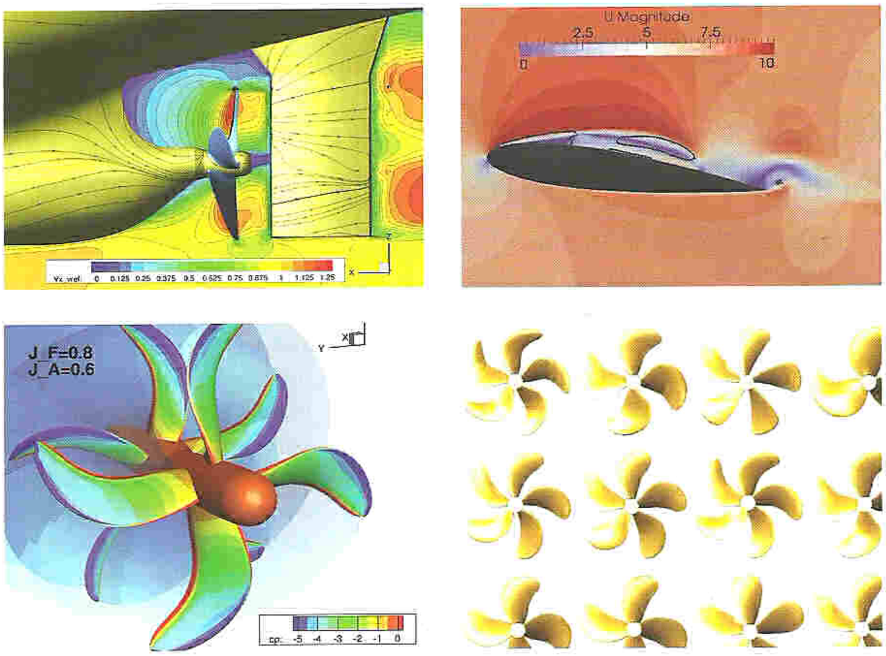
STREAMLINE realises greater efficiencies

Up to 90% of all goods handled in Europe are transported by sea and within the EU more than 40% of goods are carried by water. Increasing environmental concerns, borne out in new legislation, coupled with volatile oil prices, have created a renewed focus on fuel efficiency for the industry. Rolls Royce, SSPA, Marin and DST report

There has been little real change in the state-of-the-art for conventional screw propeller propulsion for many years with only a marginal rate of improvement during the last 50 years. More substantial progress has been achieved through the use of better propulsor configurations and improved integration of the propeller with the vessel hull hydrodynamics.

In our previous article the authors introduced STREAMLINE - STrategic REsearch for InnovAtive Marine PropuLsIoN ConcEpts (www.streamline-project.eu); a €10.9 million (US$14.39 million) programme led by Rolls-Royce, focussing on propulsion. That article covered some of the technologies under development, including Large Area Propellers, a Biomechanical thruster (the Walvisstaart POD) and methods for optimising state of the art propellers.

This paper will address the remaining concepts, first by introducing the advances in CFD that have been made and then by elaborating on some of the concepts that they have been used on, including a study on contra-rotating propeller arrangements, distributed thrust for inland waterway vessels and advanced waterjets.



STREAMLINE develops improved algorithms to combine propeller-induction and hull-induced wakefield

Advanced CFD tools

In order to optimise a propulsion system to the utmost, numerical analysis tools must be capable to distinguish between small details in flow around design variants. On the other hand, tools must be capable of predicting the restrictions of a design, like cavitation extent and pressure fluctuations and of course, tools must be fast enough to use in an optimisation procedure.

The accuracy of CFD tools has been improved by enhancing their capabilities. The tools from CRNS, HSVA/TUHH and MARIN have been extended with grid adaptivity methods, and methods for modelling moving propellers. Grid adaptivity allows for automatic refinement of the calculation grid where-ever that is needed.

The criterion for refinement can for instance be local changes in the flow, local vorticity, pressure gradient, etc. Based on such a criterion, the grid is automatically refined by splitting grid cells and the calculation is continued. Modelling of propellers in CFD requires the capability to have rotating and non-rotating parts of the geometry in one calculation. This can be achieved by having two calculation domains sliding along each other, or by having to calculation domains overlap.

Both techniques have been studied by the partners; it turns out that overlapping grids may be more generic in application, but very time consuming, whereas sliding grids perform well in reasonable calculation times.

In the attempt to increase hydrodynamic efficiency, the design of a propulsion system is restricted by cavitation extent and pressure fluctuations on the hull. To this purpose, CFD tools have been revised to enhance the modelling of cavitating flows and to introduce the possibility to describe air entrainment effects, as in the case of ventilation of propeller blades. Special attention was given to the accurate modelling of mechanisms at the origin of cavitation-induced harmful effects like erosion, pressure pulses and radiated noise. The coupling of multi-phase flow modelling with compressible flow RANSE and LES solvers has been proposed by Chalmers University as the most advanced approach to numerically detect the erosion potential of cavitating flow structures.

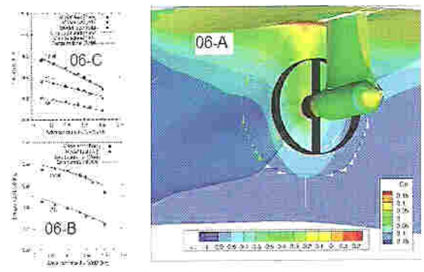
Conventional models to predict propeller-induced pressure pulses on the hull surface and noise radiated to the flow have been recast through the introduction of hydroacoustic models in which pressure fluctuations are described through wave propagation mechanisms.

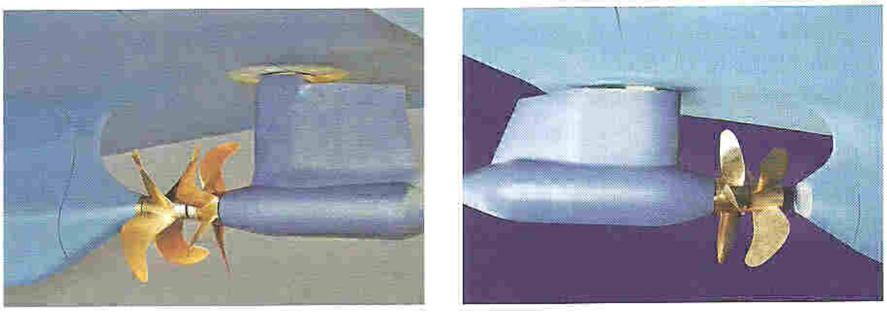
Hydroacoustics solvers based on Kirchhoff-Helmholtz and Ffowcs-Williams & Hawkings equations have been developed by MARIN and CNR-INSEAN. The full-coupling between CFD models to predict emission sources and hydroacoustics models have been accomplished and validated.

Combining the new modelling features that have been introduced, enhanced CFD methods by RANSE or LES can yield very accurate predictions of the flow around a propulsion system. However, the computational effort involved is quite large, prohibiting the use of these tools in the design process. There still is a clear need for tools based on potential flow models (typically BEM) that predict propeller flow reasonably well and are very fast. In this area, an existing BEM solver has been revised by CNR-INSEAN to improve reliability and range of applicability.

More general and fast cavitating flow and trailing vorticity models have been introduced and algorithms valid for isolated propellers have been extended to analyse complex configurations like contra-rotating propellers and podded propellers. CFD and potential flow solvers are typically applied along the design spiral through a sequential approach, in which the latter are used at early design stage, whereas CFD is applied in the last refinements steps.

Figure 1





STREAMLINE looks into the effects of CRP/ICP podded propulsion

However, a far more effective approach has been taken by MARIN, HSVA and CNR-INSEAN, which can fully exploit the complementary features of both CFD and potential flow solvers. Hybrid viscous/inviscid flow solvers have been developed in which a viscous flow solver (RANSE) is used to predict the turbulent flow around the hull with propulsion effects indirectly taken into account as forcing terms evaluated by a potential flow propeller model (BEM).

Major achievements within STREAMLINE include the development of improved algorithms to combine propeller-induction and hull-induced wakefield and the possibility to account for transient propeller flow effects. Numerical applications of hybrid RANSE/BEM solvers to determine ship propulsion factors of single screw vessels as well as to compare the performance of different contra-rotating podded propulsion layouts have been addressed.

With all the new tools and capabilities, it has become possible to optimise the propulsion system numerically. To complete an optimisation procedure, geometry variations need to be derived, and optimisation criteria need to be set. For the geometrical variations, two methods have been tested: free-form deformation and parametric variations. With free-form deformation, the geometry of a hullform or a propeller can be varied in any way the user or the optimisation procedure wants. Problems arise, of course, in limiting the variations to something that is useful and realistic. With parametric variations, existing propeller geometries are modelled with a limited set of parameters, and by varying the parameter values new geometries are derived. Of course, drawback of this method is that no revolutionary new system will be found. Both methods have been implemented by the partners, and have been challenged in the optimisation of propeller and hull aftbody of an existing state-of-art single-screw tanker design. The outcome of this optimisation study has been discussed in the previous article on STREAMLINE (Pages 28-34 Jan 2013).

Contra-Rotating Pods: CRP / ICP

Besides studying the propulsion of large propellers, present propellers and distributed propulsion, STREAMLINE also deals with podded propulsion. In this case the studies were focused on a special application of podded propulsion, namely contra-rotating podded propulsion. These studies have consisted of two main concepts (see image at top of page):

1) A pod unit placed behind a conventional main propeller called CRP (Contra Rotating Pod)
2) A pod unit with two pulling contra-rotating propellers called ICP (Integrated Contra-rotating Pod)

As a reference ship for these studies a twin screw, twin skeg, ro-ro (roll-on, roll-off) ship was selected. This ship was to a high extent optimised to achieve a very high efficiency. The ambition was also to have a safe ship with redundancy regarding propulsion therefore the twin screw arrangement. For the CRP concept, instead of two skegs, the afterbody was redesigned with one center skeg and a conventional main propeller, designed by Rolls-Royce. Also a new type of pod unit with a propeller contra-rotating to the main propeller was designed by Rolls-Royce. The ICP concept was designed by SSPA, both pod house and propellers. Here one difficult assumption was to estimate the necessary size of the pod house to have enough space for the electrical engines in full scale and necessary equipment in model scale.

Both concepts were model tested in SSPA's towing tank regarding prediction of power as well as in the cavitation tunnel to predict cavitation performance and pressure pulses. The design speed for both concepts was 20knots, a speed that should not have to create any major problems regarding cavitation. However, as always when contra-rotating systems are involved, the interaction between the two propellers can show quite special cavitation behavior, like we could see here.

For these two concepts the CRP showed a fully acceptable cavitation while the ICP were more likely to need an optimisation of propeller design, perhaps somewhat increased blade area ratio. A very low blade area ratio was chosen to result in high propeller efficiency, which in a way paid off, but the CRP design resulted anyway in the highest total efficiency. The main reason for this is that the much bigger pod house for the ICP gives some 40% higher resistance which decreases the pod efficiency significantly and thereby the total efficiency. In any case, both concepts showed significant efficiency improvements. The CRP showed a reduction in power by some 14% and the ICP 11% compared to the reference ship. The CRP can also be regarded as presenting a redundant solution.

In parallel to experimental work described above, analysis of the two contra-rotating podded propulsors by using Computational Fluid Dynamics (CFD) tools has also been carried out. The challenge here was to demonstrate the capability of the so-called hybrid RANSE/BEM hydrodynamics model (as described earlier) to provide accurate descriptions of very complex ship-propulsion solutions. In doing so this demonstrated the benefits of our cost-effective CFD simulations that are affordable at design level and do not require CFD-developer expertise and supercomputing facilities.

Though hybrid RANSE/BEM models have already been used to analyse conventional ship-propeller configurations, simulating the hydrodynamic interaction between hull and different types of contra-rotating podded propulsors represents a cutting-edge application achieved in STREAMLINE. Preliminary results from this computational study carried out by partner CNR-INSEAN have demonstrated that quantitative predictions of propulsor performance, as well as details of the flowfield, can be obtained to complement model test results and support designer's activity and decision making process (Figure 1).

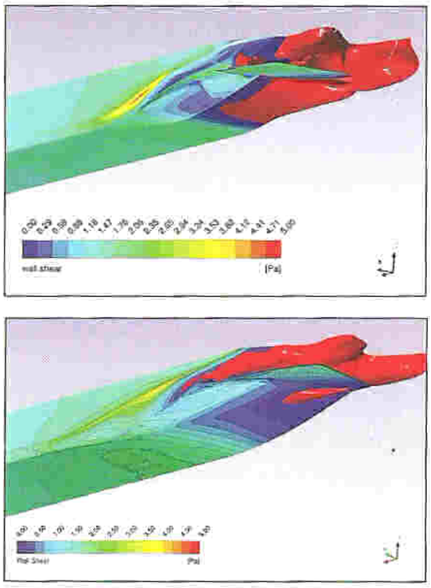


Figure 2: CFD optimisation convention code

"The STREAMLINE project is systematically tackling a full range of approaches to improving propulsive efficiency of ships"

Distributed Thrust

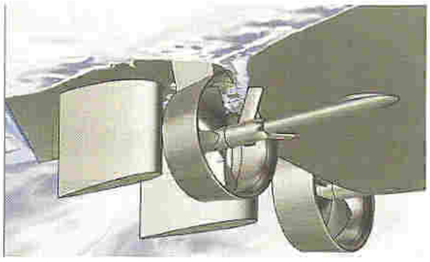
From basic momentum theory, propulsors for efficient ships should be as large as possible, since this way they can accelerate more water to a reduced velocity. However, due to spatial constraints, the actual size of a propulsor is limited. This in particular holds for inland crafts, were the restricted water depth and the unfortunate B/T ratio results in rather small possible propellers with accordingly less efficiency.

As the depth is a hard limiting factor, inland ship propellers extend close to the free water surface in order to maximise the available space. This imposes another undesired phenomenon of air suction by the propellers from the free surface. The dilemma of efficiently propelling an inland ship is thus to make the propeller as big as possible (in order to have a good efficiency) and to prevent air suction from the free surface. Sailing in ballast, were the free surface is even closer to the propeller is thus one of the limiting cases.

Inland ship designers developed special tunnels covering the propeller to prevent or delay air suction while still maintaining reasonably big propellers. STREAMLINE addresses the challenging propulsion dilemma of inland ships with both the biometric propulsion approach covered in the previous article and by the distributed thrust concept. The idea here is to cover the available main frame area of a ship by a large number of smaller propellers, thus increasing the propulsive area without excessive depth.

The resistance of a tunnel was investigated by means of model tests with comparable ships, both with and without tunnel. A resistance contribution of more than 15% of the total resistance was confirmed. Thereafter, a new aft ship design was developed and optimised by standard CFD techniques, with six instead of one or two propellers. Instead of a tunnel, a wake plate was applied that also covers and protects the propellers from air, but at the same time has less resistance and a favourable effect to flow separation (another undesired phenomenon of full inland ships that results in yet another increase of the resistance). Some results are shown in Figure 2.

The developed design also features 3% more displacement, which from a transport efficiency point of view is a nice spin-off. Two phase CFD calculations were performed firstly on the original design to check the risk of air suction and



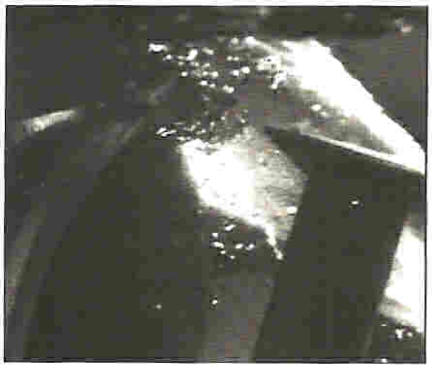


Figure 3: CFD check air suction dedicated code

to validate the code. For these calculations, the ISIS-CFD code of CNRS was extended within the STREAMLINE project to cope with such complex situations, where rotating propulsors, complex geometries and two phase flow problems simultaneously have to be dealt with. An example result of these calculations is shown in Figure 3.

Model tests were conducted in depressurised conditions to even scale down the compressibility of the air for the case that air is sucked under water by the acting propellers. The results are summarised in table 1.

The conclusion is not uniformly positive. Despite the increased displacement and an extraordinary 38% reduction in demanded thrust of the new design, the gain in power was unexpectedly small. A power reduction of 7% is still a good result however.

It has been hypothesised that the large number of rudder propellers and their corresponding drag from the housings is to blame. As the net thrust of rudder propellers is reduced by its housing's resistance, the delivered thrust of the propeller is accordingly higher. As this thrust is to be balanced by the engine's power, the net savings are accordingly reduced. It is now the plan to reduce the housing's resistance and to exploit the achieved improvements better. For this purpose, again STREAMLINE techniques partly from other WPs will be used.

Property	Change relative to baseline
Resistance	-22%
Displacement	+3%
Required thrust	-38%
Propulsive power	-7%

Table 1

Advanced waterjets

The waterjet work package is aiming at improving the low speed performance of the waterjet by reducing the losses associated with the inlet. A standard inlet is a generally a compromise between high and low speed performance. In most cases the higher speeds get priority making the inlet less suitable for lower speeds with higher losses and reduced propulsive efficiency as a consequence.

Improving the efficiency in the low speed region without compromising the high speed performance is the aim with the work carried out in the waterjet part of STREAMLINE. The work is focused on improving the performance by actively controlling the flow based on the speed of the vessel. The intention is that the active control of the flow will remove the need to compromise between a high and a low speed optimised inlet design. Instead the active control will allow for low speed inlets to operate at high speed, or vice versa, without the normally associated losses.

Conclusion

The STREAMLINE project is systematically tackling a full range of approaches to improving propulsive efficiency of ships. To date the project has shown promising results on the more radical concepts of a Large Area Propeller and biomechanical thruster and investigations have uncovered numerous interesting aspects affecting performance in distributed thrust systems and advanced waterjet designs.

The research teams have collaborated to significantly advance CFD methods and these are now being applied to taxing configurations such as the contra-rotating propeller and distributed thrust configurations. In addition, new geometric optimisation routines have already demonstrated that advanced CFD and optimisation models provide enhanced propeller design tools by introducing the capability of analysing hull/propeller/rudder interaction at early stage of design, of investigating through numerical experiments the impact of retrofitting solutions and of exploring potentially high-performance layouts from fully automated optimisation procedures. Although the project is still running, it is becoming clear that there are still good efficiency gains to be made by moving away from conventional configurations.

This article forms part of the series focussing in on projects that are part of GreenSEE Net (Green Ship Energy Efficiency Network) a gathering of EU FP7 funded projects with a common focus. The next article will describe the work of the GRIP project, which is focused on understanding energy saving devices for retro-fitting onto existing ships. *NA*

Acknowledgements

The research leading to these results has received funding from the European Union Seventh Framework Programme (FP7/2007-2013) under Grant Agreement n°233896. The editor wishes to express many thanks to all who have contributed; namely the partners in the project Management Committee and all the Work Package Leaders.

Authors:

Oliver Lane, Rolls-Royce, Programme Manager
Dr Francesco Salvatore, CNR-INSEAN
Dr Henk Prins, MARIN, Manager Research & Development
Björn Allenström, SSPA, Vice President
Dr Cornel Thill, DST, Manager Ships Hydrodynamics
Michael Forslund, Rolls-Royce, Hydrodynamic Research Centre