CFD validation of different propeller ducts on open water condition.

Alejandro Caldas, Marcos Meis and Adrián Sarasquete from Vicus Desarrollos Tecnológicos, Vigo

a.caldas@vicusdt.com
m.meis@vicusdt.com
a.sarasquete@vicusdt.com

Intro

Trawlers account for the largest percentage of fishing vessels in Galicia. This kind of ships needs to provide high thrust at low advance ratios (it's usual operating velocities are around 3.5kn when the ship is towing the fishing net); because of this fact generally their propulsion units consist on ducted propellers. This paper summarizes some of the CFD calculations performed as starting point for trawler ducted propeller studies and highlights the capabilities of CFD as a valuable tool for the prediction of propulsive factors for ducted propellers. The calculations have been performed for a controllable pitch propeller with two different nozzle geometries. For all the calculations the mathematical model employed is Reynolds Averaged Navier Stokes based, coupled with wall laws and a two equations turbulence model. A Finite Volume method has been employed for the solution of the model.

Geometries description

As it has been said before a controllable pitch propeller of 200 mm diameter was employed for the calculations. In *Fig. 1* it can be seen the propeller geometry and in *Table 1* geometry parameters are presented.

EAR	Skew	0.7 Pitch	P/D	Pm	Pm/D	Profile
0.55	6º	200 mm	1	188.46 mm	0.9423	NACA 16

Table 1. General propeller parameters

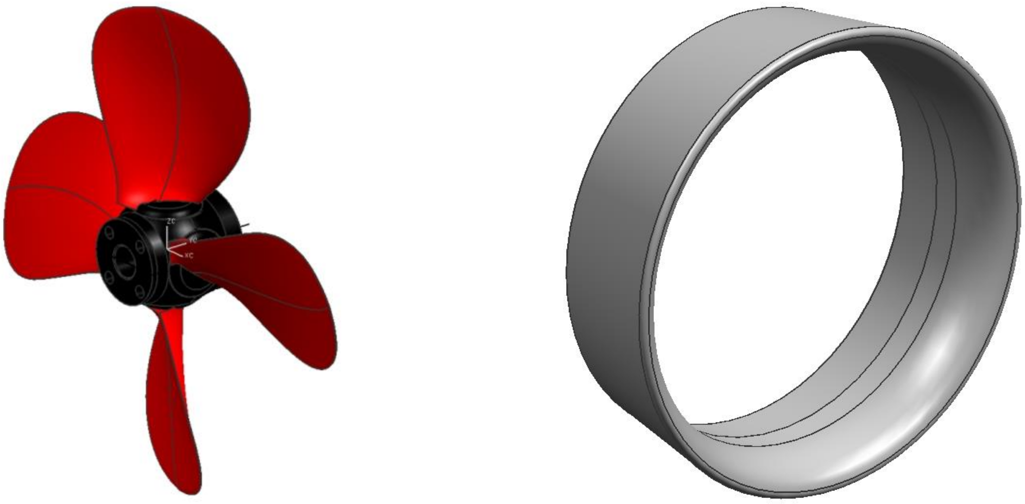

Fig. 1. Virtual propeller model

We have employed two different nozzles; both of them are 19A based but with different chord length. The first one (Nozzle 1) is 100 mm length and the second one (Nozzle 2) is 75 mm length, which corresponds with 50% and 37.5% of the propeller diameter. The internal nozzle diameter is the same for both cases, 202 mm which corresponds with a diametrical clearance of 1% of the propeller diameter.

Experimental data

The propeller model was manufactured at the Ship Design and Research Centre S.A. located in Poland (CTO). An image of the manufactured model can be seen on *Fig. 2*.

Fig. 2. Propeller model

The experiments were carried out at the CEHINAV, from the Madrid Polytechnic University in Spain; CEHINAV facilities includes a 56 m length, 3.8 m wide and 2.2 m depth tank. A calibration of the measurement instruments was carried out employing propeller nº 3297 from The National Physical Laboratory (United Kingdom). The deviation in the thrust and torque measurements were below 3% and 4% respectively. These deviations are within expectations about the usually errors in this towing tank.

Numerical simulation description

The computations were performed employing a Reynolds Averaged Navier Stokes Equations model solved in integral form and employing the Finite Volume based code Star CCM+.

As we are performing open water calculations with a uniform inflow, we have chosen a steady state temporal approximation treating the rotating propeller movement with a moving reference frame approach.

For the spatial discretization, second order schemes for both convective and viscous terms were employed and as pressure and velocities are treated in a segregated manner, the coupling between them is done by means of SIMPLE Method.

For the closure problem a two equations model were employed coupled with a wall law; for this case the k-ε turbulence model with an all Y+ wall treatment was employed.

For the domain discretization three polyhedral meshes were employed to asses the spatial convergence.

Numerical Results

The employed domain in the calculations is a cylinder which takes 5*D upstream the propeller, 10*D downstream the propeller and 5*D to the far field. Three unstructured polyhedral meshes were employed for the assessment of the spatial discretization convergence; Fig. 3 corresponds to the intermediate one (Mesh 1).

Fig. 3. Propeller mesh

Torque [N*m]	Mesh 1	Mesh 2	Iterative Error	Spatial Discretisation Error
3.01	*2.99*	*2.96*	*1.17E-06*	*0.9%*
Thrust [N]	**Mesh 1**	**Mesh 2**	**Iterative Error**	**Spatial Discretisation Error**
139.03	*133.58*	*133.77*	*6.23E-07*	*-0.1%*

Table 2. Convergence study

Before the performance of all the calculations, a spatial sensitivity analysis were carried out for J=0.2 and Nozzle 1 (*Table 2*). As we are employing unstructured grids, the mesh selection is carried out according to integral values of thrust and torque by an error. This error is computed as the difference between integral values for a mesh respect to the finest one. Theses errors estimations are shown in *Table 2* for the selected configuration. The results of spatial discretization error show us that the Mesh 1 can be used for all the simulations since the values of these ones are under 1% (we can disregard iterative errors for all cases since the magnitude order is too low).

The validation of the CFD results for the different ducted propellers against the results of towing tank test are carried out by the comparison of the different figures of merit. These figures of merit are: thrust and torque.

The validations of numerical results for Nozzle 1 are discussed in the following lines. The torque and thrust values versus the advance coefficient (J) are shown on *Fig. 4* and *5* respectively. It must be said that the presented curves for CFD calculations were interpolated from three calculation points (J=0.1, J=0.2 and J=0.5).

Agreement between the experimental data and the calculated torque is very good over the complete range of advance coefficient (the deviation is below 2% for all cases).

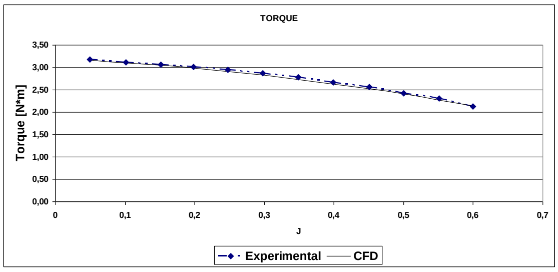

Fig. 4. Torque for Nozzle 1

The agreement between calculated and measured thrust is also good (the deviation is below 4% for all cases). Although the error for thrust is slightly higher than for torque, the curve shape is recovered.

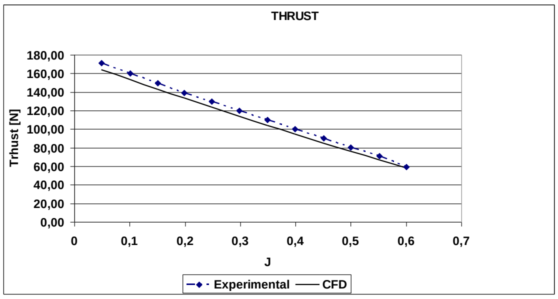

Fig. 5. Thrust for Nozzle 1

Non dimensional thrust, Kt (for propeller, nozzle and the total one), non dimensional torque, Kq, and the efficiency (η) from the CFD are visualized on *Fig. 6*. If we define a thrust distribution percentage between propeller and nozzle as Ktp/Kt, it can be seen that for low J this percentage presents values around 0.5. This means that the delivered thrust provided by nozzle and propeller is almost the same. As J increases its values our percentage value is increased too, since the Kt nozzle decrease faster than Kt propeller reaching negative values once the maximum efficiency has been reached.

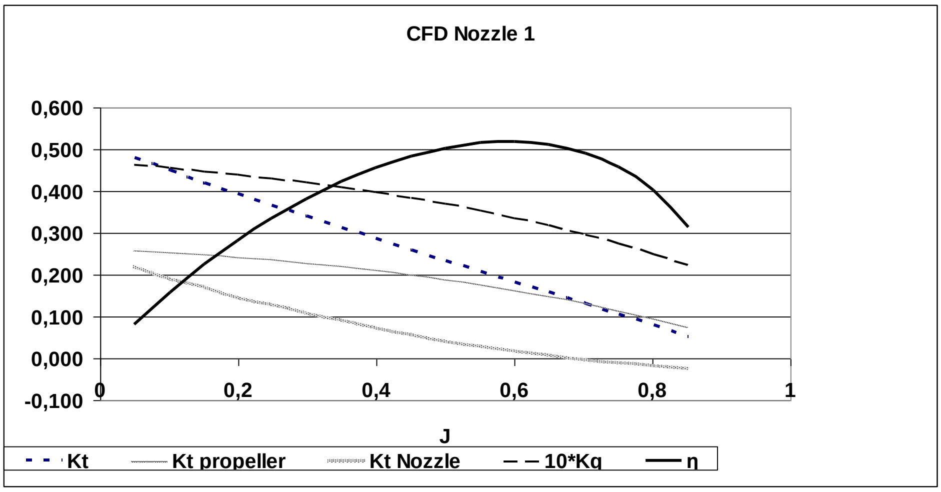

Fig. 6. CFD Results for Nozzle 1

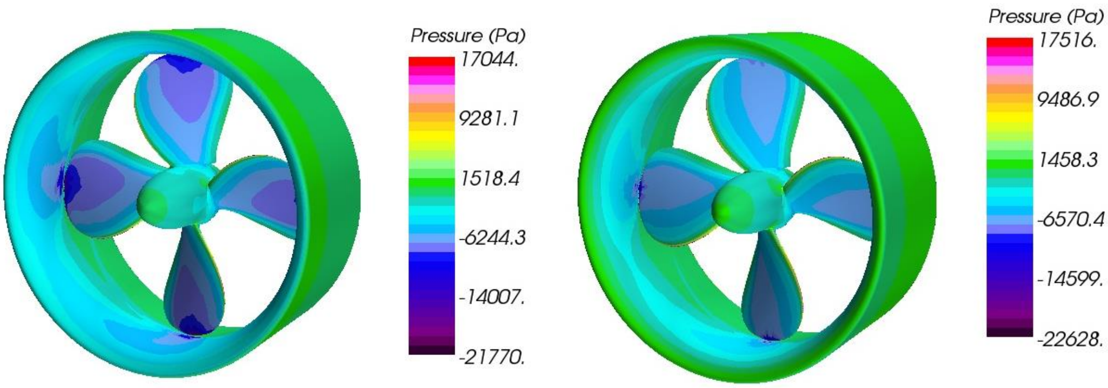

Fig. 7. Pressure distribution for J=0.2 and J=0.5

In *Fig. 7* the pressure distribution on propeller and nozzle surfaces are represented. As it can be seen for lower advance ratios the load of the propulsion system is higher (lower pressures on the suction side of the propeller and on the nozzle). It is easy to understand the importance of the gap tip length seeing the pressure distribution as it is the point of highest interaction between propeller and nozzle (a good mesh resolution is required). For lower advance ratios, the percentage of thrust from the nozzle is higher as the load of the propeller is displaced to the tip and this generates higher velocities (lower pressures) on the interior nozzle surface.

Fig. 8 and *9* represent torque and thrust calculations versus experimental data for Nozzle 2. It could be seen that discrepancies are in the same order as for Nozzle 1. *Fig. 10* represents the propulsive characteristics (Kt, 10*Kq and η) in the whole operating range for the propeller with Nozzle 2.

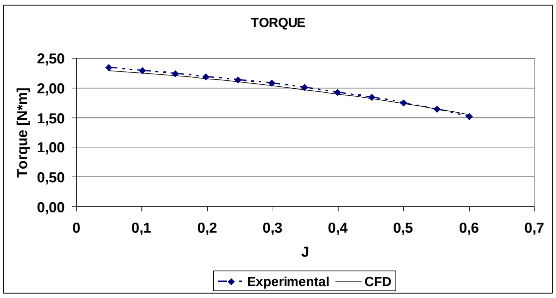

Fig. 8. Torque for Nozzle 2

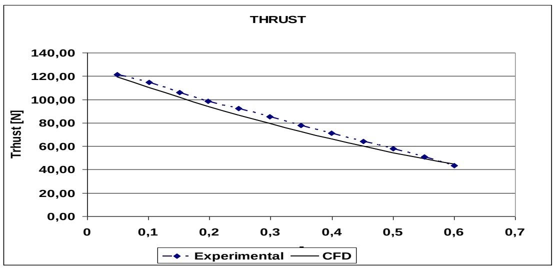

Fig. 9. Thrust for Nozzle 2

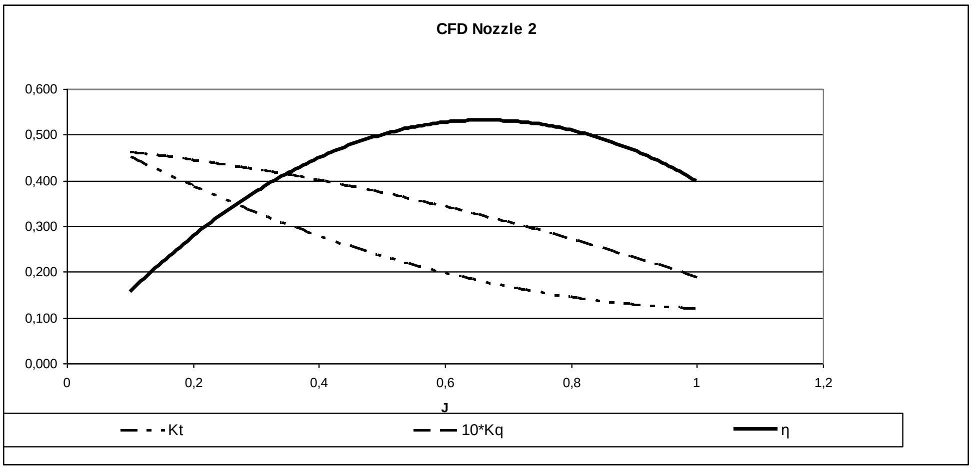

Fig. 10. CFD Results for Nozzle 2

Concluding remarks

A validation process for ducted propellers in open water condition is presented in this article. For this purpose a controllable pitch propeller was employed with two different nozzle geometries. The CFD calculations were performed employing a RANSE model solved with a Finite Volume Method. The results were compared with towing tank data and as it could be seen the agreement is good enough for design purposes.

As final conclusion it could be set that the CFD numerical model can be employed as a design tool for trawler propulsion systems.

Current and future works

The fact of locating a rudder downstream the propeller will vary the propulsive characteristics of the propulsion system [ref 2]; as a consequence the rudder must be included as an active element of the propulsion system. It is a common practice for deep sea trawlers to locate a group of three rudders with high aspect ratios downstream the propeller. At first this is a bad choice from the point of view of energy recovery for several reasons; a reduced sectional profile thickness on the blade leads to lower levels of energy recovery, furthermore, the fact of locating two of the rudders decentred from the propeller shaft results a worse working condition for rudders as the axial inflow is higher and the availability of energy recovery is lower.

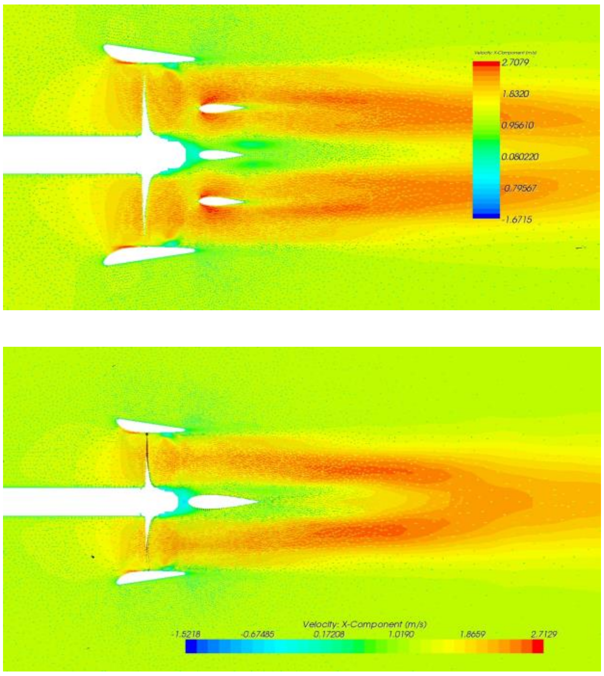

Fig. 11. Velocities profiles

	Without Rudder	Three Rudders	One Rudder
Propeller + Nozzle [N]	58.32	62.6	63.34
Propeller + Nozzle +Rudder [N]		59.4	62.36
Torque [N*m]	1.73	1.82	1.83
η	44.35%	42.81%	44.98%

Table 3. Preliminary results

In *Fig. 11* it can be seen the velocity field downstream a ducted propeller for a conventional three rudders arrangement and the same field for a single rudder. It seems, from preliminary studies, that for energy purposes it could be better to employ only one rudder aligned with the propeller hub with a higher aspect ratio than the employed one (three rudder arrangement).

The results of these preliminary calculations are presented on *Table 3*. It can be seen that for energy purposes the use of one rudder instead of three is much better and without any special modification on the rudder geometry such as Costa bulb, additional lifting surfaces, or special profile design of the rudder surface (indeed these solutions must be checked).

In addition to energy considerations, manoeuvrability aspects should be taken into account in future works as this could be an important issue for the control of the fishing operation while trawling in bad weather and reduced sailing speed.

Bibliography

[1]Carlton, J. *Marine propellers and Propulsion*. 2nd Edition. Butterworth-Heinemann 2007

[2]Sánchez Caja, A., Pylkkänen, J. V. and Spilä, Tuomas P. *Simulation of the incompressible Viscous Flow around Ducted Propellers with Rudders Using a RANSE Solver*. 27th Symposium on Naval Hydrodynamics Seoul, Korea, 5-10 October 2008

[3]Ferziger, J.H., Perić, M. *Computational Methods for fluid dynamics*. Springer. 2000

[4] Harvald, SV.AA., *Resistance and Propulsion of Ships*, Krieger Publishing Company, Malabar, Florida, 1991

[5] ITTC- *Recommended Procedures and Guidelines - CFD General Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures*

[6] User guide STAR-CCM+ (Version 4.0.6)