Second International Symposium on Marine Propulsors
smp'11, Hamburg, Germany, June 2011
Workshop: Cavitation on foil

smp'11 Workshop - Case 1: DelftFoil

M. Hoekstra[1], T. van Terwisga[1,2], E.J. Foeth[1]

[1]MARIN, Wageningen, The Netherlands
[2]Delft University of Technology, The Netherlands

ABSTRACT

As a part of the SMP-2011, a CFD-workshop was held on the computation of non-cavitating and cavitating flow for the so-called Delft Twist-11 Foil. Five parties submitted results and this paper gives a summary based on an inter-comparison of the information provided.

1 INTRODUCTION

Numerical simulation of cavitation in flowing liquids, including viscous effects, is a quickly developing field in propulsion research. The majority of these simulations are based on solutions of the Navier-Stokes equations for a variable-density fluid, under the spatial or temporal averaging assumptions of LES, DES or RANS. A model governing the evaporation and condensation processes is added to get a closed equation system.

In spite of the apparent enhanced physical realism of the simulations, compared with results of the conventional inviscid flow models, there are formidable tasks ahead. Cavitation is an extremely complex phenomenon, not only influenced by Thoma (cavitation) and Reynolds numbers but also by water quality (nuclei spectra, dissolved air content) and sometimes by experimental facility characteristics. So, once the numerical difficulties are under control, the tuning of the simulation model with experimental information is probably a long-term exercise.

Workshops in which various groups try their methods on a common test case have proven to be useful. The results elicited from such workshops, can confirm that different codes using essentially the same mathematical modeling produce similar results and they can help us understand to which extent RANS is applicable and what the added value of LES or DES may be. In connection with the Symposium on Marine Propulsors, a workshop was therefore organised with the purpose of sharing knowledge and experiences. This paper is solely concerned with one of the two test cases selected, viz. the Delft Twist-11 Foil.

The rationale for using the Delft Twist-11 Foil as a test case for numerical simulations is its focus on the physical phenomena that play a role in sheet cavitation and vortex cavitation, the latter type of cavitation occurring in the break-up process of sheet cavitation. The Twist-11 foil has been tested and documented for steady and unsteady inflow. In the latter case, an oscillation was forced in the flow by two foils in bi-plane configuration using actuating flaps. For this workshop, only the steady inflow condition is considered, which in the experiment nevertheless gives rise to unsteady behavior of the sheet cavity. Critical elements in validating the CFD are a proper simulation of cavity extent and the dynamic behavior. Quantitatively this should be reflected in proper prediction of the unsteady lift on the foil and the measured shedding frequency.

2 CASE DESCRIPTION

2.1 Geometry

The Delft Twist-11 Hydrofoil is a wing of rectangular planform, the section shape being uniform over the whole span, but the orientation with respect to the incoming flow varying in spanwise direction. If the angle of attack is 0 degrees at both ends of the wing, it is 11 degrees at mid-span. Details of the wing's geometry are given below.

The reference coordinate system is a right-hand system with the *X*-axis in flow direction, the *y*-axis in spanwise direction and the *z*-axis directed upwards. The origin of the coordinate system is at midchord and midheight of the wing section on the tunnel wall. We define also a coordinate $x=X-0.5$, so that *x* varies between 0 and 1 from leading edge to trailing edge of the foil.

The section shape is a modified symmetrical NACA 4-digit profile. Its basic shape is given by Abbott and von Doenhoff [1] by the thickness distribution:

$$z(x) = \frac{t}{0.20}\left(0.29690\sqrt{x} - 0.12600x - 0.35160x^2 + 0.28430x^3 - 0.10150x^4\right)$$

in which *x, z* and *t* have been nondimensionalised with the chord length *c*, which is 0.15 m in absolute size, while the thickness-length ratio is t=0.09. Although this section shape has a finite thickness at the trailing edge, viz. $2z_{x=1}$=0.00185, it was just too small to satisfy the requirements of the milling process by which the Delft Twist-11 foil was manufactured. A correction on the thickness distribution was therefore applied which is defined by

$$\pm\Delta z(x) = \left(\frac{x - x_{sp}}{1 - x_{sp}}\right)^2 (t_{min} - z_{x=1}) H(x_{sp})$$

with x_{sp}=0.35, t_{min}=0.0002/c while H is the Heavyside function. So the thickness is gradually increased from 35 % of the chord length towards the trailing edge to get an absolute trailing edge thickness of 0.4 mm.

The twisted foil has been constructed based on a spanwise angle-of-attack variation of the section profile given by

$$\alpha(y) = \alpha_{max}[2(|y-1|)^3 - 3(|y-1|)^2 + 1]$$

where y is again nondimensionalised with the chord length c and varies over the range $0 \leq y \leq 2$, the spanwidth being twice the chord length. As the name of the foil indicates, α_{max} =11 degrees. The angle of attack variation is achieved by rotating the section shape about the midheight point at 75 % of the chord length from the leading edge (see Figure 1).

The mid-chord point on the nose-tail line of the section profiles at the tunnel walls was positioned at the mid-height of the test section. Subsequently the whole foil was rotated about the y-axis, connecting these two points, by -2 degrees, i.e. leading edge downward.

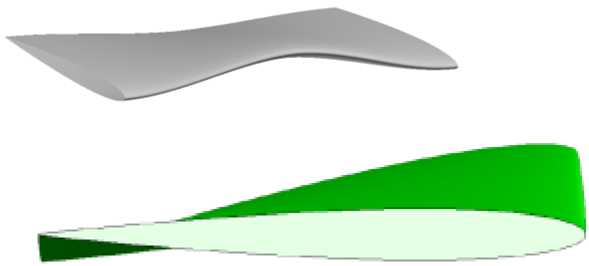

Figure 1: Rendered image of Delft Foil and illustration of angle-of-attack variation

An IGES file with the complete geometric decription of the Delft Twisted Foil as a B-spline surface was made available to participants of the Workshop. This file can be found at the Delft University ftp site: http://maritimetechnology.tudelft.nl/SHS/ Please also observe the agreement for use of this data set: http://maritimetechnology.tudelft.nl/SHS/open%20source%20license_vs2%206170.pdf

2.2 Test Set-Up

The Delft Twist-11 Foil has been tested in the cavitation tunnel of Delft University. Figure 2 gives an impression of this tunnel. Its test section currently measures LxBxH=600 x 300 x 300 mm and is shown in Figure 3. Sand roughness was applied along the leading edge as turbulence stimulation.

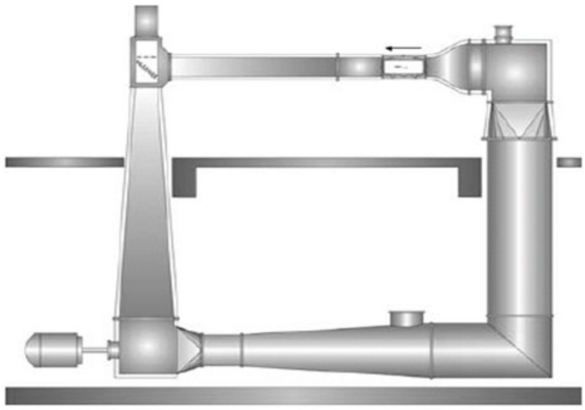

Figure 2: Cavitation tunnel at Delft University of Technology

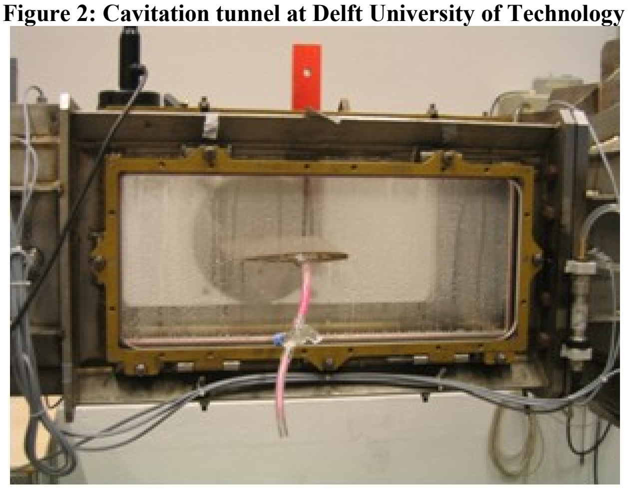

Figure 3: Close-up of test section

Next to test results under nominally steady inflow, experimental data were also gathered for oscillatory inflow conditions. The unsteady inflow was generated with two additional foils with oscillating tails in the immediate vicinity of the twisted foil (see Fig.).
A complete account of the experimental work on this foil has been given by Foeth [2].
The main objective of the current test case is to compare the numerical results of all participants for the same configurations under both wetted and cavitating flow, and - where possible - to validate them against the experimental data. We emphasize that some conditions chosen for the tasks are slightly different from the ones of the experiment.

2.3 Workshop Targets

Participants of the workshop were requested to simulate the flow around the foil in the tunnel under uniform inflow conditions. Conditions were specified for two cases: flow without cavitation (case 1) and flow with cavitation at σ = 1.07 (case 2). In both cases the wing orientation in the tunnel was specified as -2o angle-of-attack at the side walls. The size of the computation domain was prescribed: in the longitudinal direction 7 chord lengths (2c ahead of the leading edge, 4c behind the trailing edge), in the vertical direction tunnel test section height and in the spanwise direction from wall to mid-span (symmetry of flow field assumed). Because the primary interest was the prediction of the cavitation behavior on the foil it was permitted to apply free-slip conditions on the tunnel walls.

Case 1:

- Inflow speed: 6.97 m/s
- Pressure on outlet plane: 97.0 kPa
- Reynolds number based on chord length: Rn=

Requested information: field variables, forces on the foil, Cp_min and Cp_max.

Case 2:

- Inflow speed: 6.97 m/s
- Pressure on oulet plane: 29.0 kPa
- Vapour pressure: 2.97 kPa (implying σ = 1.07)
- Water temperature: 24 oC
- Water density: 998 kg/m3
- Vapour density: 0.023 kg/m3

Requested information: as under case 1, but in addition time histories of cavity volume, source term integral and forces on the foil; dominant shedding frequency.

Participants were encouraged to check grid sensitivity of the results.

Finally it may be mentioned that the same test case has featured in two earlier workshops under the EU-funded Virtue Project.

3 PARTICIPANTS AND METHODS

Results were submitted by five participants as listed below:

Table 1: List of participants

Affiliation	*Name*	*Country*	*Method*	*Mass transfer*
Chalmers University	Rickard E. Bensow	Sweden	LES, DDES, RANS	Sauer
Lloyds Register	Stewart Whitworth	United Kingdom	DES, RANS	Sauer
TUHH	Thierry Maquil	Germany	RANS	Zwart
VICUS	Marcos M. Fernandez	Spain	RANS	-
MARIN	Martin Hoekstra	Netherlands	RANS	Sauer

The contribution from Chalmers gives a comparison of the results obtained with LES, DDES and RANS (with and without eddy viscosity correction) in an attempt “to add knowledge on the impact of different modelling approaches when implemented in the same way, in the same code using the same numerics”. The results were obtained on a single grid of 2.7 MCells, although the LES simulations were extended on a second grid with 4.8 MCells. The Spalart-Allmaras turbulence model was employed in the DDES and RANS computations.

Also LR submitted an extensive data set. A comparison of DES and RANS results is included as well as a grid sensitivity study, the grid density varying from 1.17 to 19.33 MCells. Wall functions were applied. The non-cavitating flow was computed on 12 grids with the k-ω SST turbulence model. For the cavitating flow simulations three grids were used, a 3.71 MCells grid for RANS with two-equation turbulence models (k-ω SST, realizable k-ε, cubic k-ε), a 3.92 MCells grid for RANS with two kinds of Reynolds stress models and a 4.74 MCells grid for DES.

TUHH has provided results on a single grid with 0.3MCells with full near-wall resolution. The standard k-ω turbulence model was applied to both the non-cavitating and cavitating flow case.

VICUS submitted results for the non-cavitating flow (case 1) only, using RANS.

MARIN applied RANS with the k-ω SST turbulence model with and without eddy viscosity correction in the vapour zone. All results were obtained on a single grid with 1.2 MCells with full near-wall resolution. Solutions for a range of cavitation numbers are available.

4 DISCUSSION OF RESULTS

4.1 Non-Cavitating Flow

All participants having submitted results for the non-cavitating flow report an attached boundary layer on the entire foil (no boundary layer separation). The force coefficients and the extreme values of the pressure coefficient are given in the table below:

Table 2: Lift, drag and pressure extremes

	CL	*CD*	*Cp_max*	*Cp_min*
LR	0.4268 ±0.2%	0.0151	1.02	-3.18
TUHH	0.3653	0.0242		-3.23
MARIN	0.4279	0.01447	1.0187	-3.170

The results of LR and MARIN are very close to each other; TUHH’s results deviate substantially. Reasons may be the coarser grid or the presence of the two oscillator foils as rigid bodies in the simulations. Also the turbulence model may have played a role, bearing in mind that the turbulence model is also acting as the laminar-to-turbulent transition model.

Chalmers did not submit results for case 1, but earlier results obtained in 2008 by Huuva [3] showed for the force coefficients in RANS computations CL=0.431 and CD=0.0163, and in LES simulations CL=0.379 and CD=0.0156.

The grid sensitivity study carried out by LR shows scattered results without a clear trend from coarse to fine grids for the lift coefficient. Maybe this was hardly to be expected because the grids are unstructured and therefore not geometrically similar in the sense as required by the Richardson extrapolation process. Even so, the variation in the lift force on the 12 grids was within ±0.2%. The much smaller drag values on the other hand show a clear trend (practically linear) of decreasing CD with grid refinement. For Cp_min LR estimated the uncertainty range as -3.18±1.0%.

That Cp_max is slightly above 1 is to be expected because the reference pressure has been chosen as the pressure at the outlet plane. In an unbounded flow field the reference pressure is preferably chosen as the pressure in absence of the foil. In a confined flow like the flow in a water tunnel, the choice of the reference pressure is not as obvious. The drag of the foil implies that there must be a pressure drop from inlet to outlet. Therefore, with the average pressure at the inlet taken as a reference pressure for Cp determination, all Cp values, so also the value of Cp_max, would drop.

4.1 Cavitating Flow

The cavitating flow is of course of primary interest in this workshop. The results submitted by the participants have turned out to be a rich source of information. This is not only because the results of different participants can be compared, but also because some participants made explorations beyond the conditions of the requested case. Notably the differences in results of RANS, DES and LES models are revealed.

The following table gives an overview of the contributions:

Table 3: Overview of approaches

	LES	*DES*	*RANS_corr*	*RANS*
Chalmers	x	x	x	x
LR		x		x
TUHH				x
MARIN			x	x

RANS_corr means that RANS equations are solved but that a reduction of the eddy viscosity is applied in the vapour region as a crude attempt to account for the interaction between turbulence and cavitation. This correction has first been proposed in [4] and is here referred to as the Reboud correction. LR reports to have tried - in a late stage of their work - to run RANS with this correction but without success.

Chalmers, LR and MARIN indicate that the RANS-solution without correction leads to nominally steady flow and a stationary cavity. In contrast, TUHH comes with an almost cyclic cavitation behavior, the cavity practically vanishing in each cycle; it is a breathing cavity rather than a dynamic cavity with cloud shedding. LR reports that the cavity is stationary irrespective of the turbulence model used. Only with the quadratic Reynolds stress model unsteady behavior seemed to develop but the computations broke down so it is not clear whether the dynamic behavior would sustain. Perhaps the mass transfer model plays a role in invoking dynamic behavior (TUHH using Zwart's transfer model, the other participants using Sauer's model), but before we can conclude on such a sensitivity confirmation by other studies is needed.

From this limited amount of results one might be tempted to conclude that multiphase RANS models without eddy viscosity correction apparently lead to steady or at most "breathing" cavities without shedding. But this is not true in general. For example, Hoekstra [5] has found for the NACA0015 foil at six degrees angle-of-attack dynamic behavior below a certain level of the cavitation number with evident shedding by applying RANS without the eddy viscosity correction. Actually, activating the Reboud correction in simulations for this slightly thicker foil seems to spoil the correlation with experimental data.

The results of RANS with correction (Chalmers and MARIN) show different results. In the Chalmers results shedding is hardly visible, while in the MARIN results there is somewhat better correlation with the experiment as well as with results previously reported by Li et al. [6].

All participants report that the dynamic behavior of the cavity is not exactly periodic in the simulations. Nevertheless FFT analysis yields a dominant frequency and the results are summarized in the following table:

Table 4: Overview of computed oscillation frequency

	LES	*DES*	*RANS_corr*	*RANS*
Chalmers	34	30	28	-
LR		35±2.5		
TUHH				38.79
MARIN			38	

As indicated by the LR result these frequencies have a considerable uncertainty band, partly because the number of cycles of the dynamic process included in the time series on which they are based ranges between 10 and 30, roughly. Moreover, MARIN's results indicate that the shedding frequency is dependent on the cavitation number: the frequency decreases with a decrease of sigma.

The experimental shedding frequency can be derived from Figure 4 where the cavity length is plotted against time against the green background representing the foil chord. This plot gives an impression of the periodicity of the behavior. A shedding frequency of 32.2 Hz is derived from this graph. Notice the slightly different conditions with respect to inflow speed and cavitation number.

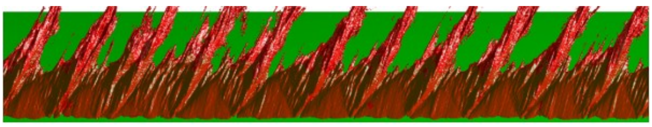

Figure 4: Top view of the experimental observations of cavitation on the Twist-11N foil at α=-2 deg. Flow from bottom to top with time progressing from left to right for a period of 0.3417 s. Experiments carried out with upstream biplane oscillator at rest. U0=6.83 m/s, σ=1.11.

Interesting is also the comparison of the eddy viscosity distribution in the vapour region between DES, RANS_corr and RANS reported by Chalmers. The DES and RANS results are more similar than the DES and

RANS_corr results. This seems to be in contrast with the assumption that a reduction of the eddy viscosity level in the vapour region is needed to stimulate dynamic behavior.

For the RANS simulations LR reports a total vapour volume of 3x10-6 m3, MARIN 2.4x10-6 m3 while TUHH found more than 10 times that amount: 5x10-5 m3 as an average (5x10-10 m3 as a minimum value and 2x10-4 m3 as a maximum). Chalmers reports for LES a variation of the vapour volume between 4x10-6 m3 and 10x10-6 m3; LR for DES between 4.5x10-6 m3 and 11x10-6 m3 and MARIN for RANS_corr between 3x10-6 m3 and 9.5x10-6 m3. So the results of the participants employing Sauer's model are fairly consistent, while Zwart's model seems to produce considerably more vapour, although this is not evidenced by the iso-surface plots of the cavity shape.

As a check of the numerical accuracy, participants have been requested to compare the integral of the cavitation source term (right-hand side of transport equation for the vapour volume fraction) and the time-derivative of the integrated vapour volume. Both quantities must be equal. Unfortunately, only Chalmers and MARIN produced these data. Both show very nice agreement.

5 CONCLUDING REMARKS

Numerical problems in simulating cavitation of foils with RANS, DES or LES still remain. This is partly reflected by the observation that different people with different codes, but solving essentially the same mathematical problem (same type of equations, same turbulence model, same cavitation model), do not produce similar results.

Cavitation predictions seem sensitive to grid density and time resolution, which is maybe what should be expected. In most LES simulations the filter width depends on the grid density, while in RANS simulations of time-dependent flows the resolved frequencies are supposedly incorporated as well in the turbulence model. This means that the mathematical model is changing with the spatial and the temporal resolution.

On the other hand it is a pleasure to observe that in some respects there are reassuring consistencies in the results. For instance, the spreading in the predictions of the oscillation frequency is modest. Besides, the picture of the differences between RANS and DES/LES simulation results has also become clearer.

The usefulness of this kind of workshops can hardly be overestimated. Also the present workshop has been informative and instructive to all participants and has put us all in a better position for making reliable numerical simulations of cavitation.

REFERENCES

Abbott, I.A. and von Doenhoff, A.E.: Theory of Wing Sections, Dover Publications (1958).

Foeth, E.J.: The structure of Three-Dimensional Sheet cavitation, PhD Thesis, Delft University, ISBN 978-90-6464-236-4 (2008).

Huuva, T.: Large eddy simulation of cavitating and non-cavitating flow, PhD, Chalmers University of Technology (2008).

Coutier-Delgosha, O., Reboud, J.L. and Fortes-Patella, R.: Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation, Jnl. Fluids Engineering, Vol. 125, pp. 38-45 (2003).

Hoekstra, M.: Exploratory RANS simulations of partial cavitation and its dynamics, paper to be presented at MARINE2011 Symposium, Lisbon (2011).

Li, D.-Q. and Grekula, M.: Prediction of dynamic shedding of cloud cavitation on a 3D twisted foil and comparison with experiments, 27th Symp. on Naval Hydrodynamics, Seoul, Korea (2008).