THE DRIVING FORCE OF A SHIP: SIMULATION OF THE FLOW AROUND MARINE PROPELLER

THOMAS SCHULTZ, ALEXANDER CHUPRIKOV,
ALEXEY KHARITONOVICH, TATIANA TREBUNSKIKH

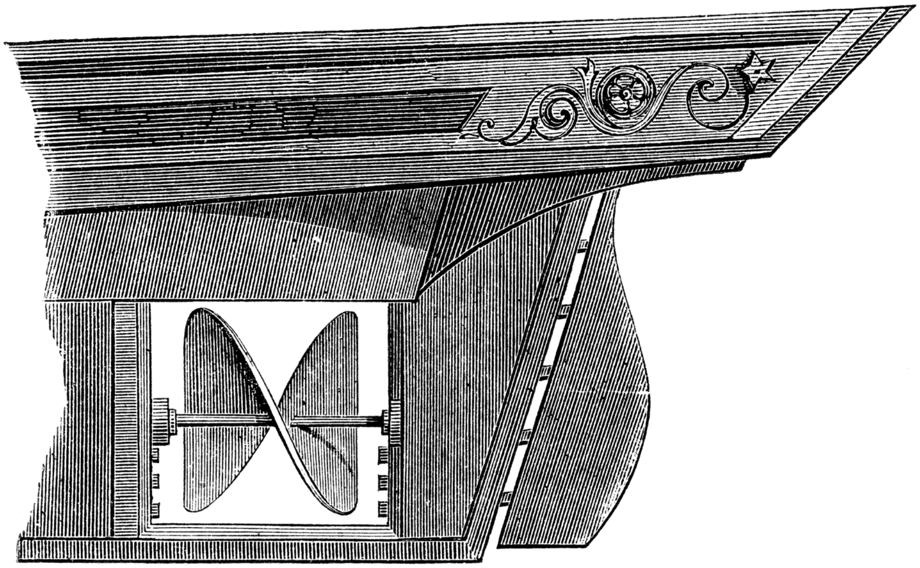
WHITEPAPER

Mentor®
A Siemens Business

MECHANICAL ANALYSIS

www.mentor.com

A small snappy scooter, a half kilometer long tanker and many other different types of ships move through the water using propellers. A marine propeller is a descendant of Archimedes' screw designed to lift water for irrigation and bailing boats. In 1752 Burnelli received a prize for a design of the propeller-wheel and in 1793 French mathematician Paucton proposed the use of propeller as propulsion systems for ships [1]. Three decades later, Czech-Austrian inventor Ressel created the first screw propeller design with multiple blades around a conical base. In an accident in 1836 a damaged two turns wooden propeller was converted into one turn propeller during a voyage that led the boat to travel at double speed. Since then a bladed marine propeller has been used to propel ships through water.



Despite the many years this type of propeller has been used, there are still many studies investigating ways to improve characteristics like strength, durability, efficiency, cavitation behavior and so on.

Experiments are expensive and time consuming, so the most effective way to estimate propeller performance is to use Computational Fluid Dynamics (CFD) which not only provides integral coefficients but also the full detailed picture of the flow around propeller blades including cavitation areas.

To evaluate the capabilities of FloEFD™, the Potsdam Propeller Test Case (PPTC) conducted by Schiffbau-Versuchsanstalt Potsdam GmbH (SVA) has been chosen [2] as a test case. A significant portion of the work by the SVA are research projects. With its three departments, Resistance and Propulsion; Propellers and Cavitation; and Maneuvering and Waves, the SVA covers the entire spectrum of modern shipbuilding testing practice.

The propeller VP1304 which was designed to generate a tip vortex is a controllable pitch propeller with right-handed direction of rotation. The main propeller geometric characteristics are presented in Table 1. The geometry (Figure 1) is available in several neutral formats which can be imported into any CAD program. For this test case, FloEFD for CREO was used and geometry was imported into CREO directly. FloEFD is a CFD tool embedded into Siemens NX, Solid Edge, CATIA V5, Creo and SolidWorks. FloEFD is convenient for design engineers as they conduct CFD analysis inside their preferred CAD tool. Due to high level of automation and SmartCells™ technology, FloEFD enables design engineers to conduct CFD simulations easily on real product geometry, i.e. without any simplification or special geometry adjustments.

TABLE 1: THE MAIN GEOMETRIC CHARACTERISTICS OF VP1304 PROPELLER

Diameter	D = 0.25 m
Pitch ratio r/R = 0.7	$P_{0.7}/D = 1.635$
Area ratio	$A_E/A_0 = 0.77896$
Chord length r/R = 0.7	$c_{0.7} = 0.10417$ m
Skew	$\theta_{EXT} = 18.837°$
Hub ratio	$d_h/D = 0.3$
Number of blades	Z = 5
Sense of rotation	right

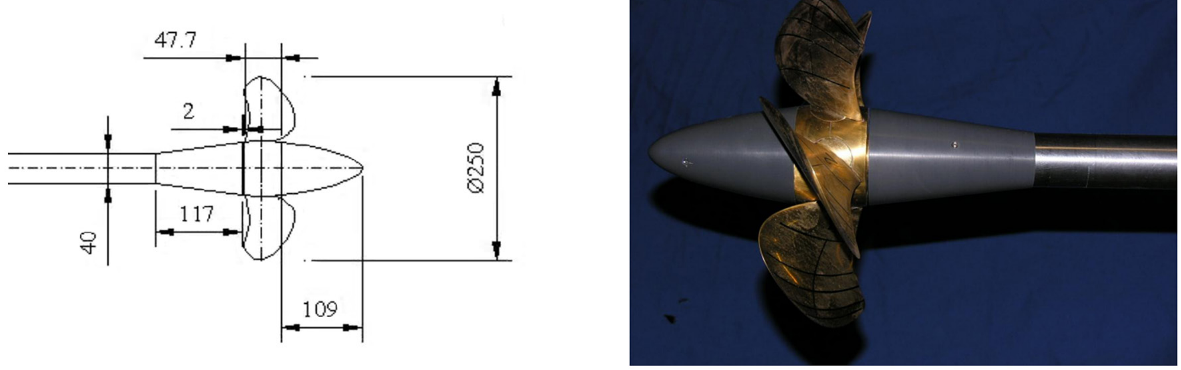


Figure 1. VP1304 propeller model

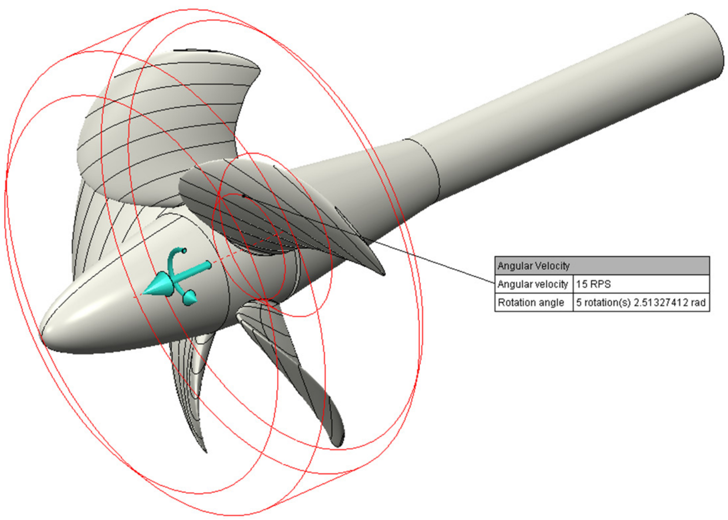


Figure 2. VP1304 scaled model in CREO with FloEFD rotation region

In the simulation for the open water test, two geometric scales of the model are considered: scaled model with scale ratio of 12 and a full model. For these two cases, water properties slightly differ from each other and the rate of revolutions for the scaled model is 15 1/s and 4.33 1/s for the full model. The results of the simulations for five advance coefficients J = 0.6, 0.8, 1.0, 1.2, 1.4 are compared with experimental data. For this comparison, thrust coefficient (K_T), torque coefficient (K_Q) and open water efficiency (η_0) are under consideration.

The mesh in FloEFD was created as uniform with a high mesh density in the vicinity of the blades and extended in the direction to boundaries of the computational domain with some ratio. In addition several local meshes were defined to resolve the propeller blades shape and the area between them for a better representation of flow structure. Due to SmartCell technology it is not necessary to resolve boundary layers by mesh in the traditional CFD sense, it is enough to represent a shape of the object correctly. This approach drastically saves mesh size and, therefore calculation time as well as user time needed for creating a proper mesh. The total mesh size for this task is six million cells (Figure 3).

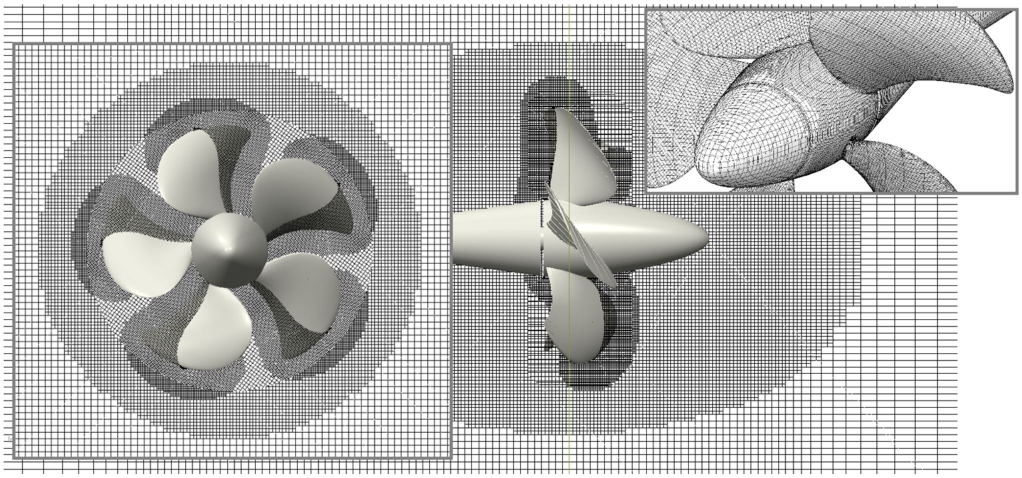


Figure 3. FloEFD mesh with local refinement around propeller blades

The calculations were conducted in the transient regime using "Sliding" mesh technology for the simulation of the propeller rotation. The time step for the scaled model is 0.0024 s and – 0.0083 s for the full model. The simulations were run for 150 time steps for each case and the equivalent angle of rotation for a time step was 13°. It took about 40 minutes to simulate one revolution on an Intel Xeon 3.1 GHz, 8 CPU, 196 Gb RAM. The total workflow and engineer/computational efforts are shown in the Table 2.

TABLE 2. WORKFLOW AND ENGINEER/COMPUTATIONAL EFFORTS

Task	Time, min
Model preparation/Repair	0
Physics model setup	10
Mesh setup/Grid generation	30 (partially computer time)
Solving	350 per case (10 jobs in parallel)
Post-Processing	240
Total human investment	**280 (4.5 h)**
Total time (engineer + solver)	**630 (10.5 h)**

The propeller performance is usually represented using non-dimensional coefficients like thrust and torque coefficients and open water efficiency. The comparison of the simulation results and experimental data in terms of these coefficients is shown in Tables 3, and 4 and Figures 4, and 5. These figures show good agreement between FloEFD data and measurements with relative errors weighted over all advance coefficients for the scaled model of 0.62 % for K_T, 2.65 % for K_Q and 3.18 % for η_0. For the full model these errors are 1.69 % for K_T, 4.46 % for K_Q and 5.86 % for η_0. All data shows that K_T and K_Q decrease while an increase of J and η_0 achieves a maximum at J = 1.4.

TABLE 3. THE OPEN WATER CHARACTERISTICS OF THE *SCALED* PROPELLER MODEL

	FloEFD			Experiment			Error, %		
J	K_T	$10K_Q$	η_O	K_T	$10K_Q$	η_O	K_T	$10K_Q$	η_O
0.6	0.615	1.405	0.418	0.629	1.396	0.430	2.26	0.60	2.84
0.8	0.505	1.200	0.536	0.510	1.178	0.551	0.89	1.86	2.69
1.0	0.400	1.004	0.634	0.399	0.975	0.652	0.21	3.00	2.70
1.2	0.296	0.801	0.704	0.295	0.776	0.726	0.23	3.26	2.94
1.4	0.187	0.584	0.713	0.188	0.559	0.749	0.40	4.56	4.74

TABLE 4. THE OPEN WATER CHARACTERISTICS OF THE *FULL* PROPELLER MODEL

	FloEFD			Experiment			Error, %		
J	K_T	$10K_Q$	η_O	K_T	$10K_Q$	η_O	K_T	$10K_Q$	η_O
0.6	0.613	1.406	0.416	0.631	1.386	0.435	2.84	1.46	4.23
0.8	0.504	1.201	0.534	0.512	1.168	0.558	1.65	2.89	4.41
1.0	0.398	1.008	0.629	0.401	0.965	0.662	0.77	4.51	5.06
1.2	0.294	0.809	0.693	0.297	0.766	0.740	1.13	5.66	6.43
1.4	0.186	0.592	0.700	0.190	0.549	0.770	2.09	7.78	9.16

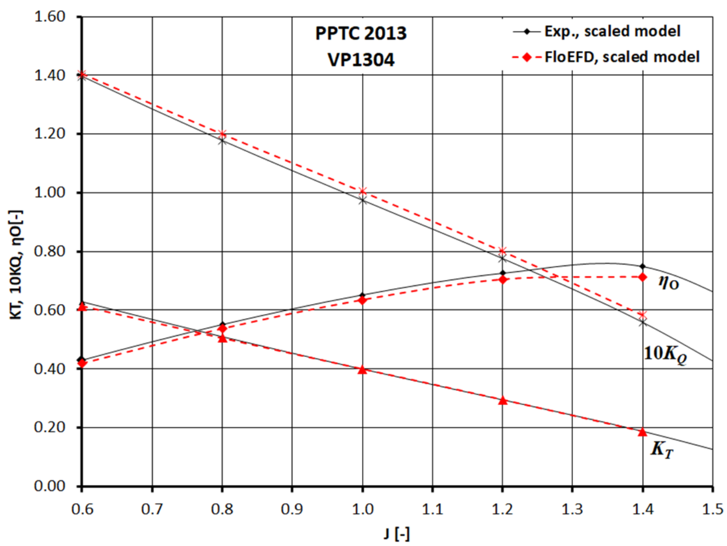


Figure 4. The open water characteristics of the scaled propeller model

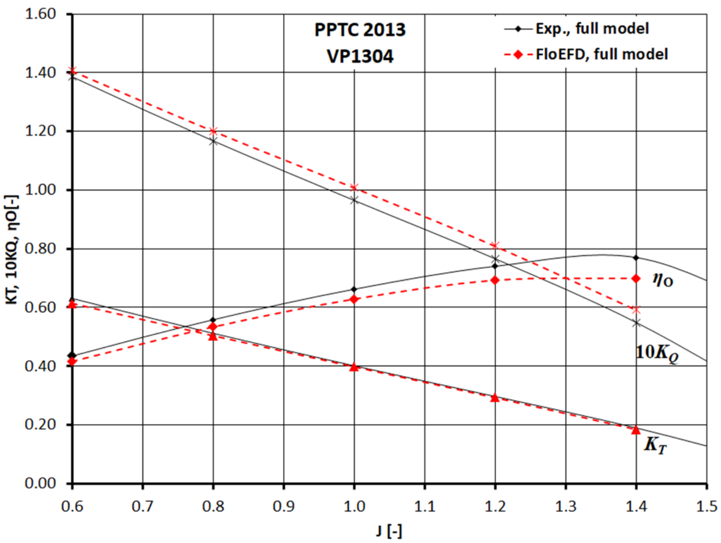


Figure 5. The open water characteristics of the full propeller model

The images in Figure 6 show velocity and pressure distribution around the propeller for J = 0.6 and 1.2 at the center cross section on the side and front planes for the scaled model (Figures 6 and 7).

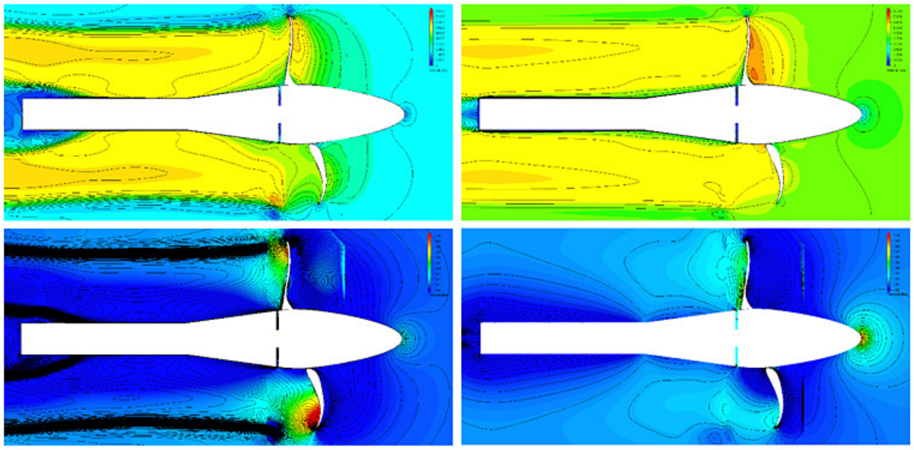


Figure 6. Velocity (top) and pressure (bottom) distributions at J = 0.6 (left) and J=1.2 (right) for the scaled model at the central cross section on the side plane.

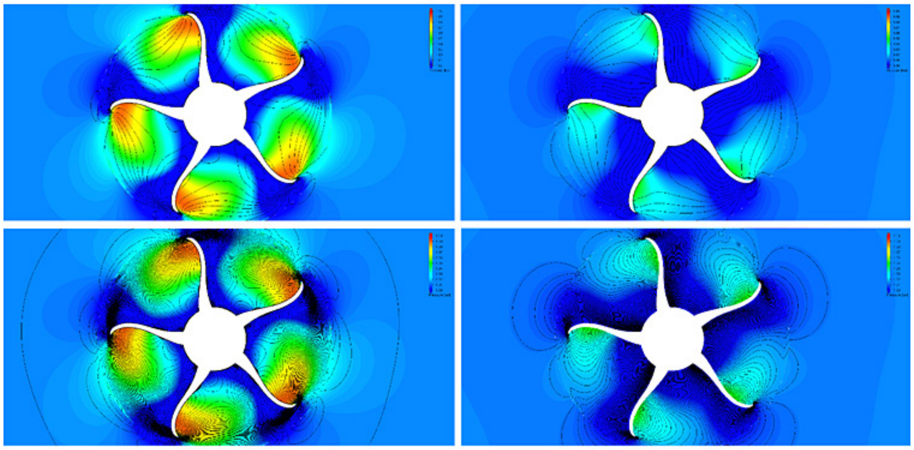


Figure 7. Velocity (top) and pressure (bottom) distributions at J = 0.6 (left) and J=1.2 (right) for the scaled model at the central cross section on the front plane.

Figure 8 shows pressure distribution at the central cross section and isosurfaces of vorticity for the scaled (angular velocity is 15 1/s) and the full (angular velocity is 4.33 1/s) models.

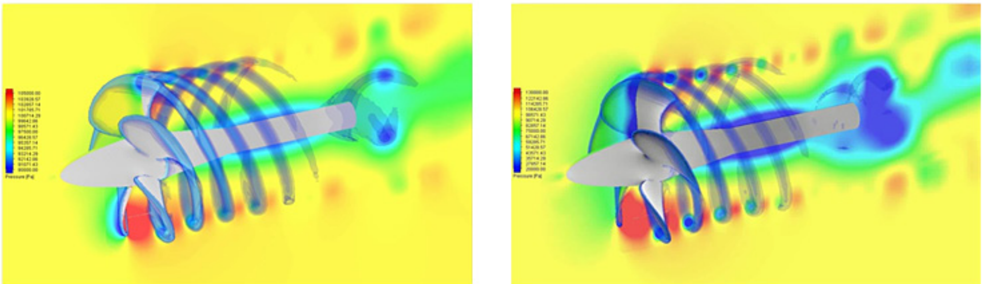


Figure 8. Pressure distribution at the central cross section and isosurfaces of vorticity for the scaled (left) and the full (right) models.

It would seem that to increase the thrust it is enough to increase the rate of revolutions, but this action can lead to a decreasing in the pressure on the face tip of the propeller blades up to the level when water starts boiling, forming a lot of bubbles. When a bubble leaves the low pressure zone, it collapses with a microexplosion. The cavitation process is not only the source of noise and vibration, but also damages the blade surfaces. To create a higher performance marine propeller it is necessary to investigate this process in more detail.

In PPTC from SVA there is experimental data of the cavitation regime for the scaled model in terms of thrust coefficient, so it is possible to compare FloEFD results with the measurements for such complexities for this prediction phenomena. The case with J = 1.019 and cavitation number (σ_n) = 2.024 is considered. For this test, the rate of revolutions is 24.987 1/s.

The comparison between FloEFD predictions and experimental data are shown in Table 5 and Figure 9. It should be noted that they demonstrate good agreement with each other.

TABLE 5. THE CAVITATION TEST CHARACTERISTICS OF THE *SCALED* PROPELLER MODEL

	J	σ_n	K_T
FloEFD	1.019	2.024	0.3703
Experiment			0.3725
Error, %	-	-	0.6

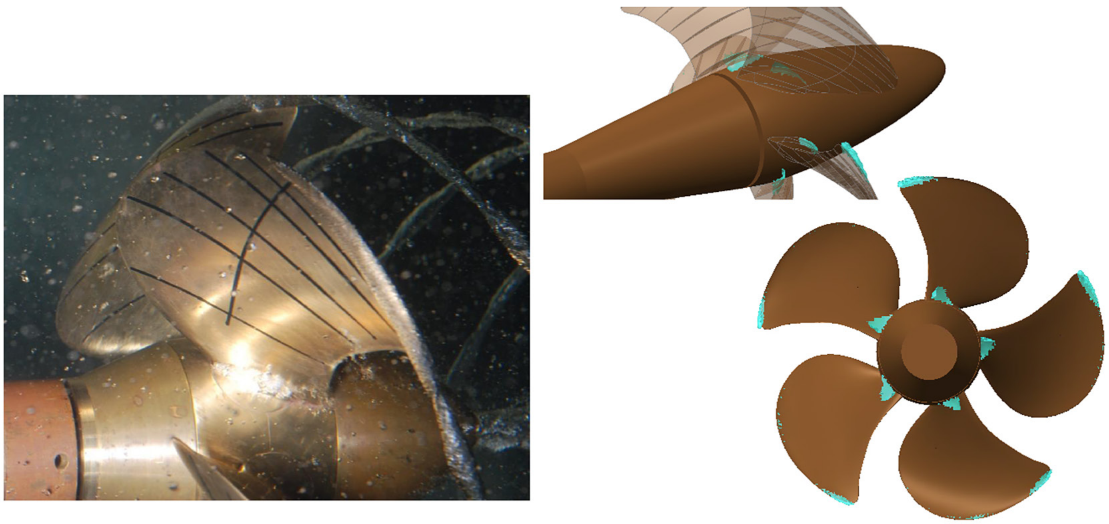


Figure 9. Cavitation observation in the experiment (left) and isosurfaces for vapor volume with value of 20% in FloEFD (right)

This study demonstrates the ability of FloEFD to solve complex marine applications by not only to finding characteristics for the open water tests but for cavitation regimes as well. The option to conduct such a study within a preferred CAD environment is a useful benefit for design engineers who want to optimize the propeller in terms of its performance from the early design stages. Unique SmartCell technology decreases user time needed for creating mesh as well as computational time by reducing mesh cell counts which is especially critical for conducting a series of analysis.

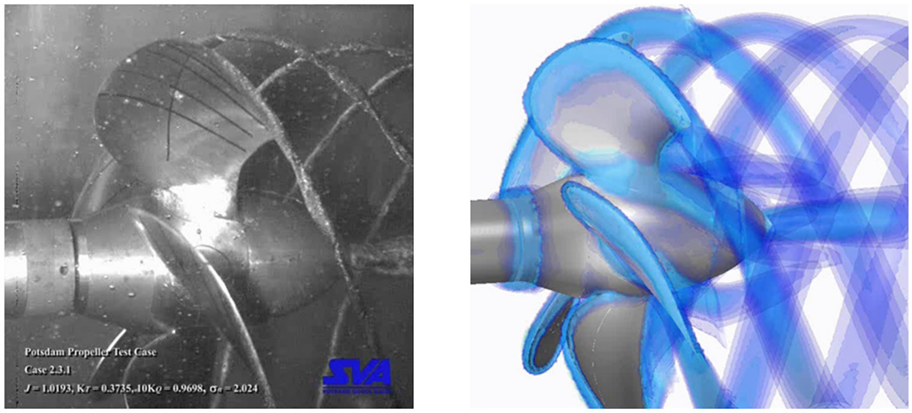


Figure 9. Flow observation in the experiment (left) and in FloEFD (right) for the cavitation regime

REFERENCES

1. Korotkiy R., Neyding M. Behind the stern, in the foam of the sea. Chemistry and life, vol. 10, 1976 (in Russian)
2. **https://www.sva-potsdam.de**

Corporate Headquarters
Mentor Graphics Corporation
8005 SW Boeckman Road
Wilsonville, OR 97070-7777
Phone: 503.685.7000
Fax: 503.685.1204

Sales and Product Information
Phone: 800.547.3000
sales_info@mentor.com

Visit www.mentor.com/company/office_locations/ for the list of Mechanical Analysis Division Offices

Mentor
A Siemens Business

MGC 03-18 TECH17190