R/V Nawigator XXI

Aim

The aim of this test case is to obtain insight into the capability of CFD to predict model-scale cavitation dynamics, pressure pulses and underwater radiated sound. This test case is designed such that a series of computations starting from open-water, non-cavitating conditions will be carried out leading-up to the study of a cavitating propeller operating in a wake behind a ship. The different conditions allow for decoupling of the sources of discrepancy between the different submissions and provide a wealth of comparison materials. The metrics to be investigated are the propeller revolution rate, time- and frequency-domain pressure signals, flow-visualisation data, and underwater radiated sound.

Description

Within the organization of the Wageningen 2025 CFD workshop there is an interest to include a test-case dedicated to propeller cavitation in model-scale conditions. In the past, two workshops dedicated to cavitating flow simulations have been organized under the umbrella of the International Symposium on Marine Propulsors: the first one in 2011 and a second one in 2015:

- SMP2011: Test cases included the Delft Twist hydrofoil and the Potsdam Propeller Test Case (PPTC) in open water cavitating conditions (https://www.marinepropulsors.com/smp/files/downloads/smp11_workshop/smp11_workshop/I-1_Hoekstra.pdf).
- SMP2015: the Potsdam Propeller Test Case evaluated in non-cavitating and cavitating conditions at an inclined shaft (https://www.sva-potsdam.de/en/smp15-propeller-workshop/).

The PPTC tested at an inclined shaft remains the most industrially-relevant test case as it attempts to reproduce cavitation patterns similar to those expected on a propeller working in a wakefield behind a ship. However, because the propeller was operating in open water conditions with the unsteady loading caused by the shaft inclination angle only, the dynamics of the cavitation were relatively small.

For that reason, it has been decided to expand the propeller cavitation test-case to include a ship wake field, thereby achieving a much more realistic set up. To encourage advancement in the application of high-fidelity methods in marine CFD, it has been decided to focus the test case on simulations including the hull instead of, for instance, prescribing the wakefield or using boundary element methods. Furthermore, participants are highly encouraged to carry out grid sensitivity studies and provide underwater radiated sound predictions to, for the first time, allow thorough benchmarking of the acoustic and fluid dynamic solvers for cavitation and hydroacoustics. The selection of the test case and the operating conditions are discussed in the present memo.

The intended test-case is the Polish research ship Nawigator XXI that has been selected by the International Towing Tank Committee (ITTC) for benchmarking underwater radiated noise measurements of cavitating propellers. Although the ship is relatively small, it is a single-screw vessel with a hull representative of a commercial vessel. As such, it is considered the most promising choice for the current workshop. It is the only test case for which the geometry of both the hull and the propeller can be openly shared. Moreover, the existence of old full- and model-scale measurement data, as well as the expected availability of future experimental results, could make this an extremely relevant canonical test case. An overview of the ship is provided in Figure 1.

Figure 1: Overview photographs of the Nawigator XXI research vessel and its propeller [4].

References

[1] Binkhorst, B.-J., 2005. Results of the full scale LDV measurement campaign onboard of the Nawigator XXI (No. D1.6, G3RD-CT-2002-00810), European Full-Scale Flow Research and Technology (EFFORT). MARIN - Maritime Research Institute Netherlands, Wageningen, the Netherlands.

[2] Bugalski, T., 2005. Basic model test results for the training/research vessel, Nawigator XXI (No. D2.3, CTO RH-2005/T-133E), European Full-Scale Flow Research and Technology (EFFORT). CTO S.A., Gdansk, Poland.

[3] Foeth, E.-J., Bosschers, J., 2021. Memo on CP469 geometry adjustments. MARIN - Maritime Research Institute Netherlands, Wageningen, the Netherlands.

[4] Hallander, J., 2014. Propeller noise experiments in model scale (No. AQUO D2.5), Achieve QUieter Oceans by shipping noise footprint reduction (AQUO), EU FP7 - Collaborative Project no 314227. SSPA, Gothenburg, Sweden.

[5] Rizzuto, E., Villa, D., Gaggero, S., Kellet, P., Haimov, E., Molinelli, E., Hallander, J., Li, D.-Q., 2014. Predictive theoretical models for propeller URN (No. AQUO D2.3), Achieve QUieter Oceans by shipping noise footprint reduction (AQUO), EU FP7 - Collaborative Project no 314227. Università degli Studi di Genova Department of Electrical, Electronic, Telecommunication Engineering and Naval Architecture (DITEN), Genoa, Italy.

[6] Salinas, R., Moreno, A., 2014. On-site measurements-Experimental data for accurate identification and quantification of Cavitation Noise and other sources (No. AQUO D3.3), Achieve QUieter Oceans by shipping noise footprint reduction (AQUO), EU FP7 - Collaborative Project no 314227. SSPA, Madrid, Spain.

[7] Viviani, M., Tani, G., 2014. Cavitation tunnel report CP469 propeller (No. MV-TUN-1402), Achieve QUieter Oceans by shipping noise footprint reduction (AQUO), EU FP7 - Collaborative Project no 314227. Università degli Studi di Genova Department of Electrical, Electronic, Telecommunication Engineering and Naval Architecture (DITEN), Genoa, Italy.

Geometry of the Hull and Propeller

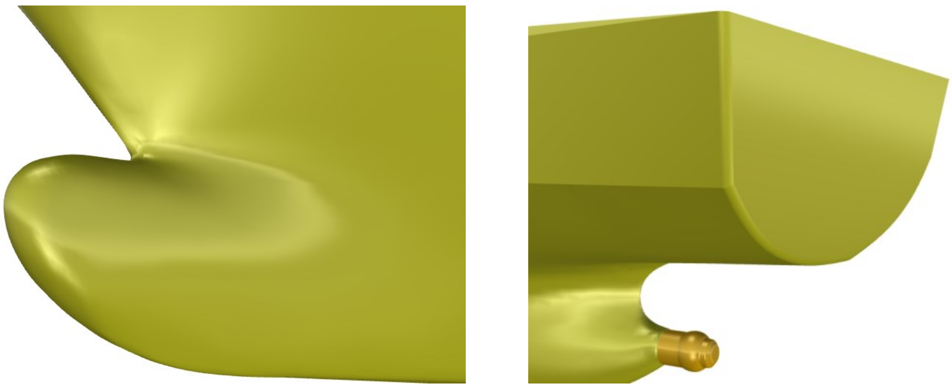
Hull

The main particulars of the vessel are presented in Table 1. The hull is fitted with a bulbous bow and has a relatively slender hull of block coefficient of 0.623. The service speed of the vessel is 12 knots, corresponding to Froude number of 0.2654, making it representative of many merchant vessels. The aft part of the hull is fitted with several observation windows that have been used to conduct cavitation observations and Laser Doppler Velocimetry (LDV) measurements. The full-scale ship is fitted with a horn rudder but this has been omitted in the provided geometry to simplify the numerical set up. Simulations are to be carried out using the design draughts, i.e. at a slight bow-up trim.

Table 1: Main particulars and relevant characteristics of the Nawigator XXI research vessel.

Ship type	Research Vessel, built 1998	
Owner	Maritime University of Szczecin	
Max speed [kts]	13.0	
More info:	https://www.pm.szczecin.pl/en/facilities/research-training-vessel/	
	Full-scale	Model-scale
Scale factor	1:1	1:7
Length overall [m]	60.3000	8.6143
Beam [m]	10.5000	1.5000
Draught (FPP/APP) [m]	3.1500 / 3.2000	0.4500 / 0.4571
Displacement (t)	1150.00	3.35

In order to make the geometry easier to handle in CFD mesh generation programs, the hull has been locally smoothed and rebuilt, as shown in Figure 2. The bulbous bow has been completely rebuilt to remove small edges and discontinuities present in the original geometry available from the design documentation. Part of the stern under the gondola has also been replaced with one large surface instead of several smaller ones. The transom has also been modified by adding a 10 cm radius (at full-scale dimensions), making a double-body grid easier to generate.



a) Bow b) Stern

Figure 2: Detail of the smoothed geometry.

Propeller

Nawigator XXI is fitted with a 4-bladed controllable pitch propeller with design number CP 469. Main particulars of the propeller are summarised in Table 2 and its outline is depicted in Figure 3.

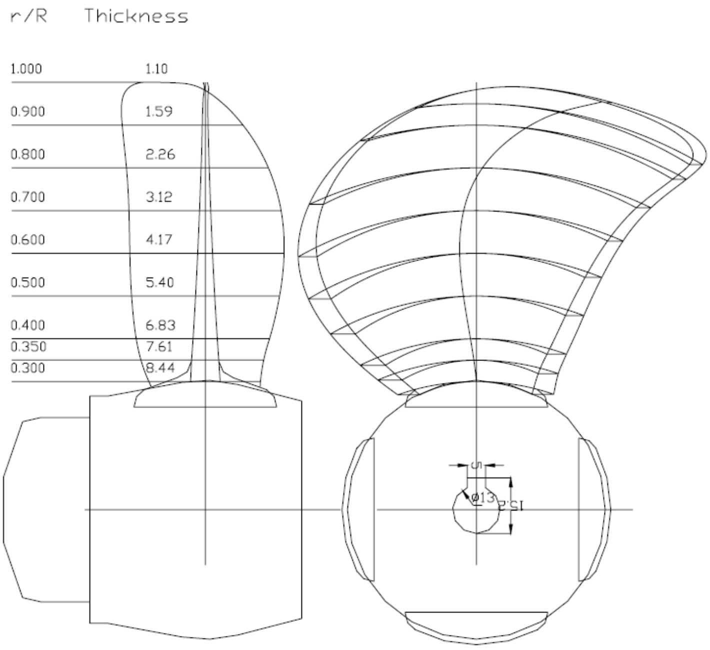


Figure 3: Outline of the propeller and hub. Figure from the original CTO report [2].

The original propeller geometry available from CTO report [2] had to be adjusted to make it more suitable for manufacturing of a model-scale variant and carrying out of CFD computations. This is reported in detail in [3]. The applied changes included:

- Closing the leading edge of the section profiles by setting the section thickness to zero at x/c=0.
- Defining a blade section at r/R=0.9 by interpolating the propeller shape with consideration for its smoothness.
- Setting the chord length at the tip to zero.
- Adjusting the pitch at r/R=0.7 to 0.91 from the design value of 0.94 to match the operating point proposed for the current test case.
- Fairing of the radial distributions using Bezier curves.
- Setting trailing edge thickness of 0.5 mm for all sections for model-scale diameter of 226 mm that was used in the original CTO measurements. Note that the current test case employs a different scale factor.

The final radial and sectional parameters of the propeller shape are shown in Figure 4 and Figure 5.

Note that the propeller is left-turning when looking forward! See Figure 1.

The hub shape has been obtained from digitising the design drawings available from the cache of technical documents about R/V Nawigator XXI. Some of the detail about the adjustable pitch mechanism have been simplified, replacing the hub with a simple revolved shape. Its profile is depicted in Figure 6. Figure 7 presents open water characteristics of the propeller measured in two experimental campaigns.

In the original measurements at CTO and UNIGE no mention was made of tripping the flow on the propeller blades. This leads to much potential uncertainty when considering comparisons to the past experimental data. For the purpose of this workshop, it is advisable to employ suitable numerical tools in order to ensure fully-turbulent flow, e.g. by relying on RANS models (in whole or inside the boundary layer through the use of DES-type turbulence models) or by stimulating upstream turbulence using synthetic turbulence generation techniques.

Table 2: Main particulars of the CP 469 propeller fitted to R/V Nawigator XXI.

Propeller Type	Controllable pitch propeller	
Propeller blade number	4	
Propeller design pitch (P/D)	0.942	
	Full-scale	Model-scale
Propeller diameter (m)	2.2600	0.3229

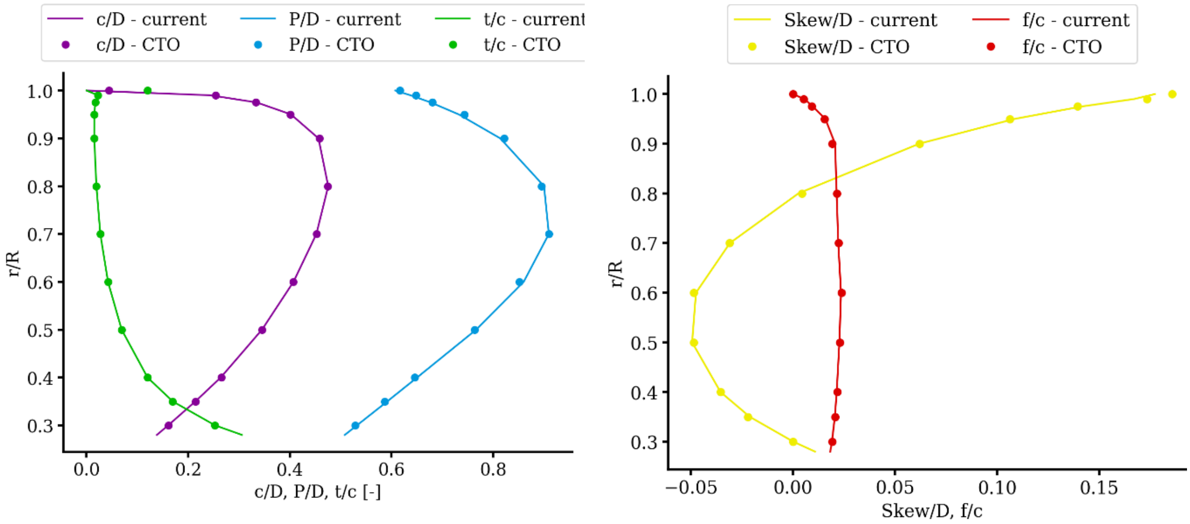


Figure 4: Radial distributions of propeller design parameters. P/D corrected to a nominal value of 0.91 as used in the currently proposed test case. Original values described in the CTO report [2] compared to the smoothed shape used in the current test case.

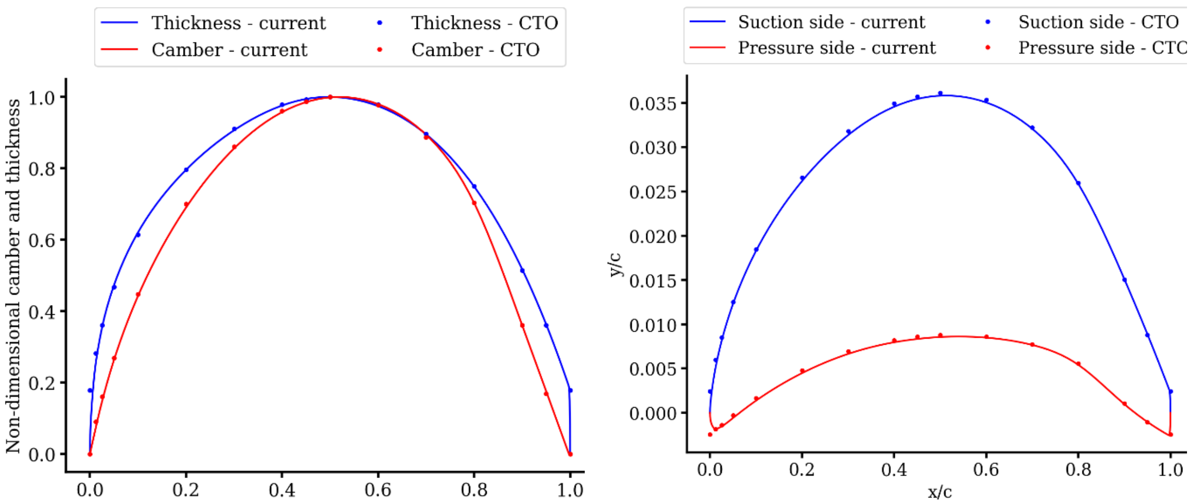


Figure 5: Description of the NACA profiles making up the propeller. Left: Non-dimensional thickness and camber distributions, right: section shape at x/R=0.7. Original values described in the CTO report [2] compared to the smoothed shape used in the current test case.

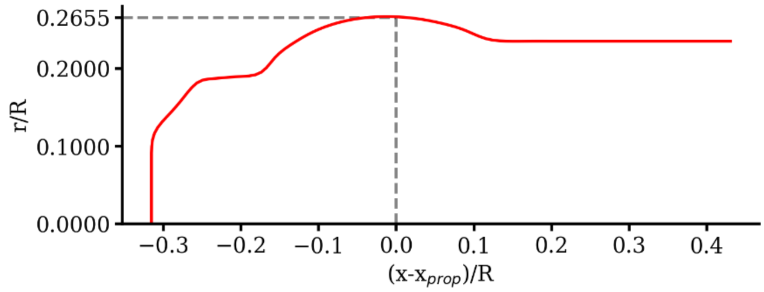


Figure 6: Outline of the revolved hub shape non-dimensionalised with propeller radius.

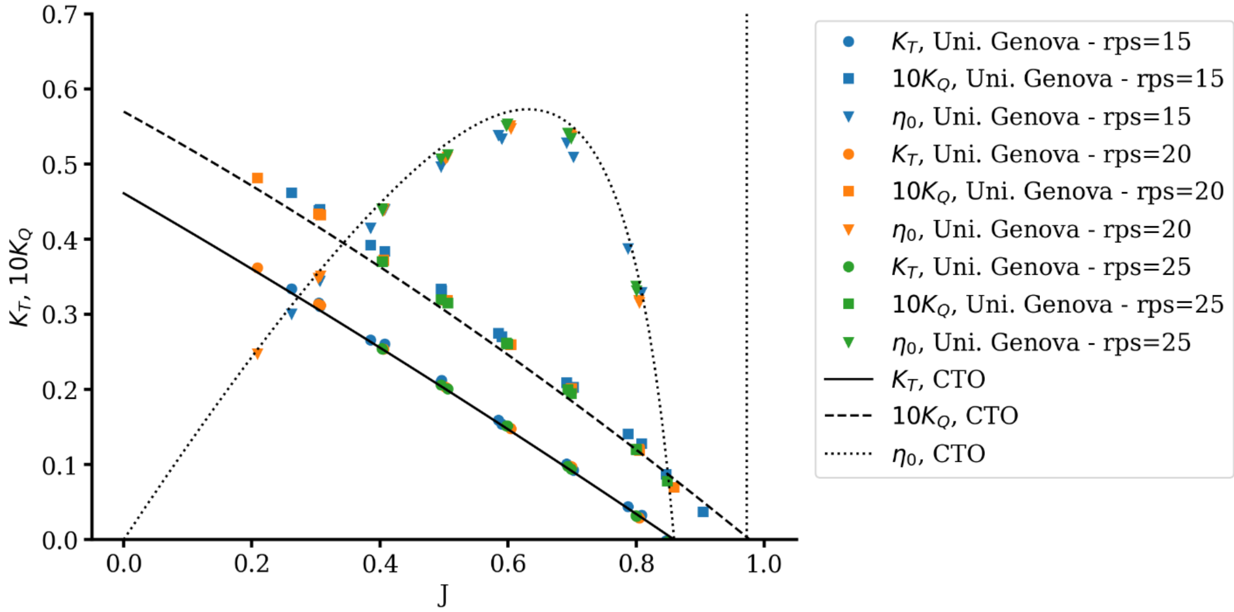


Figure 7: Open water characteristics of the CP 469 propeller. Data from the original towing tank tests of CTO [2] has been interpolated to P/D of 0.91. Data from the University of Genova was obtained in a cavitation tunnel and measured at P/D of 0.91 over a range of Reynolds numbers [7].

Description of available measurements

Full-scale data

Two sets of relevant full-scale measurements exist for the vessel. These have been documented in:

1. EFFORT deliverable 1.6 [1] – the campaign was conducted in the Norwegian Sognefjord and the surrounding area and focused on Laser Doppler Velocimetry (LDV) measurements in front of the propeller, as well as speed, power and thrust measurements.
2. AQUO deliverable 3.2 [6] – these measurements focused on collecting underwater radiated noise (URN) data for a range of vessels, including R/V Nawigator XXI.

For the current test case, no use is planned for the full-scale data. Designing the proposed computations to match the experimental conditions would lead to a large range of possible uncertainties and sources of relative error, diluting the focus of the workshop. However, note has been taken of the range of conditions explored in the full-scale measurements and the proposed numerical operating points fall within a similar regime. This makes the currently proposed computations realistic and should enable the participants to pursue comparisons with full-scale data in related studies. Interested participants should take note of the availability and reliability of the available full-scale data before carrying out further investigations outside of the scope of the workshop.

Model-scale data

The main set of model-scale experiments available for R/V Nawigator XXI is available from the measurement campaign performed at the Ship Hydrodynamic Division of Ship Design and Research Centre S.A. as part of deliverable 2.3 of the EFFORT project [2]. The measurements were targeted to both provide a full-spectrum of results for the model-scale ship but also to match the conditions seen during the matching full-scale operational points. This set of measurements was carried out at a scale factor of 1:10 using both a stock propeller and the propeller fitted to the real ship and considered for the current test case, i.e. CP 469. The results include towed and self-propulsion towing tank data, wake field measurements, and wave cut measurements.

The second set of model-scale measurements was performed at the University of Genoa within AQUO deliverable 3.3 [7]. These focused on cavitation observations for pitch settings of 31% and 79% of the digital pitch indicator onboard of R/V Nawigator XXI, which correspond to P/D of 0.464 and 0.910, respectively. It should be noted that the higher pitch setting is below the design value but has been widely used during the full-scale trials. Presented results include photographs of cavitation extent, force data, as well as cavitation-induced pressures at the tunnel wall.

AQUO deliverable 2.5 presents model-scale noise measurements for R/V Nawigator XXI that were carried out by the University of Genova and CEHIPAR [5]. The report focuses on the comparison of model- and full-scale noise measurements.

Fluid properties

Participants should use in their numerical setup fresh water and air properties and other values defined in Table 3. These correspond to propulsive test conditions for measurement 13067 from the CTO report [2].

Table 3: Properties of the water and air during the trials

Property		Value	Units	Notes
	Water properties			
Density	ρ_w	999.2	kg/m^3	
Dynamic viscosity μ	μ_w	1.18105·10^{-3}	Pa-s	
Kinematic viscosity ν	ν_w	1.18200·10^{-6}	m^2/s	
	Vapour properties			
Density	ρ_v	0.028	kg/m^3	
Dynamic viscosity μ	μ_a	9.996000·10^{-6}	Pa-s	
Kinematic viscosity ν	ν_a	3.571510·10^{-4}	m^2/s	
Saturated vapour pressure	p_v		Pa	To be derived from σ_n and rps for each condition
	Other properties			
Acceleration of gravity	g	9.8067	m/s^2	

Instruction for participants

General information

For the current workshop test case, the following in-behind condition has been chosen as the reference point for all the computations:

- 79% pitch indicator setting (P/D=0.91).
- K_T=0.26 – higher than at working point WP3 (K_T=0.247) from the University of Genoa data set.
- Scale factor 1:7 – larger than the 1:10 used in previous experiments by CTO and University of Genoa to better match the expected future campaigns at large cavitation test facilities.
- Two cavitation numbers
    1. σ_N=2.10 – lower than working point WP3 (σ_N=2.26) from the University of Genoa data set.
    2. σ_N=2.79 – corresponds to the A2 condition proposed for the ITTC round robin tests. These will likely be carried out in the future so the condition remains a blind numerical exercise.

Recall that the cavitation number is defined as $\sigma_n = \frac{p-p_v+\rho g h}{\frac{1}{2}\rho\, n^2 D^2}$ where n is the rps, ρ is the water density in kg/m^3 and D is the propeller diameter in m, p_v is the saturated vapour pressure in Pa, h is the tip immersion in m, and p is the reference dynamic pressure in Pa. The proposed definition is consistent with the original cavitation tunnel tests carried out at the University of Genoa.

Figure 8 compares the proposed working points to the data available in various experimental data sets. The proposed deviations from the available experimental data have been chosen to make the proposed conditions more suitable for comparison of CFD predictions of cavitation. In particular, the cavitation number of 2.10 should lead to a flow regime where pronounced cavitation is present. The nearest condition for which measurements are available is WP3 from the University of Genova data. This produces a tip vortex cavity at blade angles of 0° and 90°; tip vortex cavity in the wake at 180° and 270° blade angles that is almost attached to the blade, and a suction-side sheet cavity at 0° blade angle stretching from 80% of the radius to the tip over 20% of the blade chord [7]. Therefore, for the proposed numerical working point pronounced sheet and vortex cavities are expected.

For open water computations, two advance coefficients have been chosen. One corresponds to the thrust identity condition for the chosen K_T and computed using the CTO open water measurements. The second selected condition keeps the same rotation rate for the propeller but lowers the advance speed such that the loading approximately corresponds to the conditions that the propeller blade sees near the top-dead centre of the wake. This has been computed using the available wake survey data.

For all the conditions, the participants are encouraged to ignore the free surface and treat it as a symmetry plane, i.e. to carry out the computations in the so-called double body configuration. This should greatly reduce the computational cost and allow focus to be placed on the cavitation phenomenon. However, an optional case has been specified where participants can include the free surface for the in-behind conditions.

While the computations are not going to be compared to experimental measurements directly, participants are encouraged to use a consistent domain cross-section 18 m wide and 8 m deep. This corresponds to the dimensions of the Depressurised Wave Basin facility at MARIN at which the model will likely be tested in the future.

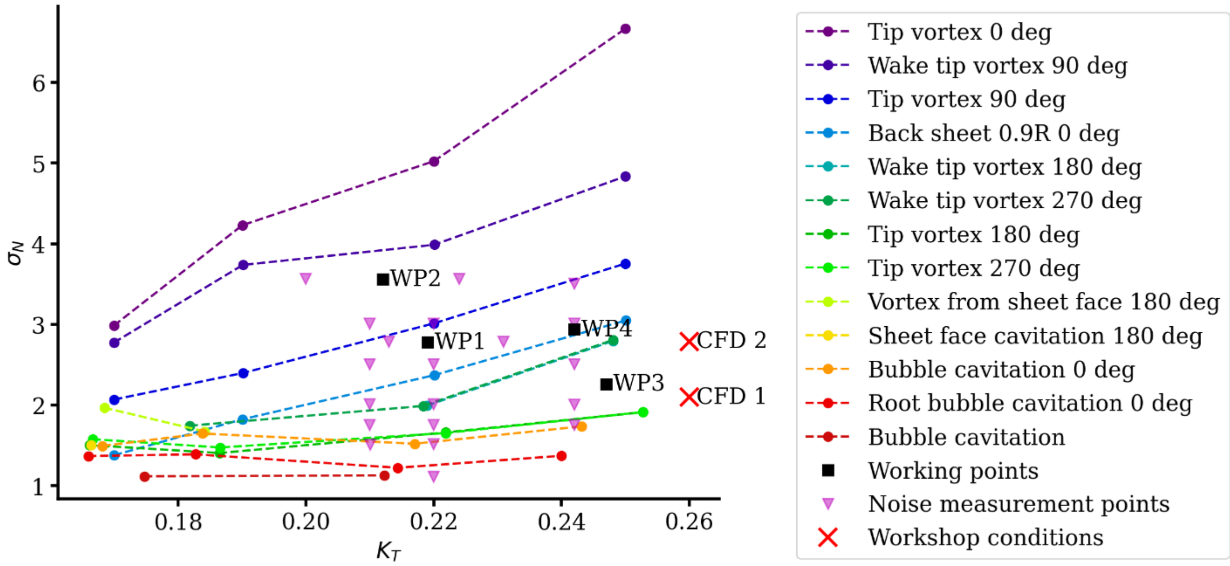


Figure 8: Summary of model-scale experimental conditions present in the data of the University of Genova at P/D=0.91. Operating points 1-4 correspond to: WP1 – 230 rpm, V_s=12.0 kts, K_T=0.219, σ_N=2.78, WP2 – 203 rpm, V_s=11.0 kts, K_T=0.212, σ_N=3.56, WP3 – 255 rpm, V_s=12.0 kts, K_T=0.247, σ_N=2.26, WP4 – 223 rpm, V_s=11.6 kts, K_T=0.242, σ_N=2.94. Operating points used for noise measurements are also indicated. All data are plotted on top of the cavitation inception diagram obtained from [7]. The red crosses indicate the proposed conditions for the current workshop test case.

For the in-behind computations, the correct wake of the hull needs to be determined. Figure 9 presents the wake fraction distribution measured using LDV during the CTO experimental campaign. For the present case, this may be used for validation of the nominal bare hull computations. The wake exhibits a characteristic distribution of velocity deficit with two bilge vortices on either side of the gondola and a wake peak at the top blade position.

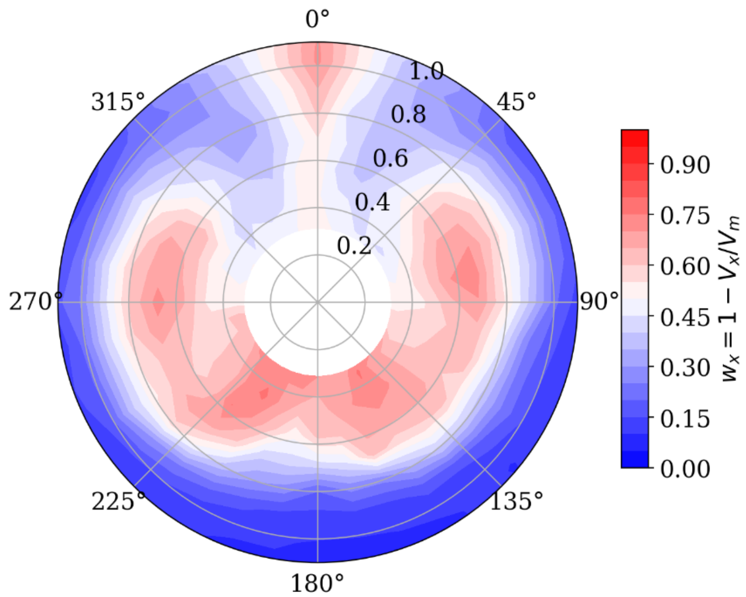


Figure 9: Nominal wake fraction obtained from LDV measurements and presented by CTO [2].

For determination of cavitation-induced pressures, an array of virtual pressure sensors has been designed as shown in Figure 10. The array has a spacing of 0.2D in both x (longitudinal) and y (athwartships) directions and is centred at the propeller location.

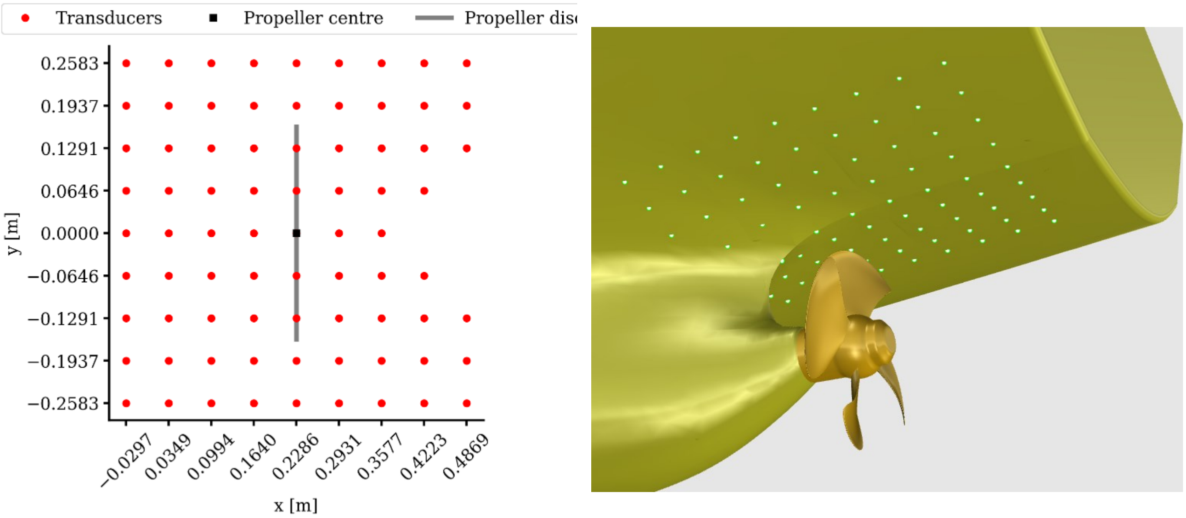


Figure 10: Locations of virtual pressure transducers to be used for induced pressure evaluation (values at scale factor 1:7). Left: top view in the ship coordinate system, right: rendering of the locations (green points) around the propeller.

For willing participants, computation of farfield radiated sound is also highly encouraged. Figure 11 shows the suggested placement of virtual hydrophones in the ship coordinate system. These have been created according to the ISO 17208-1:2016 standard and mimic what would be used should new sea trials be carried out for the vessel. The standard specifies measuring underwater radiated sound in the beam aspect, but for completeness, hydrophones at the centreline of the ship have also been added at the same depths.

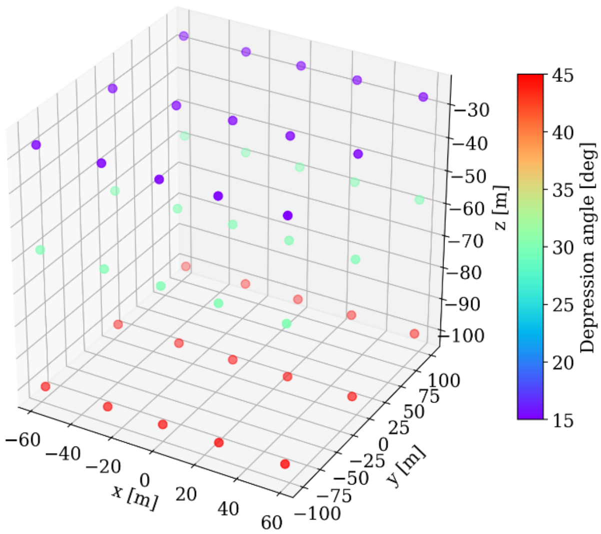


Figure 11: Locations of virtual hydrophones to be used for underwater radiated sound predictions. Dimensions showed at full scale.

Handout summary

In the workshop handout, the following resources are provided:

1. STEP files of the geometry divided into: geometry_hull, geometry_propeller, geometry_hub, geometry_waterplane, geometry_perpendiculars. The provided geometry is at full scale and the correct draughts at FPP and APP.
2. Rhino 3dm file with the same geometry in a single assembly. The provided geometry is at full scale and the correct draughts at FPP and APP.
3. MARIN propeller geometry file "CP469_0910PD_correctedForCFD.dat" that may be used to regenerate the geometry provided as a surface in STEP and 3dm formats. Description of the file

layout is given in the “CP469_readme.txt” file and a Python function for reading it is also provided with the handout.

4. “hubOutline.ipcl” – outline of the hub shape given as an indexed point cloud (full-scale dimensions).
5. “pressureTransducers.ipcl” – locations of the pressure transducers given as an indexed point cloud. An additional “obj” and “dat” files are also provided. The former may be visualized with, for instance, Paraview, and the latter is compatible with the MARIN CFD code, ReFRESCO. Locations are given at full-scale dimensions.
6. “hydrophones.csv” – locations of farfield hydrophones for acoustic pressure computation. Note that these are given as model scale values.
7. “Nawigator_publicLiterature.zip” – collection of articles that reference the Nawigator XXI measurement data.
8. “Nawigator_data.xlsx” – spreadsheet containing relevant data extracted from reports of the EFFORT and AQUO projects that may be used for checking of the computations.
9. “operatingPoint.json” – text file with the given test conditions, such as thrust coefficient, cavitation number, etc. Prefixes “ow”, “ib”, and “ibc” refer to open water, in-behind (wetted), and in-behind (cavitating) conditions. Note that the rotation speed given for the in-behind conditions is an initial guess and an iterative procedure should be employed to arrive at the correct K_T. See guidelines for detail.
10. “particulars_m.json” – text file with key particulars of the ship at model-scale 1:7.
11. “RV_Nawigator.ipynb” – jupyter notebook with Python code for reading the propeller geometry and available measurement data, as well as preparing most of the figures and values provided in this handout.

Guidance on simulation set-up

- Simulations are to be carried out at model scale of 1:7. All geometry files are provided at full-scale dimensions for consistency and hence need to be scaled accordingly.
- Constant ship speed of 2.3331 m/s at model scale, corresponding to full-scale speed of 12 kts, should be used for all computations except for the open water condition at lower advanced ratio.
- For open water computations, a fixed advance ratio, and hence rpm, is defined for each case.
- For in-behind conditions, the average thrust-coefficient of the propeller blades in non-cavitating conditions has been defined. This should ensure like-for-like comparison between the different submissions. Appropriate iterative procedure should be applied for the in-behind, non-cavitating simulation (Case 4) to obtain rpm that yield K_T of 0.26. Recall that $K_T = T/(\rho\, n^2 D^4)$ where T is the thrust in Newtons. Initial guess of 12.5 rps at model scale should provide a reasonable estimate for initialising the iterative procedure. Please note that the same rps should be used for defining the cavitation number for in-behind computations. Presence of cavitation will change the K_T but the magnitude of this change is one of the quantities that will be compared between the submissions.
- Origin of the global coordinate system is at the aft perpendicular, centre line and keel line.
- Nominal wake computations should be carried out with a dummy hub.
- The provided “modelParticulars.json” and “operatingPoint.json” files contain all the necessary quantities, such as water viscosity, density, thrust coefficient, etc. that should be used in the computations.
- Open water computations should also include the original hub. The domain should include either an additional upstream fairing or straight shaft extending up to the inlet. Forces acting on the shaft/fairing should not be included in the submission. Hub and blade forces should be monitored separately, see detailed description of required files.

- Simulations should be run at the design draughts of 3.20 and 3.15 m at aft and fore perpendiculars at full-scale, which corresponds to the design condition of the ship. The provided geometry files already account for this. The resultant trim angle is 0.0528 degrees.
- Propeller normal direction is (0.9999995737778135, 0, 0.0009232790431308681) and the propeller centre at full scale with sinkage and trim applied is at (1.5988, 0, 1.2515) m from the origin.
- Participants are not required to include the free surface and may choose to carry out double-body computations with a symmetry plane instead. A separate case for willing participants (Case 6) has been defined to allow submissions with a free surface. This should be provided in addition to a double-body computation.
- Cavitation number should be set with respect to propeller tip position at 12 o'clock, consistently with the University of Genoa dataset. This corresponds to z=2.3815 m at full scale, or depth of 0.8185 m. An example computation of the corresponding saturated vapour pressure is shown in the provided jupyter notebook.

Guidance on convergence and grid independence

Participants are highly encouraged to carry out convergence studies for the open water and in-behind cavitating test cases (No. 2.1 and 5.1). To this end, the use of procedures described by Eca and Hoekstra is advised (https://www.marin.nl/en/research/free-resources/verification-and-validation/verification-tools). Due to the substantial cost of the computations, emphasis should be placed on the grid independence, ensuring that the iterative convergence and time step size have a negligible effect on the results. For meaningful results, participants who choose to provide grid sensitivity results should submit data for at least three geometrically-similar grids.

To differentiate between the main submission and grid sensitivity cases, folder names corresponding to each grid should be named "Case_1_1_grid2", where the folder name (Case_1_1) will be assumed to refer to the main grid that has been chosen for the submission. Contents of the folders for the intermediate grids should be identical to the main submission specification and the workshop organisers will extract the necessary information and carry out consistent discretisation uncertainty estimates.

Guidance on post-processing

- As much data processing as possible will be performed by workshop organisers using automated scripts to remove data processing variability from the comparisons. To make this possible, participants are asked to adhere to the post-processing guidelines as closely as possible and to be open to iterating the submission in coordination with the organisers ahead of the final submission deadline.
- This also applies to non-dimensionalisation. Therefore, all provided quantities should be dimensional and expressed in SI units, as mentioned in the detailed file descriptions.
- The flow and time-resolved data should be synchronised and presented as a function with the blade angle. This should be zero at the top blade position (blade pointing along the z-axis as in the provided CAD file) and positive clockwise when looking from astern (in the positive x-direction). Adopted blade rps will be used by workshop organisers to determine the instantaneous time for each data point.
- A summary file for each computation should be provided. At the very least, this should include the rps of the propeller, saturated vapour pressure in Pa (for the cavitating cases), time step size in seconds, and the number of cells included in the computational grid. The file should be named "summary.json" and follow the json convention (https://en.wikipedia.org/wiki/JSON). Example file:

```
{
  "rps": 11.0,
  "pSat": -10000,
  "deltaT": 0.1,
  "Ncells": 1000000,
  "Nprocessors": 128
}
```

Recommended file contents going beyond the minimum required:

- CFD solver name e.g. “CFDsolver”: “OpenFOAM”
- Turbulence model name, e.g. “turbulenceModel”: “RANS k-omega SST”
- Momentum convective discretisation scheme, e.g. “scheme_momentum”: “QUICK”
- Where the pressure was specified, e.g. “pressure_reference”: “outlet”
- Domain length downstream/upstream/width/depth, e.g. “domain_length_downstream”: 50.0
- Time discretisation scheme, e.g. “scheme_time”: “Second-order backwards”
- Wetted surface area of the model, e.g. “model_WSA”: 0.00
- Volume of the model, e.g. “model_volume”: 0.00
- Iterative convergence criterion type, e.g. “convergence_norm”: “Linf / L2 / ForceTotalX”
- Iterative convergence value, e.g. “convergence_criterion”: 0.001
- Total number of revolutions computed, e.g. “revolutions_total”: 10
- Number of cores/GPUs used, e.g. “processor_count”: 256
- Number of wall-time hours, e.g. “total_wall_time”: 100
- Processor/GPU type, e.g. “processor_type”: Intel Core i9 14900
- Description of changes to the geometry, e.g. “geometry_modifications”: “None”
- Other miscellaneous comments, e.g. “comments”: “None”

- General information about submitting flow field data:
  - All flow field data should be saved in Paraview-compatible formats. The preferred format is CGNS, but others, such as TecPlot ASCII, vtk, vtm, Ensight Gold, will be accepted as well, as long as they can be read by Paraview. Participants are asked to verify their submission files accordingly.
  - To reduce the size of the data files as much as possible, only the required fields should be included in the submission. Paraview users could use the “PassArrays” filter to ensure this.
  - Variable names should be made consistent with the CGNS notation. Notably: CoordinateX, CoordinateY, CoordinateZ, Pressure, VelocityX, VelocityY, VelocityZ, SkinFrictionX, SkinFrictionY, SkinFrictionZ. Full naming convention may be found at: https://cgns.github.io/CGNS_docs_current/sids/dataname.html . The CGNS standard does not name volume fractions specifically – please use “VapourVolumeFraction”.
  - Coordinate system consistent with the provided geometries should be used for all computations. Where this is not possible (e.g. in panel method codes with radial coordinate systems), appropriate conversions of outputs should be applied.
  - All unsteady flow data should be extracted for a single revolution and the filenames should be appended with the corresponding instantaneous angular position of blade 1 in degrees, e.g. cavityIso_0.5_36.cgns.
  - For RANS simulations, data from the last computed revolution should be provided. For scale-resolving simulations, such as DES or LES, participants are encouraged to compute phase-averaged quantities for at least several revolutions. Where this is not possible due to computational cost, the last revolution data may be provided.
  - Data should be provided at a resolution of 1 degree.
- General information about submitting text-based data:
  - Data describing integral cavity volume, hull pressures, radiated pressures, and forces should be collected for at least three revolutions.

- All time-resolved data (forces, induced pressures, integral cavity statistics) should be provided in csv format. Data should include the instantaneous time value in seconds, the instantaneous angular position of blade 1 in degrees, and the required data in SI units.
- Force variable naming convention should follow the following example "Time, BladeAngle, ForcePressureX, ForcePressureY, ForcePressureZ, ForceFrictionX, …, ForceTotalX, …".
- Pressure signal variable naming convention should follow the following example: "BladeAngle, Pressure1, Pressure2, …" with integers referring to individual receivers/hydrophones in a given set.

- Types of flow field data
  - Cavity iso-contours "cavityIso_XX_YY.cgns" – iso-contours of the vapour volume fraction field at values of XX=0.1, 0.5, and 0.9. Please correct data to (1-VoF) for codes solving for the liquid volume fraction. For codes using level set method, participants should only provide the level set function contour and name the file as if it denoted volume fraction of 0.5. YY should be the instantaneous blade angle in degrees.
    Fields names expected: CoordinateX, CoordinateY, CoordinateZ
  - Blade surface data "bladeSurf_YY.cgns" – data defined on the surface of the propeller blades, excluding the hub. Fields required: vapour volume fraction (1 scalar), dynamic component of the surface pressure, i.e. excluding the hydrostatic part (1 scalar), skin friction vectors defined as τ_w in Pa (3 scalars), face area in m^2 (1 scalar), local normal vector at the face centre defined as a unit vector (3 scalars).
    Fields names expected: CoordinateX, CoordinateY, CoordinateZ, VapourVolumeFraction, SkinFrictionX, SkinFrictionY, SkinFrictionZ, Area, NormalX, NormalY, NormalZ
  - Inflow monitoring plane "inflowPlane_YY.cgns" – plane defined at x= 0.32857 m at scale factor 7 (x=2.3 m at full scale) with normal vector (1, 0, 0), thus ignoring the trim angle. Fields required: velocity (3 scalars), vorticity (3 scalars), pressure (1 scalar), turbulence kinetic energy (1 scalar). Field names expected: CoordinateX, CoordinateY, CoordinateZ, VelocityX, VelocityY, VelocityZ, VorticityX, VorticityY, VorticityZ, Pressure, TurbulentEnergyKinetic.

- Types of text-based data:
  - Forces in Newtons acting on the propeller blades and computed in the global coordinate system. The hub should not be taken into account – "forces_blades.csv".
    Variables expected: Time, BladeAngle, ForceTotalX, ForceTotalY, ForceTotalZ.
  - Moment acting on the propeller blades, i.e. excluding the hub, computed around the propeller rotation axis – "moment_blades.csv". The units should be Nm.
    Variables expected: Time, BladeAngle, MomentTotalX.
  - Forces acting on the hull, i.e. excluding the hub and propeller – "forces_hull.csv".
    Variables expected: Time, BladeAngle, ForceTotalX, ForceTotalY, ForceTotalZ.
  - Forces acting on the hub only – "forces_hub.csv".
    Variables expected: Time, BladeAngle, ForceTotalX, ForceTotalY, ForceTotalZ.
  - Moment acting on the hub, computed around the propeller rotation axis – "moment_hub.csv". Variables expected: Time, BladeAngle, MomentTotalX.
  - Time traces of pressures acting on the pressure transducers on the hull using the same transducer ordering as provided in the locations file – "pressure_transducers.csv". Variables expected: Time, BladeAngle, Pressure1, Pressure2, …
  - (Optional) Time traces of farfield induced pressure – "pressure_farfield.csv". Computing this quantity will most likely require the participants to utilise some form of an acoustic analogy in order to calculate the pressure in the acoustic farfield. Due to the added difficulty of doing so, this part of the submission is to be considered optional but highly encouraged. Variables expected: Time, BladeAngle, FarfieldPressure1, FarfieldPressure2, …
  - Time trace of total cavity volume inside the computational domain – "cavityVolume.csv". This should be calculated by multiplying each cell volume with the local vapour volume fraction and summing

over all cells.
Variables expected: Time, BladeAngle, CavityVolume

- (Optional) Time traces of the final residual of each field variable being solved – “residuals_Linf.csv” and “residuals_L2.csv” for L-infinity and L2 norms. These should be computed at the end of each time step and the header should simply name all the fields as per the CGNS convention.
  Variables expected: Time, BladeAngle, Linf_Pressure, Linf_VelocityX, …, L2_Pressure, L2_VelocityX, …
- (Optional) Time traces of wall time in seconds and outer loops required per time step – “convergenceStatistics.csv”. If the file is provided, the number of cores/GPUs should be provided in the summary file. For clarity, the NoOuterLoops refers to the number of nonlinear iterations per time step – this is only applicable to SIMPLE-like algorithms. For time-explicit codes, a value of zero should be specified.
  Variables expected: Time, BladeAngle, WallTime, NoOuterLoops

Guidance on the submission process

- For each computation, a separate folder with the required data files should be created.
- Folder names should refer to the case and condition numbers e.g. “Case1_2” for case 1 and condition 2. For cases with a single operating condition “Case1” should be used.

Definition of the Package of Cases

Case #	1.1	1.2	2.1	2.2	3	4	5.1	5.2	6 (optional)
Condition label	Open water, non-cavitating, zero gravity		Open water, cavitating, zero gravity		Nominal wake, double body	In-behind, double body, fixed sinkage & trim, non-cavitating	In-behind, double body, fixed sinkage & trim, cavitating		In-behind, fixed sinkage & trim, with free surface, cavitating
Advance/model velocity [m/s]	2.3330	1.2759	2.3330	1.2759	2.3330	2.3330	2.3330		2.3330
Advance ratio / thrust coefficients / rps	J=0.39175	J= 0.21424	J=0.39175	J= 0.21424	N/A (no propeller)	K_T=0.26	rps from Case 4		rps from Case 4
Grid study condition			Yes				Yes		
Cavitation numbers at the tip	N/A		σ_n=2.10		N/A	N/A	σ_n=2.10	σ_n=2.79	σ_n=2.10
Result files required	• summary.json • forces_blades.csv • forces_hub.csv • moment_blades.csv • moment_hub.csv • bladeSurf_YY.cgns		• summary.json • forces_blades.csv • forces_hub.csv • moment_blades.csv • moment_hub.csv • cavityVolume.csv • bladeSurf_YY.cgns • cavityIso_XX_YY.cgns		• summary.json • forces_hull.csv • forces_hub.csv • inflowPlane_YY.cgns	• summary.json • forces_hull.csv • forces_hub.csv • forces_blades.csv • moment_blades.csv • moment_hub.csv • pressure_transducers.csv • bladeSurf_YY.cgns • inflowPlane_YY.cgns	• summary.json • forces_hull.csv • forces_hub.csv • forces_blades.csv • moment_blades.csv • moment_hub.csv • pressure_transducers.csv • cavityVolume.csv • cavityIso_XX_YY.cgns • bladeSurf_YY.cgns • inflowPlane_YY.cgns		• summary.json • forces_hull.csv • forces_hub.csv • forces_blades.csv • moment_blades.csv • moment_hub.csv • pressure_transducers.csv • cavityVolume.csv • cavityIso_XX_YY.cgns • bladeSurf_YY.cgns • inflowPlane_YY.cgns
Optional result files	• pressure_farfield.csv		• pressure_farfield.csv			• pressure_farfield.csv	• pressure_farfield.csv		• pressure_farfield.csv

Document Revisions

- 2024-11-04: Initial version for the website.
- 2024-11-20: Updated hull figure to include the propeller.
- 2024-12-17: Made it clear that the propeller is left-turning when looking forward.