyration	k_{yy}	44.625	m	$0.25 \cdot L_{pp}$
Radius of gyration	k_{zz}	44.625	m	$0.25 \cdot L_{pp}$

Note: draughts, CB and CG are given according to definitions usual in naval architecture, ie. in the body fixed (untrimmed) coordinate system.

The geometry of the hull and its appendages is given in the files **JoRes1Tanker.***, in IGES (*.iges), STEP (*.stp) and Rhinoceros (*.3dm) formats. In the CAD files, the hull, rudder, FlowPike, propeller, and PBCF are defined as separate entities. Also two dummy entities are included: **dummyHub**, which should replace the propeller and PBCF in the resistance predictions, and **dummyDrivePOW**, which can be used for the propeller+PBCF open water predictions.

The main particulars of the ship are given in Table 1 and those of the propeller in Table 2. The hull form comes with a rudder (fixed and movable parts are blended into one), propeller hub, propeller blades, propeller boss cap fins (PBCF) and PIV unit (FlowPike). The propeller and PBCF are fixed to each other and do not rotate with respect to another. Figure 1 shows the propeller and PBCF surfaces in the CAD file.

Table 2: Propeller information

Characteristic		Value	Units	Notes
Propeller Diameter	D_P	6.5	m	
Pitch Ratio at 0.7R	$\frac{P_{0.7R}}{D_P}$	0.7947	-	
Expanded Blade Area Ratio	$\frac{A_E}{A_0}$	0.445	-	
Hub diameter Ratio	$\frac{D_H}{D_P}$	0.15	-	
No. of blades	Z	4	-	
Propeller Position	$(x, y, z)_p$	3334.6, 0, 3489.3	mm	in ship coordinate system

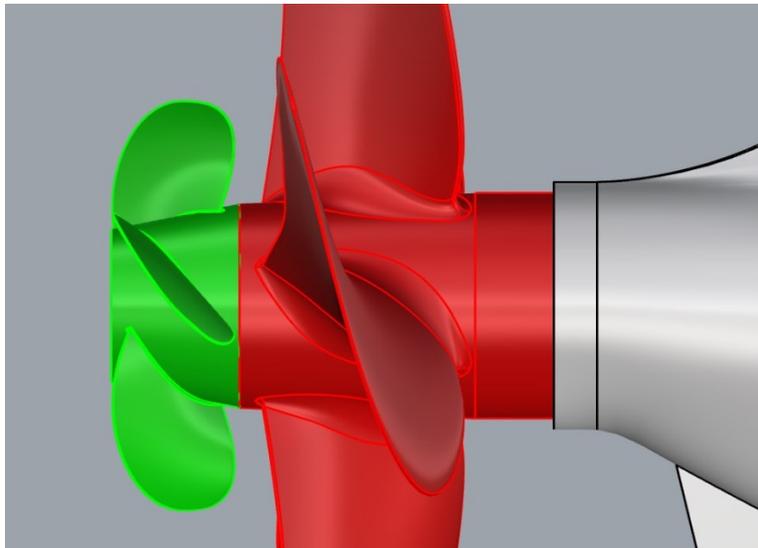


Figure 1: Propeller (red) and PBCF (green)

The geometry is provided in even keel condition. The CAD geometry global coordinate origin is defined at the aft perpendicular (x positive forward), centreline (y positive to port side) and the baseline of the vessel (z positive up): $(x, y, z)_{origin} = (AP, 0, BL) = (0.0m, 0.0m, 0.0m)$. To transform the hull form from the even keel condition into the correct floating position, the geometry is to be rotated around the y-axis with origin $(x, y, z) = (L_{pp}/2, 0, T_M) = (89.25m, 0.0m, 10.47m)$ by $\theta = 0.0578^\circ$.

The roughness of the hull, rudder and propeller and PBCF were measured and converted into an equivalent sand grain roughness resulting in the values in Table 3.

Table 3: Average measured and equivalent sand grain roughness values

Surface	Average measured roughness [μm]	Equivalent sand grain roughness [μm]
Hull	218	53
Rudder	243	63
Propeller and PBCF	5.57	3.79

The superstructure or bilge keels are not considered and not included in the CAD geometry. The influence of superstructure and bilge keels on the hull resistance is to be incorporated during the powering predictions using an additional resistance, according to Table 4. The point of application of this additional force should coincide with the centre of gravity given in Table 1.

Table 4: Summary of added air and bilge keels resistance to be used during powering prediction

Ship Speed (kn)	Air Resistance (kN)	Bilge Keels Resistance (kN)	Total Added Resistance (kN)
14.14	17.79	10.25	28.04
13.34	15.83	9.20	25.03
11.25	11.26	6.69	17.95

PIV Measurement System FlowPike

The PIV measurements were performed in the plane as illustrated by the field of view (FOV) in green in Figure 2.

The plane in which the flow measurements were taken (the FlowPike's field-of-view) can be defined as follows (all in the ship-fixed coordinate system as used in the CAD file):

Origin (intersection FlowPike axis - laser axis): $(x, y, z) = (5.128, -0.928, 8.009)$ m

The plane can be defined using two vectors 1 and 2, or by its normal vector N:

- Vector 1: $(0.008459, 0.999291, -0.036677)$ m and Vector 2: $(0.190922, 0.03439, 0.981003)$ m
- Vector N: (normal vector to plane): $(0.981569, -0.015300, -0.190495)$ m

The FlowPike is rigidly connected to the ship hull and therefore moves with the dynamic trim and sinkage.

The average velocity (3 components in ship-fixed coordinate system – V_x/V_s , V_y/V_s and V_z/V_s) should be calculated for this plane over at least 4 propeller revolutions.

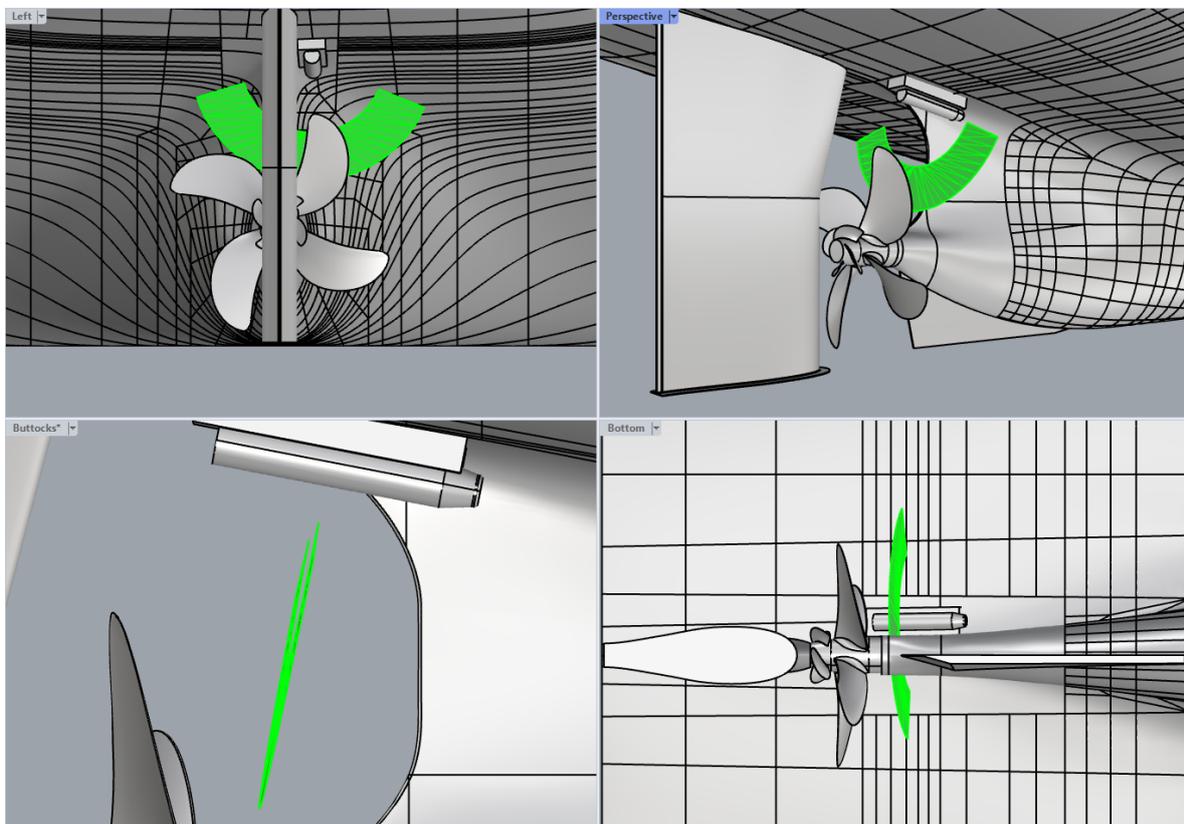


Figure 2: PIV plane location

Fluid properties

Participants should use in their numerical setup the seawater and air properties and other values defined in Table 5. Note the specification of the reference pressure at the still water surface!

Table 5: Properties of the water and air during the trials

Property		Value	Units	Notes
Water properties				
Density	ρ_w	1029.0	kg/m ³	Measured before trials
Temperature	T_w	22.8	°C	Measured before trials
Dynamic viscosity μ	μ_w	$1.23026 \cdot 10^{-3}$	Pa-s	Calculated from Temperature and Salinity
Kinematic viscosity ν	ν_w	$1.19559 \cdot 10^{-6}$	m ² /s	Calculated from Dyn. Viscosity and Density
Air properties				
Density	ρ_a	1.212	kg/m ³	Calculated from Temperature
Temperature	T_a	19.9	°C	Average from measurements during trials
Dynamic viscosity μ	μ_a	$1.8200 \cdot 10^{-5}$	Pa-s	Calculated from Temperature
Kinematic viscosity ν	ν_a	$1.5102 \cdot 10^{-5}$	m ² /s	Calculated from Temperature
Reference pressure at still water surface	p_{ref}	101325	Pa	Assumed value
Other properties				
Acceleration of gravity	g	9.8067	m/s ²	

The Froude and Reynolds numbers are based on the above quantities and defined as:

$$F_r = \frac{V_s}{\sqrt{g \cdot L_{pp}}}, \quad Re = \frac{V_s \cdot L_{pp}}{\nu_w}$$

Variable Definitions and Data Reductions

Global Quantities

Global quantities are to be given in CSV files (https://en.wikipedia.org/wiki/Comma-separated_values), with values separated by commas. The first line of the CSV file should contain a header, indicating the variables listed in each column. The variable names should comply to the datafile names specified hereunder. **The second line of the file should contain the units of the variables in each column.** Values need to have at least 6 significant digits.

Ship Quantities

See also Table 1.

	Symbol	Unit	Data reduction	Datafile name
Ship velocity	V_s	m/s	V_s	Vs
Dynamic sinkage	σ	m	$T_M(V_s > 0) - T_M(V_s = 0)$	sink
Dynamic trim	τ	deg	$\theta(V_s > 0) - \theta(V_s = 0)$	trim

Trim is positive for $T_{FP} > T_{AP}$ (trimmed by the bow). Dynamic sinkage is the increase of the draught at midship and is positive down ($T_M(V>0) > T_M(V=0)$). The values are to be given relative to the neutrally buoyant condition specified in Table 1.

Propeller Quantities

V_A is the flow velocity magnitude in m/s, n is the propeller rotation rate in rev/s and D_p the propeller diameter, see also Table 2. T and Q are the thrust and torque acting on the propeller, the PBCF, or both together.

	Symbol	Unit	Data reduction	Datafile name
Advance velocity	V_A	m/s	V_A	Va
Propeller rate	n	rev/s	n	n
Advance ratio	J	-	$J = \frac{V_A}{nD_p}$	J
Thrust coefficient	K_T	-	$K_T = \frac{T}{\rho_w n^2 D_p^4}$	KTp (for propeller) KTpbcf (for PBCF) KT (for prop+PBCF)
Torque coefficient	K_Q	-	$K_Q = \frac{Q}{\rho_w n^2 D_p^5}$	KQp (for propeller) KQpbcf (for PBCF) KQ (for prop+PBCF)

Note that the propeller thrust and torque are ship-fixed quantities and are assumed to be positive. **The advance velocity V_A should be used for non-dimensionalisation of pressure and shear stress.**

Force Quantities

Although the surface pressure and friction fields will be post-processed by the organisers to obtain global forces, participants are also asked to provide the forces according to their solver or data processing. Earth-fixed forces are to be provided (F_x positive towards the bow, F_z positive upward), according to the following table.

	Symbol	Unit	Data reduction	Datafile name
Friction force	F_f	N	-	Ff = [Ffx, Ffy, Ffz]
Pressure force	F_p	N	-	Fp = [Fpx, Fpy, Fpz]

The forces are to be given for each surface defined in the CAD file, by appending the surface name to the datafile name. For instance, the friction force on the rudder should be given as **Ffrudder** = [Ffrudderx, Ffruddery, Ffrudderz].

The pressure forces acting on non-closed surfaces need to be corrected for missing hydrostatic contributions which normally would act on the parts that would close that surface. This means that at zero

speed, the forces F_{p0} on each surface need to be extracted. For the requested conditions, the corrected pressure force values should then be: $F_p = F_{pnon-closed} - F_{p0}$.

Other Quantities

Some additional optional calculations are suggested below. For these, additional variables may need to be given, according to the following table.

	Symbol	Unit	Data reduction	Datafile name
Cells in the grid	n_{Cells}	-	-	nCells
Faces on the ship surface	n_{Faces}	-	-	nFaces
Time step size	Δt	s	-	deltaT

The number of faces should contain all ship surfaces that are present in the calculation.

Local Quantities

Local quantities (field data) are to be given in files that can be imported into Paraview (e.g. CGNS (<https://cgns.github.io/>), native Paraview VT* files, Enight Gold *.case (seems to be best option for output from StarCCM+), or Tecplot (*.dat ASCII format preferred to improve compatibility)). Participants may be asked to check compatibility and convert files if needed. Do not include more variables than asked for, to limit file sizes and processing times.

The variable names should comply to the datafile names specified hereunder. Vector values may either be given as a vector (e.g. **cf**), or by providing their individual components (e.g. **cfx**, **cfy**, **cfz**). In all files, grid coordinates (using the geometry CAD file system of axes) need to be included, with the appropriate naming conventions for the given format (e.g. **CoordinateX**, **CoordinateY**, **CoordinateZ** for CGNS, or **coords** in Paraview). Coordinates should be specified in full scale dimensional values, in metres.

Local quantities can be given as Cell or Point values (please do not include both, to limit file sizes). If needed, Cell values will be converted to Point values in the post-processing by the organizers.

Surface Quantities

For local quantities, the ship length L_{pp} and speed V_s are used for non-dimensionalisation. For the density ρ , the local density of the fluid should be used (ρ_w in water and ρ_a in air). The surfaces defined in the CAD files need to be retained in the surface field data.

	Symbol	Unit	Data reduction	Datafile name
Pressure (incl hydrostatic comp)	c_p	-	$c_p = \frac{p - p_{ref}}{\frac{1}{2} \rho V_s^2}$	cp
Friction	c_f	-	$c_f = \frac{\tau}{\frac{1}{2} \rho V_s^2}$	cf = [cfx, cfy, cfz]
AirVolumeFraction	α	-	α	vof
Local fluid density	ρ	kg/m ³	$\rho = \rho_w(1 - \alpha) + \rho_a \alpha$	rho
Cell blanking	-	-	-	iBlank

For the propeller open water results, use the advance velocity V_A instead of the ship velocity V_s .

Positive directions for friction coefficient c_f components are positive towards the bow for x and upward for z .

In Volume-of-Fluid methods, the AirVolumeFraction defines the ratio of volume of air to the volume of a cell and is 1 in air, and 0 in water. Generally, **vof=0.5** is considered to indicate the interface between air and water. If other methods of describing the interface are used (e.g. Level set), **vof** should be reconstructed in order to identify whether a cell is in air (**vof=1**) or in water (**vof=0**).

Overset solutions may need to include `iBlank`, with `iBlank=0`: hole, `iBlank=1`: in-cell, and in-between values can be used to weigh overlapping cells if needed. `iBlank` is an optional variable in the output. If it is not included, it is assumed that `iBlank=1`.

Time-averaged surface data is preferred. However, if the solver is unable to average data on moving objects (such as the propeller and PBCF), then phased-average data is to be provided, for the propeller in the top position (original CAD orientation of the propeller). If this is also not possible, the participant is required to provide an instantaneous solution with the propeller in the top position, and additionally provide a Csv file with the time traces of the integrated pressure and friction forces on each surface. The layout of the Csv file should be the same as the requested Csv file with time-averaged data, but with the time (datafile name `time`, in seconds) as additional column.

PIV Plane Quantities

	Symbol	Unit	Data reduction	Datafile name
Velocity	V'	-	$V' = \frac{V}{V_s}$	<code>V = [u, v, w]</code>
Vorticity	ω'	-	$\omega' = \frac{\omega \cdot L_{pp}}{V_s}$	<code>omega = [omegax, omegay, omegaz]</code>
Turbulence kinetic energy	k'	-	$k' = \frac{k}{V_s^2}$	<code>tke</code>
AirVolumeFraction	α	-	α	<code>vof</code>
Cell blanking	-	-	-	<code>iBlank</code>

Turbulence kinetic energy should contain both resolved and modelled k .

Participants are requested to extract the full extents of the plane defined above from their CFD solution, i.e. do not restrict the provided flow field to the green area shown in Figure 2. This will enable better and more detailed comparisons between the CFD solutions.

Note that the PIV plane is attached to the ship and will therefore move with the initial and dynamic trim/sinkage of the ship. The coordinates and velocity components for the PIV plane results are therefore ship-fixed. Positive directions for velocity and vorticity components are positive towards the bow for x and upward for z .

Submission of data

Data files should be collected in one zip file per submission and provided to the organizers according to the instructions on the W2025 website. According to the instructions below, specific filenames are requested. In your submission, `<INSTITUTION>` should be replaced with the appropriate (abbreviated if it is too long) name of your institution. If optional cases are submitted, or multiple calculations are done for the same case, names of the datafiles need to be appended with a unique identifier, separated from the rest of the file name with a hyphen/minus (-), e.g. an optional contribution of MARIN for the propeller open water case, where roughness is not taken into account, could be named `JoRes1-MARIN-0-not_rough.csv`.

Definition of the Package of Cases

The following package with mandatory cases is proposed. To participate in the workshop, all cases have to be submitted.

Table 6: Full scale propulsion package (mandatory)

Case	Title	Metrics
1 (O)	Propeller + PBCF open water	K_T and K_Q for range of J values
2 (R)	Resistance	F_f and F_p , trim and sinkage, wave pattern at V=8.13, 11.25, 13.34, 14.14 and 16.27 kn
3 (P)	Propulsion incl added resistance from bilge keels and superstructure	Total longitudinal force, K_T , K_Q , n , trim and sinkage, wave pattern, flow field at PIV plane at V=13.34 kn. Validation metrics will be K_Q , n and the velocities in the PIV plane.

If a participant only can do propulsion predictions using body forces and for that the open water curves are used as input, the participant may contact the organizers for the open water data.

Case 1 Propeller + PBCF open water (O)

Conditions

- The propeller should be fitted to a shaft with a diameter of the propeller hub diameter (**dummyDrivePOW** in CAD file). The shaft should extend fully towards the inflow boundary, see Figure 3. The **dummyDrivePOW** entity should not be taken into account in the evaluation of the forces on the propeller and PBCF. The forces acting on the propeller and on the PBCF should be extracted separately.
- Fixed propeller revolutions should be used, and the velocity is to be adjusted to obtain the right advance ratio J . The propeller revolutions n should be set to 1.6 rev/s.
- Gravity, free surface, side walls or sea floor should not be taken into account.
- Roughness according to Table 3 should be applied in the CFD calculation. No roughness is to be applied to the **dummyDrivePOW** surface.
- Note: the grid around the propeller and PBCF should be of similar density and topology as the one around the propeller for the propulsion case.
- J values to be calculated:

J	0.1	0.3	0.5	0.6	0.7	0.8
---	-----	-----	-----	-----	-----	-----

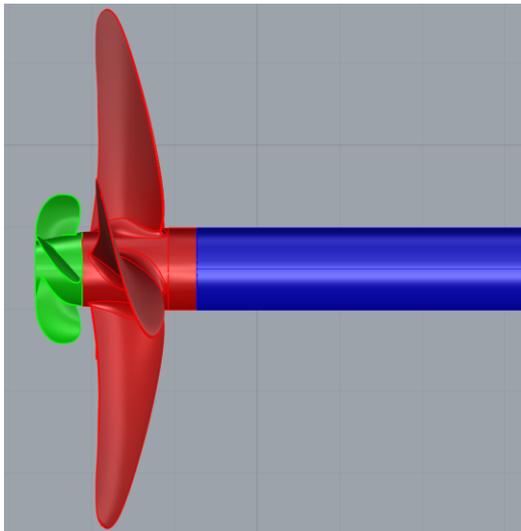


Figure 3: Proposed propeller open water shaft setup (flow from right to left)

Requested results

- JoRes1-<INSTITUTION>-0.csv: Csv file with for each J value a row containing:
 $J, n, Va, KTp, KTpbcf, KT, KQp, KQpbcf, KQ$
- JoRes1-<INSTITUTION>-0-s-J<Jvalue>.1f>.*: for each J value a separate flow data file should be provided, with (if needed: time-averaged, for the propeller in the original CAD position) variables:
 cp, cfx, cfy, cfz

Optional calculation(s)

Participants are encouraged to do additional studies, comprising, but not limited to:

- Same as above, but without roughness.

Case 2 Resistance (R)

Conditions

- With rudder
- Propeller and PBCF replaced by **dummyHub** from CAD file, see Figure 4.
- Calm water condition
- Free trim and sinkage: FR_{z0}
- Roughness according to Table 3 should be applied in the CFD calculation.
- Five speeds:

No.	1	2	3	4	5
Speed [kn]	8.133	11.25	13.34	14.14	16.266
Speed [m/s] (V_s)	4.184	5.787	6.863	7.274	8.368
Froude number (F_r)	0.100	0.138	0.164	0.174	0.200
Reynolds number (Re)	625×10^6	864×10^6	1025×10^6	1086×10^6	1249×10^6

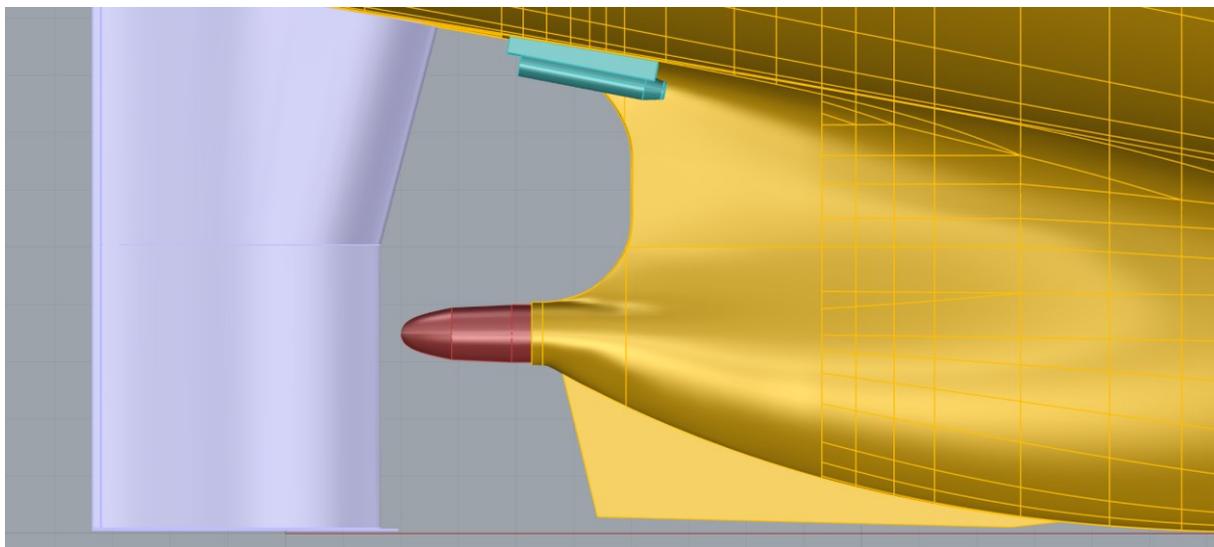


Figure 4: Proposed resistance setup

Requested results

- **JoRes1-<INSTITUTION>-R.csv**: Csv file with for each F_r value a row containing the (time-averaged) variables:
 F_r , V_s , $sink$, $trim$, $F_f<surface>$, $F_p<surface>$
 F_f and F_p should be given for each <surface> being surfaces **hull**, **rudder**, **FlowPike** and **dummyHub**, for x, y, and z directions.
- **JoRes1-<INSTITUTION>-R-s-Fr<Frvalue:%04.2f>.***: for each F_r value a separate surface field data file should be provided, with (time-averaged) variables:
 vof , ρ , cp , cf_x , cf_y , cf_z , $iBlank$
- **JoRes1-<INSTITUTION>-R-piv-Fr<Frvalue:%04.2f>.***: for each F_r value a separate PIV plane data file should be provided, with (time-averaged) variables:
 vof , u , v , w , ω_x , ω_y , ω_z , tke , $iBlank$
- **JoRes1-<INSTITUTION>-R-fs-Fr<Frvalue:%04.2f>.***: for each F_r value a separate data file with the (time-averaged) free surface (iso-surface of $vof=0.5$) should be provided. No additional variables are to be included.

Optional calculation(s)

Participants are encouraged to do additional studies, comprising, but not limited to:

- Same as above, but without roughness.

Case 3 Propulsion (P)

Conditions

- With rudder
- With Propeller and PBCF
- Propeller revolution rate to be adjusted to attain self-propulsion, when adding bilge keel and superstructure resistance according to Table 4.
- Calm water condition
- Free trim and sinkage: FR_{z0}
- Roughness according to Table 3 should be applied in the CFD calculation.
- One speed:

No.	1
Speed [kn]	13.34
Speed [m/s] (V_s)	6.863
Froude number (F_r)	0.164
Reynolds number (Re)	1025×10^6

Requested results

- JoRes1-<INSTITUTION>-P.csv: Csv file with for each F_r value a row containing the (time-averaged) variables: Fr , V_s , sink, trim, n, K_{Tp} , K_{Tpbcf} , K_T , K_{Qp} , K_{Qpbcf} , K_Q , $F_f<surface>$, $F_p<surface>$
 F_f and F_p should be given for each <surface> being surfaces hull, rudder, FlowPike, propeller and pbcf, for x, y, and z directions.
- JoRes1-<INSTITUTION>-P-s-Fr<Frvalue:%04.2f>.*: for each F_r value a separate surface field data file should be provided, with (time-averaged) variables: vof, rho, cp, cfx, cfy, cfz, iBlank
- JoRes1-<INSTITUTION>-P-piv-Fr<Frvalue:%04.2f>.*: for each F_r value a separate PIV plane data file should be provided, with (time-averaged) variables: vof, u, v, w, omegax, omegay, omegaz, tke, iBlank
- JoRes1-<INSTITUTION>-P-fs-Fr<Frvalue:%04.2f>.*: for each F_r value a separate data file with the (time-averaged) free surface (iso-surface of $vof=0.5$) should be provided. No additional variables are to be included.

Optional calculation(s)

Participants are encouraged to do additional studies, comprising, but not limited to:

- Formal numerical discretisation uncertainty study. A Csv file with global quantities similar to the above is to be provided, but with rows containing results for each grid and time step size, with additional variables δt , nCells and nFaces. Note that for a formal uncertainty study, geometrically similar grids need to be generated. Additionally, it should be noted that when using wall functions, the average y^+ should be constant across the grids, while for wall resolved calculations the y^+ should be refined with the refinement factor of each grid. Combinations of grid sizes and time steps are required if the discretisation uncertainty is to be estimated in space and in time (at least five calculations are required). Alternatively, the time step can be refined together with the grid simultaneously (i.e. keeping constant Courant number), to limit the number of calculations involved (at least three calculations are required).
- Same as the requested calculation, but without roughness.
- Same as the requested calculation, but without roughness and without additional bilge keel and superstructure resistance.

Document Revisions

- 2024-11-11: Initial version for the website.
- 2024-12-13: Update of hydrostatic properties in Table 1, and specification of point of application of superstructure and bilge keel resistance.
- 2024-12-18: More accurate definition of PIV plane.
- 2025-05-13: Small clarifications for the output (highlighted).
- 2025-05-28: Small consistency improvement (highlighted).