

Application of fully viscous CFD codes in the design of non cavitating propellers for passenger vessels

Gianpiero Lavini¹, Lorenzo Pedone¹, Davide Harpo Genuzio¹

¹MC-ARC, Naval Architecture Department, Fincantieri S.p.A., Trieste, Italy

ABSTRACT

A significant hydrodynamic energy saving on a cruise ship and an outstanding comfort on board can be achieved only improving simultaneously hull, wake and propeller design. A propeller optimisation work should therefore be carried out first acting on the wake field in which it is running. Nowadays the most sophisticated fully viscous codes are able to predict very correctly not only wave profiles and pressure distribution but also the wake field and the effective power PE both on model scale and true scale. These powerful tools have a strong impact in the hydrodynamic design as they allow to investigate a very large number of hull and appendages modifications giving the possibility to get highly optimised wake and hull lines. The performance of the vessels in terms of power absorption and wake distribution can also be known in advance before tank testing. Fincantieri design dept. is widely applying this state of the art technology in the design of hull and propeller for commercial passenger ships. This paper describes the application of fully viscous CFD tools in the joined design of hull wake and propeller in order to get an efficient propeller free from cavitation and inducing very low pressure pulses. This proposed design method has been adopted in the design of three different new prototypes of passenger ship which proved after tank testing outstanding performance in terms of propeller excitation.

Keywords

Propeller design, Ranse, wake field, twin screw, pressure pulses

1 INTRODUCTION

The standard propeller design method consists in considering some input parameters as power thrust, speed and wake and developing a blade geometry which fulfils the specification requirement concerning the efficiency and the pressure pulses. These input parameters are considered fixed and the propeller designer has just to set the blade geometry characteristics to achieve the requested specification performance. Among the mentioned input data it should be emphasized that the wake profile plays a fundamental role in getting a successful propeller as far as cavitation, vibration and efficiency are concerned. The proposed design

procedure to get a non cavitating propeller for a twin screw ship does not consider the wake as a fixed input data, but as an integral part of the propeller design loop. The wake optimisation integrated with the blade design represents in fact a decisive factor to achieve the goal to cancel the cavitation from the blade surface. If cavitation is noticed on the blade surface action can be taken either changing the blade geometry or improving the wake profile to cancel it. Hull and propeller design are therefore not two separate jobs to be carried out one after the other but an integrated activity in which the attention of the designers is therefore concentrated not only on the blade geometry development but should be extended also to the optimization of the wake field in which the propeller is rotating. It is understood that hull and propeller design must be carried out in parallel and not in sequence as it happens nowadays in most of the cases when the blade design is started once the hull lines are already finalized and steel production is underway in the ship yard.

2 WAKE-PROPELLER DESIGN APPROACH

In a standard design sailing condition at 85% MCR practically all propellers fitted to a conventional merchant twin screw vessels are affected by some cavitation. The propeller designer has at his disposal many tools to reduce the amount of cavities on the propeller surface, as for instance changing the rpm, reducing the tip blade loading, increasing the blade area ratio or the number of blades. As a consequence all these actions can imply a reduction in efficiency. It is well known to all propeller designers that the main source of cavitation is the wake field in the propeller disk. In most of the cases the designer has to deal with a wake which has been already defined and no actions can be taken for some further improvement because when the propeller design is started the ship hull is already in production and no modifications can be applied. Starting from these preliminary remarks it can be seen how much important is to carry out the hull and the propeller design simultaneously in order to apply all requested modification to the hull and appendages geometry in a preliminary stage till an optimum wakepropeller combination is achieved. Until few years ago this procedure could not be

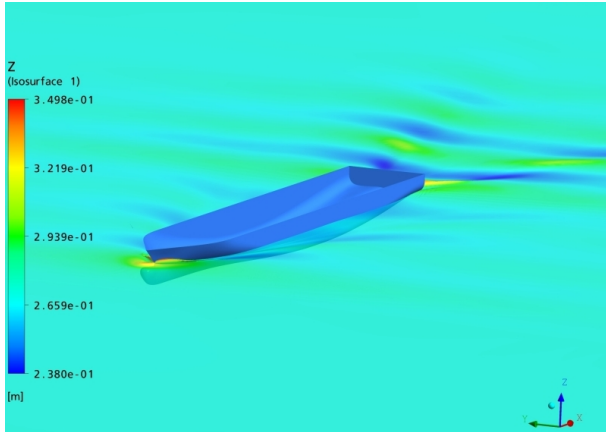


Figure 1: Free surface simulation

followed as the wake analysis was just a final measurement after completion of tank testing and very seldom a wake optimisation process was carried out after that tank tests were completed proving that the speed contractual commitment was achieved. Nowadays new very powerful CFD tools are available to measure the wake contour allowing to examine a great many hull and appendages configuration till an optimum and satisfactory wake profile is found. It is to be noticed that after a lot of comparisons between CFD calculations and wake and effective power measurement carried out in the model tank, it has been realised that the new fully RANSE code are able to predict very correctly the wake distribution and the effective power PE. Therefore in the design of a non cavitating propeller for a twin screw ship the CFD tools can be used in the design loop at the same time both in optimizing the wake field and the blade design without any preliminary data from tank test.

3 WAKE FIELD

Wake is very strongly related to the global hull form and small modification in the stern area can very difficultly change the wake contour on the propeller disk. Therefore one of the first items to be considered to get an optimum wake is to investigate the influence of the main hull parameters on the wake pattern. Most of them, as length, breadth, depth, displacement are normally set by the ship designer in order to fulfill the specification requirement concerning dead load, stability and general arrangement spaces, but a proper selection of the position of center of gravity fitting engine rooms, fuel and fresh water tanks in the most convenient position can yield a forward shifting of the center of buoyancy. Extensive CFD analysis performed varying systematically the position of the center of buoyancy on some twin screw vessels have proved that it has large benefits on the propeller incoming wake. The other important point is where the designer should locate the propeller disk. Till the introduction of the new CFD software the propeller position was selected according to the machinery and shaft line arrangements considering the limits

imposed by the shaft lengths and bearing positions. The possibility given by the viscous CFD calculation to estimate in the most correct way the wake speed all around the hull can allow the naval architect to evaluate which are the most convenient positions to locate the propeller in areas free from vortices and where favorable pressure gradients are present. One very helpful action in this direction is to move the engine position forward as far as possible. Once a favourable area has been selected, efforts have to be made in order to see if some local suitable hull modification can produce further improvements. As a general rule which well confirms the good practice and experience the propellers should be fitted as much as possible far from boundary layer and in an area where the water is decelerating as close as possible to the transom. Apart from the case of pods, the propeller is moved by a long shaft line which should be designed in such a way to minimise the shadow of the bossings, stern tubes and supporting brackets. As a good rule, a success in each design field is not achieved by means of a single action but is due to the sum of various factors, each giving a positive contribution. In the progress of the actual CFD work we became aware that an outstanding wake improvement could be attained acting simultaneously on many various items. In order to reduce as far as possible the shadows coming from the appendages the most effective actions was to adopt the following actions: cancellation of the hull bossing substituted by the installation of longer shaft with an intermediate T support bracket, selection of an L main bracket support instead of V brackets, shaft lubrication in water, selection of a low inclination of the shaft line in the longitudinal and horizontal plane, bossing and propeller hub shaped with good hydrodynamic profiles, adoption of twisted brackets. The improvement obtained separately by acting on each listed single item could be not very remarkable but the total superimposed effect give a surprising good result.

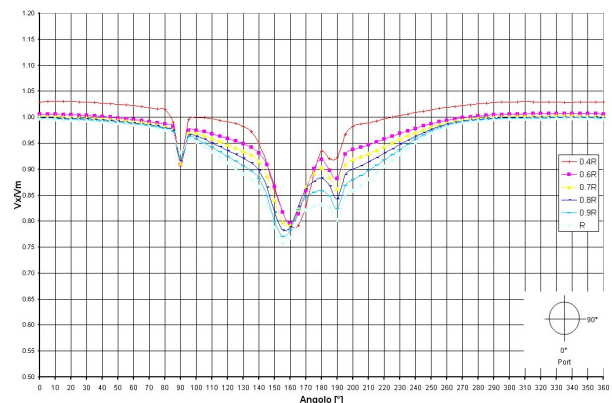


Figure 2: Numerical axial wake

4 NUMERICAL ANALYSIS

To perform the numerical wake optimisation work the Ansys CFX code has been adopted.

4.1 Wake analysis with free surface

Usually, during design process, the first simulation has to take into account the free surface in order to get a correct effective power prediction (See Figure 1). Ansys CFX allows performing multiphase simulation using a homogeneous eulerian model based on interface capturing method with VOF approach, so both water and air have to be modeled. The domain around the hull is meshed by a fully hexa grid using ICEM CFD. Normally for this type of simulation 6 mln of cells are used. A large portion of these cells are used to describe adequately free surface and boundary layer. The distribution near the hull is configured to obtain Y^+ values between 1 and 20. In order to eliminate disturbances on free surface due to boundary condition on inlet and outlet, the domain has to be quite long and to limit aspect ratio the use of longitudinal extended cells far away from the hull is not possible; also for this reason number of elements rises easily. The convergence of the computation is strictly related to the quality of the mesh in terms of determinant, angle and aspect ratio; much more in case of simulations with free surface. Modelling turbulence and wall treatment has always been some of the greatest challenge in cfd field, and this is even truer in naval application due to high Reynolds number. In every day use, the SST model coupled with the automatic wall treatment has proved to be the best compromise between results and robustness. This last point can be fulfilled only for lower Reynolds number, obtained with a simulation in model scale. Results obtained by simulation have been extensively compared with tank tests experimental data in terms of integral quantities giving very good responses; discrepancies in effective power are now lower than 2%. Comparisons on wave cuts obtained with potential codes have shown good agreements also quite far from the hull even if the code introduces numerical diffusion; this phenomenon can be controlled using a second order advection scheme and a well distributed grid.

4.2 Wake analysis with double model

To check the effects of hull and appendages modification on the underwater velocity profile is not required to perform expensive simulation with free surface. Using a simpler single phase simulation, called also double model simulation, it is possible to save cells and cpu resources and obtain a solution in a very short time. In this case the upper boundary is positioned at an appropriate draft and characterized by a symmetry condition. The position of this plane should be as near as possible to the dynamic draft of the ship; however typical bow and stern shapes of a cruise vessel impose a higher position to avoid problems with the mesh around transom and bulb. For wake survey purpose the introduction of the appendages is fundamental;

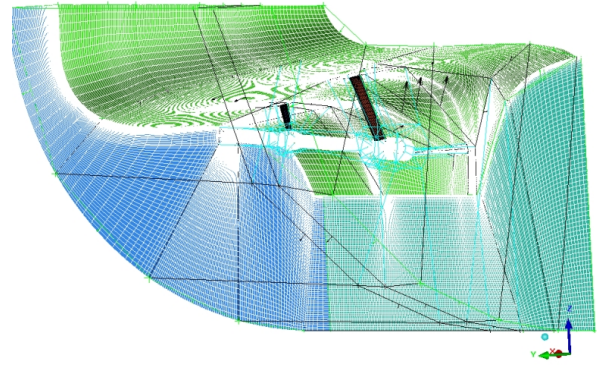


Figure 3: Mesh around appendages

this creates some new items in meshing process. To obtain a good quality mesh is possible to use the General Grid Interface connection capabilities (GGI) of Ansys CFX. This option allows the solver to solve an un-connected multi-domain; in this way it is possible to prepare two different meshes, one that contains the shaft line and brackets and the other all the rest of the hull, keeping the block structure around the appendages clear and editable. Interfaces where the solver interpolates values do not introduce any numerical disturbances. The simulation process during hydrodynamic design can be splitted into two different phases. The first phase aims to find the correct orientation of struts and brackets so the hull and the shaft line are modelled. During the post processing it is possible to check the orientation of velocity vectors finding the correct position of shaft brackets. At the same time the shape and magnitude of the shaft wake can be checked. This is a fundamental information to avoid overlap between wakes from different elements. The simulation can take into account the accelerating effect of the propeller using an actuator disc. Also the effect of the rotation of a wet shaft can be taken into account using a rotating boundary condition. This task can help to check the effect of rotational velocity imposed by the shaft boundary layer on the wake in propeller plane. Once the correct position and orientation of shaft, brackets and struts has been found, a fully appended model can be meshed. With GGI capabilities it is possible to use hexa elements both around the hull and around appendages. The global number of elements is 2mln (Figure 3). From the post processing it is possible to gather a lot of useful information that were hidden to the designer up to now. For example it is possible to visualize streamline from appendages and control the quality of the flow that crosses the propeller plane. From C_p plots on struts the designer can check the correct orientation looking at the position of any stagnation point (Figure 5). The viscous CFD application has yielded in fact the possibility to visualize under water each single detail of the shaft line and the interaction between different elements. Any indication of high or low pressure areas can be

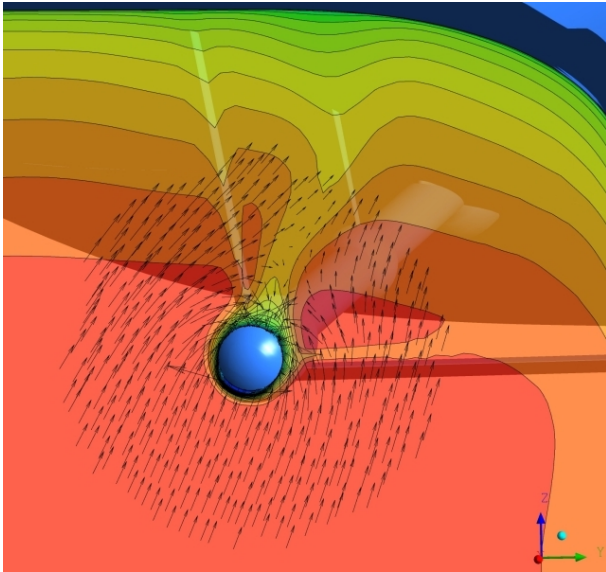


Figure 4: Nominal wake in propeller plane

detected and alternative shapes or different orientation of each element constituting the shaft line can be improved till a very smooth speed and pressure distribution is achieved all around the shaft. Viscous CFD calculations showed that one significant parameter which proved to be very successful is the adoption of the water lubrication which has the effect to strongly reduce the shadow produced by the shaft and to transmit to the water flow a favorable rotation before entering the propeller disk. In addition adopting shaft lines with very low inclination in the longitudinal plane and inclined following the stern water flow in the horizontal plane produced a strong reduction of the tangential components on the propeller disk. It has been seen that wake fields with the highest axial peak $1-w$ in the range of 0.75-0.8 (see attached wake in Figure 2) with very gentle gradients were obtainable allowing the propeller to rotate in an optimum environment. At the same time the highest tangential components can be less than 0.1. It has been proved that with a proper hull and appendages study this goal can be achieved not only on a fast and slender hull form but also on a low speed and full ship. Till some year ago when the CFD wake optimisation technique were not available these results were not easy to be achieved mainly because tank testing was the only tool to estimate correctly the wake and because large model modification to improve the wake could not be performed due to the involved costs and time schedule for production.

5 PROPELLER DESIGN

The philosophy to design a cavitation free propeller in a design sailing condition delivering 85% of MCR starts from the first goal to achieve a wake peak $1-w$ higher than 0.75 and tangential components less than 0.1 on the whole propeller disk. In the proposed design loop after completing a preliminary blade design action can be taken for a fur-

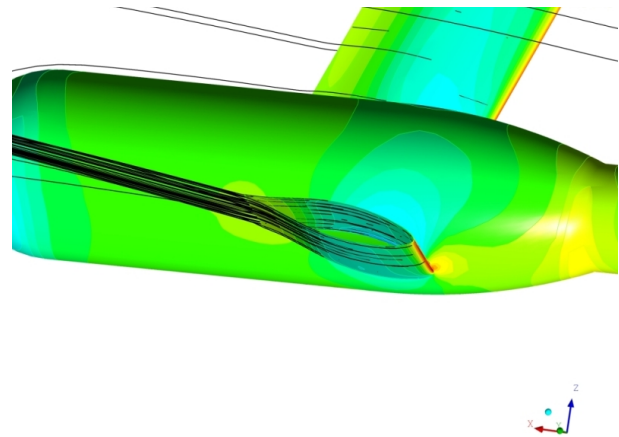


Figure 5: streamline around horizontal strut

ther wake improvement in case traces of cavitation are still present. After that the first goal to get the desired wake values has been achieved, the blade design can be started following a usual calculation loop to estimate in sequence:

1. Diameter
2. Number of blades
3. Area ratio
4. Rpm
5. Chord, Skew, Thickness, Rake, Pitch and camber distribution

5.1 Diameter

As a good design practice once the propeller position has been selected the propeller diameter has to be chosen in order to provide a tip clearance higher than 30% D

5.2 Number of blades

In order to achieve a cavitation free propeller for a high comfort cruise vessels the selection of the number of blade is usually limited to 5 or 6 blades. The solution adopting 4 blades can be taken into account only if the following criteria related to the specific load per blade are fulfilled.

5.3 Area ratio

A preliminary good selection of the blade area and the number of blades can be done according to a simple formula depending on the power density delivered by each blade. The propeller power in kW delivered by a single blade should be less than 700 kW/m². This simple assumption proved to be very effective in starting a propeller design free from sheet cavitation and tip vortex in model scale. In the design of a cavitation free propeller the selection the correct rpm holds a paramount importance. As a matter of fact, besides the wake field, the pressure pulses level is strongly depending on the rpm selection. The pressure pulses exerted by the propeller on the hull are given by the modification of the pressure field caused by the blade

displacement and the development and collapse of cavitation bubbles on the blade surface and in the slip stream. The pressure pulses caused by these two different phenomena are transmitted through the water to the hull producing and exciting fluctuating force. The hydrodynamic propeller action can be therefore theoretically subdivided in two combined contributions, the first due to the blade displacement effect (rotation and thickness effect) and the second related to cavitation depending on the blade load and the cavitation number. If the rpm are reduced by increasing the pitch, the propeller load and the non-cavitating contribution are consequently increased (and viceversa). As far as the cavitating contribution is concerned this consideration is not so easy. A RPM reduction increases the value and the margin against cavitation, but the consequence is a higher loaded blade with deeper and wider low pressure area on the blade suction side, increasing therefore the risk of cavitation. Starting from a preliminary blade design the amount of cavitation must be evaluated considering different pitch-RPM configurations. Pressure pulses due to propeller displacement and cavitation should be calculated as well considering also with care the pressures gradient distribution calculations along the sections profiles. It has been noticed that dealing with a very uniform wake with low tangential components and a suitable load distribution rpm can be selected in such a way to cancel the leading edge sheet cavitation.

5.4 Skew

It is well known that skew is an effective way to reduce pressure pulses. Instead of skew, it is more correct to talk about the sweep of the leading edge which should be suitably matched to the wake peak it has to enter. The wake peak should be carefully checked to optimize the sweep shape of the leading edge. It has to be mentioned that dealing with an FPP the skew angle is limited to a maximum angle of about 35 deg for strength reasons. However even with this limitation, the wake peak is so less pronounced and extended that high skew values are useless. The skew influence on pressure pulses can be more correctly predicted in the design condition through 3D potential codes calculations. By means of these codes it is possible to highlight the three dimensional flow behaviour on the blade and the pressure distribution field surrounding it and to apply consequently suitable skew and pitch adjustment to reduce the tip depressurized areas where cavitation can arise.

5.5 Rake

Rake is applied in combination with skew and pitch in order to provide the blade with the characteristic spoon shape producing at the same time two beneficial effects: it produces a stiffening of the blade structure as its curvature yields a very strong shape and at the same time the tip bending behaves as an energy recovery device. The tip spoon shape and the flow field around it has to be optimised only

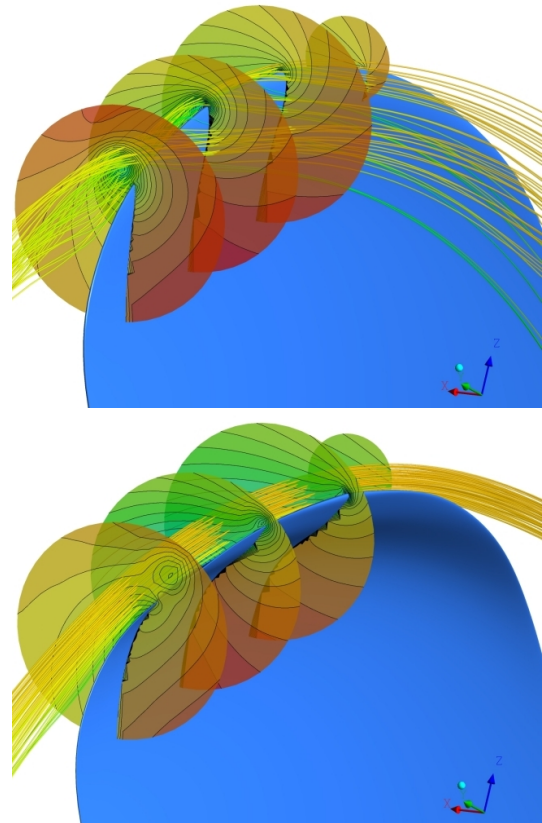


Figure 6: Tip vortex configuration

through Rans codes. As it can be seen a suitable definition of the spoon shape can yield in the end to the cancellation of the tip vortex roll up and consequently to reduce the danger of tip vortex cavitation.

5.6 Computational tools

In the process of blade designing, three different tools are adopted. In the first stage a conventional two dimensional lifting surface program is used. Such tool can correctly predict the cavitation and the pressure pulses for a conventional propeller. But in the case where 3D effects are getting important, as in the case of propeller with unconventional form like the spoon tip propeller where the combination of pitch skew and rake produce a significant curvature to the blade surface, the lifting surface codes do not calculate accurately the flow behaviour especially around the blade tip. All the information related to the tip vortex can not be computed correctly. Consequently additional calculations have to be carried out with new state of the art tools like potential panel codes and fully viscous Rans codes. In a second design stage the potential panel codes are adopted to get a more accurate calculation concerning pressure fluctuation on the hull, pressure fields on the blade surface and cavitation predictions. Compared to the lifting surface programs these new codes can give good information concerning the 3D flow behaviour around the tip area. However these codes have some limitation as they are not

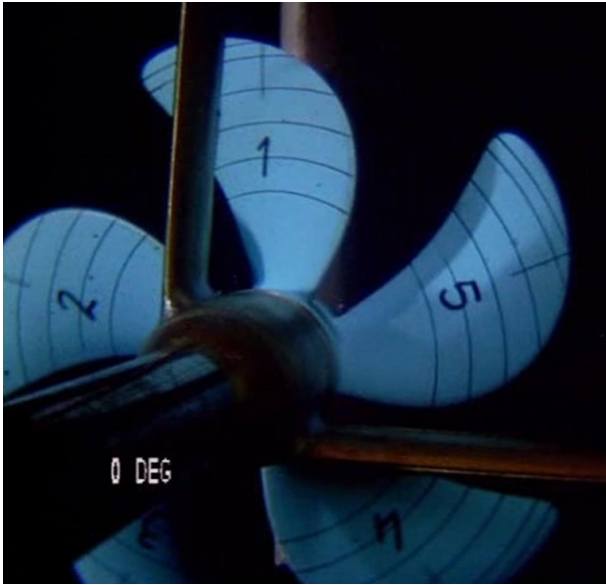


Figure 7: Tank test, design condition

able to simulate the complex tip vortex behavior and seem to be less accurate in off design condition where the viscous effects are getting more significant. These codes have the advantage to be very quick to be used and important information concerning pressures and cavitation can be obtained in a very short time. In a last stage when the blade geometry is almost finalized showing the absence of sheet cavitation, fully Rans code calculations are run. This code requires more time for the blade meshing definition and therefore can be used just for the last finishing touch to the blade geometry especially to define the tip spoon shape suitable to cancel the tip vortex. Pressures and speeds distribution can be predicted taking into account all the free 3D effects related to viscosity including the estimation of the speed around the tip vortex. Calculation can not be performed in wake yet but calculations are made at different J values relevant to various $1-w$ values. In addition cavities estimation is not used yet as a standard tool in the design. Even with these limitations it has been seen that a lot of useful information can be derived concerning the pressure distribution on the blade tip area and in the simulation of the tip vortex speed analysis (see Figure 6). The simulation approach is based on single blade methodology as used in turbo-machines simulations. The procedure for the open water simulation has been developed in collaboration with the University of Trieste, Italy. Using periodic interfaces it is possible to use a well defined mesh optimizing resources demand. Also for this problem a multi-domain is used; the inner domain (Figure 8) is the rotating one in which the blade is placed; the outer is a static and periodic domain. The used mesh is fully Hexa and it counts around 2mln of cells. Interaction and mutual effects between blades are computed even with a single blade thanks to the periodic boundary condition. This point imposes a periodic rota-

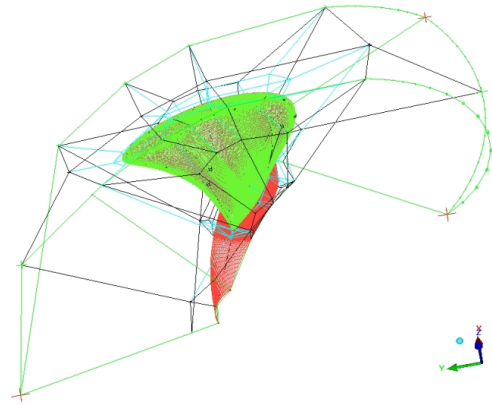


Figure 8: Propeller mesh blocking

tional domain respect propeller main axis. The angular dimension of the cylindrical sector is function of the number of blades; periodic lateral surfaces are two helicoids with constant pitch similar to the mean geometrical pitch of the blade. The use of a parametric solid modeller has proved to be very effective for the creation of the domain because it allows modifying main parameters of one of the two domains in a small time keeping mutual congruence. The mesh is obtained using extensively O-Grid capabilities of ICFM CFD. Especially in the tip region, a more defined zone is requested to represent tip vortex at best but using hexa elements is difficult to keep a good quality. Also the connection between blade and hub is a critical part for angle and determinant. For this reason also in these parts of the domain additional O-grid has to be used.

	Grt [t]	Ps [kW]	V [kn]
Ship 1	125000	44000	24
Ship 2	85000	24000	22
Ship 3	14000	4500	17

Table 1: Ship characteristics

6 APPLICATION

The described design approach has been followed in the design of three hull and propeller prototypes relevant to three passenger twin screw vessels. Relevant characteristics are attached in Table 1. As discussed the first item was to achieve with a conventional shaft line wake peaks $1-w$ higher than 0.75. The design and testing condition is always referred to a service speed at 85% MCR. The three vessels are quite different as far as displacement, main dimensions, block coefficient, power and speed are concerned. The first is a very large cruise ship fitted with a high power propulsion plant, the second is a fast medium size passenger ship and the third is a small very luxury passenger vessel. The only common element is that they have been designed adopting the same wake optimisation philosophy and the same specific load on the blades and the

Single amplitudes in kPa and phase in degrees.

Transd. no.	1. Harmonic		2. Harmonic		3. Harmonic		4. Harmonic	
	Ampl.	Phase	Ampl.	Phase	Ampl.	Phase	Ampl.	Phase
P1	0.07	56	0.03	-133	0.01	99	0.00	159
P2	0.05	-47	0.02	105	0.01	56	0.00	161
P3	0.11	12	0.03	176	0.01	97	0.00	169
P4	0.16	74	0.03	-159	0.01	106	0.00	-165
P5	0.10	-57	0.02	142	0.01	80	0.00	-161
P6	0.05	2	0.01	-175	0.01	60	0.00	-180
P7	0.21	-176	0.02	165	0.01	94	0.00	-160
P8	0.14	-4	0.04	170	0.01	111	0.01	-172
P9	0.42	87	0.05	-170	0.01	98	0.01	-148
P10	0.56	-166	0.03	143	0.01	101	0.01	-150
P11	0.29	-62	0.02	-175	0.01	93	0.00	-149
P12	0.09	9	0.02	180	0.01	88	0.00	-163
P13	0.01	-74	0.06	167	0.01	107	0.00	-146
P14	0.21	123	0.05	-179	0.01	125	0.01	-154
P15	0.31	-153	0.03	180	0.01	120	0.01	-130
P16	0.14	-62	0.03	-172	0.01	106	0.00	-132
P17	0.03	30	0.02	-170	0.01	102	0.00	-148
P18	0.08	169	0.06	174	0.02	111	0.00	155
P19	0.10	177	0.05	179	0.01	112	0.00	-159
P20	0.06	-146	0.04	-175	0.02	106	0.01	178
P21	0.01	178	0.03	-173	0.01	106	0.00	-153

Figure 9: Pressure pulses - tank results

same blade optimisation loop by means of potential and Ransce code calculation. Marin test carried out at the depressurized towing tank have well proved the expected performance. As a result the radiated pressure pulses by such propellers are extremely reduced as shown by the attached Marin test (Figure 9). Pressure values in the order of 0.5-0.6 kpa can be achieved. The efficiency at the same time can be kept at values higher than 0.7 (see Figure 10). The total integrated force is very low as well. To get an indication of the quality of the ship vibration the van de Kooy criteria can be used. This criterion takes into account the contribution of all the harmonics integrated forces and provides a reliable reference level for a good comfort on board. If the attained value is lower than that indicated by the van de Kooy criteria the ship is expected to achieve a very good comfort with low levels of vibration. For all these three vessels values ten time less than suggested by van de Kooy have measured as indicated in Table 2. This means practically total absence of vibration on board. Vibration FEM calculation performed using as input such excitation forces have given extremely low values of vibration in the stern areas just above propellers. The attached figures are showing very low pressure values (see Figure 9) only related to the displacement contribution and without any cavitating contribution. At the same time the tip vortex inception diagrams are indicating that in model scale the cavitation tip vortex have been almost suppressed.

	van de Kooy [kN]	Attained value [kN]
Ship 1	188.9	26.1
Ship 2	116.4	9.9
Ship 3	28.3	2.6

Table 2: Tank extrapolated vertical forces

7 CONCLUSION

The extensive use on cruise ships of fully viscous codes in the definition of each single detail of the hull geome-

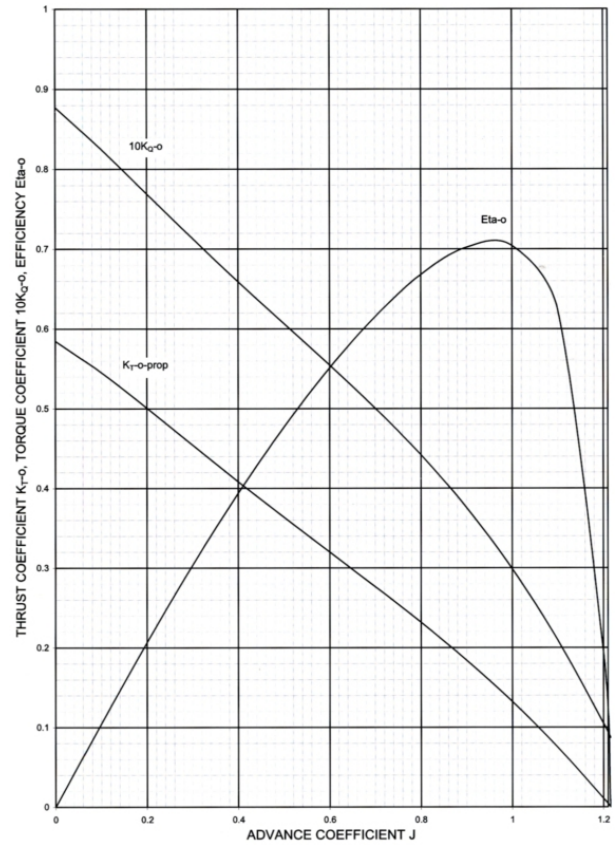


Figure 10: Open water - tank results

try, rudder, shaft, bracket and bossings, yielded significant improvements in the quality of wake profile. These CFD viscous tools were used as well to design a new type of efficient propellers suitable to operate in this optimum wake field without any kind of cavitation on the blades or cavitating tip vortex in model scale. As a result these propellers produced a large improvement of the propeller load on the hull structure. Validation tests carried out at Marin vacuum model basin showed that very low pressure levels and total integrated forces were achieved for all these passengers ships indicating also that no broad band excitation coming from the tip vortex was present. We can conclude therefore that is not necessary to penalize efficiency to get a low excitation on the ship and a consequent good comfort on board provided that hull, wake, and propeller are designed together with a proper use of CFD viscous tools. Adopting such procedure to design a propeller free from cavitation for all the considered vessels a total vertical excitation forces ten time less than suggested by the van de Kooy rule have been achieved indicating practically absence of vibration on board.

REFERENCES

Bosschers, J., Vaz, G., Starke, A., and Wijngaarden, E. V. (2008). 'computational analysis of propeller sheet cavitation and propeller-ship interaction'. In *Proceedings of Marine CFD 2008*, RINA, Southampton, UK.

Carlton, J. (2007). Marine Propellers and Propulsion.

Larsson, L., Stern, F., and Bertram, V. (2003). Benchmarking of computational fluid dynamics for ship flows: The gothenburg 2000 workshop. In Juornal of Ship Research, vol 47 No 1.

Morgut, M., Genuzio, H., and Nobile, E. (2008). Analisi idrodinamica di eliche navali in flusso indisturbato mediante ansys-cfx 11 (hydrodynamic analysis of ship propellers in open water condition using ansys-cfx 11). In Proceedings of Enginsoft Conference 2008, Venice, Italy.

Pedone, L. and Lavini, G. (September 2007). Rans codes challenge podded advantage. In The Naval Architect.

Raven, H., van der Ploeg, A., Starke, A., and Eca, L. (2008). 'towards a cfd-based prediction of ship performance - progress in predicting full-scale resistance and scale effects'. In Proceedings of Marine CFD 2008, RINA, Southampton, UK.

Senocak, I. and Iaccarino, G. (2005). Progress towards rans simulation of free-surface flow around modern ship. In Center for Turbulence Research, Annual Research Briefs 2005.