# POWER FROM THE SEA

## DCNS uses ANSYS simulation and services to optimize design of marine renewable energy technology.

By Vincent Melot, Thermodynamics Engineer, DCNS Group, Nantes, France

cean thermal energy conversion (OTEC) is a marine renewable energy technology. It is based on exploiting the temperature gradient in tropical seas between the surface seawater (around 25 C) and deeper waters (around 5 C at about 1,000 meters). Unlike many renewable energy technologies, OTEC produces reliable electrical power that is not affected by weather conditions. This temperature gradient produces a continuous supply of electricity. In the medium term, OTEC will help to respond to the growing need for electricity in regions located in the tropics and ensure their future energy

Reliable models can provide answers in minutes rather than days, without compromising the fidelity of results.

▲ OTEC Rankine cycle

Stagnant fluid **Under vapor velocity** **Inundation**
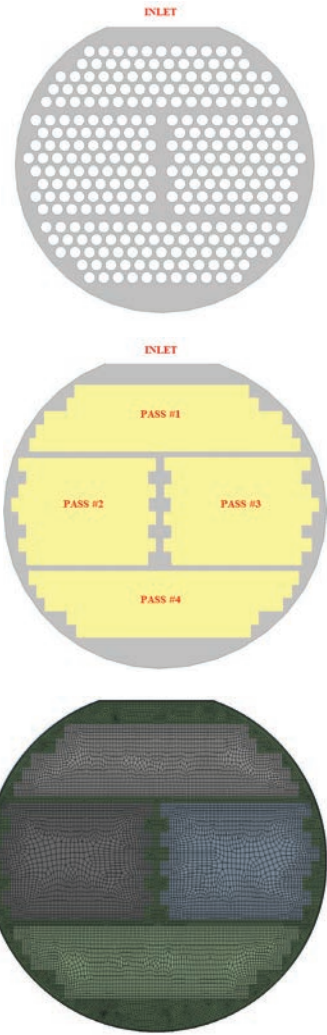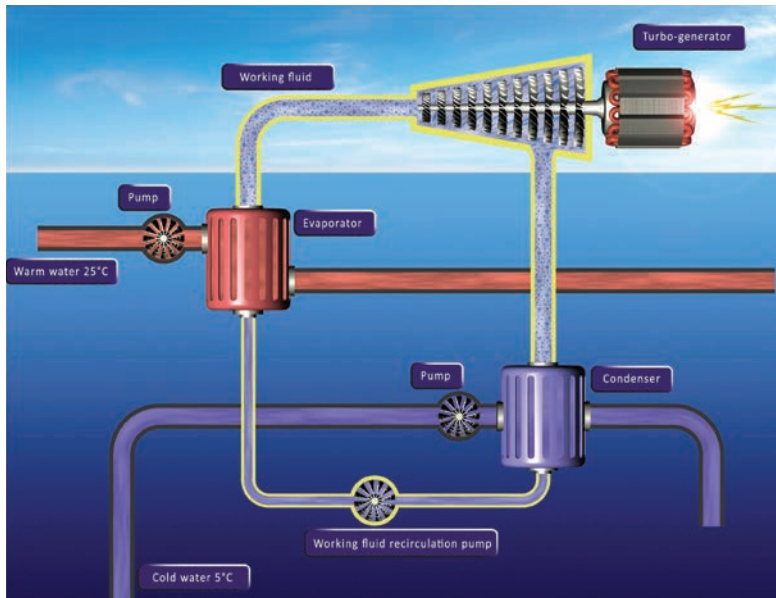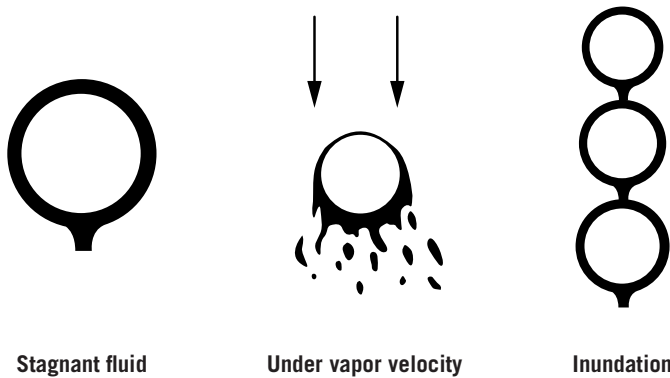
▲ Condenser phenomena must be accounted for in the design of a condenser for a Rankine cycle turbine.

▲ Tube bundle (upper), porous media model (center) and meshing (bottom)

independence. This innovative solution is a green alternative to fossil fuels, which are still used extensively on isolated sites that are not connected to continental power networks.

DCNS now offers a turnkey ocean thermal energy floating offshore solution to produce 16 MW. The Group is developing a floating OTEC power plant in Martinique, in the framework of the NEMO project (New Energy for Martinique and Overseas), under a European grant obtained in July 2014.

To facilitate design of the condensers for this plant, the DCNS research team employs ANSYS simulation software and services to speed the product development process. This installation showcases the potential of OTEC technology to benefit overseas island territories

that are not connected to the continental power grid by helping them to achieve energy self-sufficiency.

RANKINE CYCLE

With its experience in propulsion system design for submarines and surface vessels, DCNS developed an OTEC based on a traditional Rankine cycle, in which ammonia is used as the working fluid in a closed loop. The liquid ammonia enters a steam generator where it is heated by the warm water near the surface of the sea. The liquid ammonia becomes a gas by boiling at a constant pressure. The turbine requires a constant supply pressure, so heat is added to generate more vapor to replace the vapor being consumed by the turbine. The gas expands through a turbine that drives an

electrical generator, then enters a condenser where it is cooled by the cold sea water and becomes liquid again. This liquid is pumped to the steam generator and the cycle is repeated.

The condenser, which is critical to the performance of a Rankine cycle turbine, contains a large number of horizontal tubes around which the working fluid (ammonia in this case) circulates. A cooling fluid, typically cold seawater, circulates inside the tubes. Condensation of the working fluid outside the tube assembly involves several complex phenomena. Gas condenses as it contacts the tubes to create a liquid

This project made it possible to put powerful simulation tools into the hands of engineers responsible for product design and development.
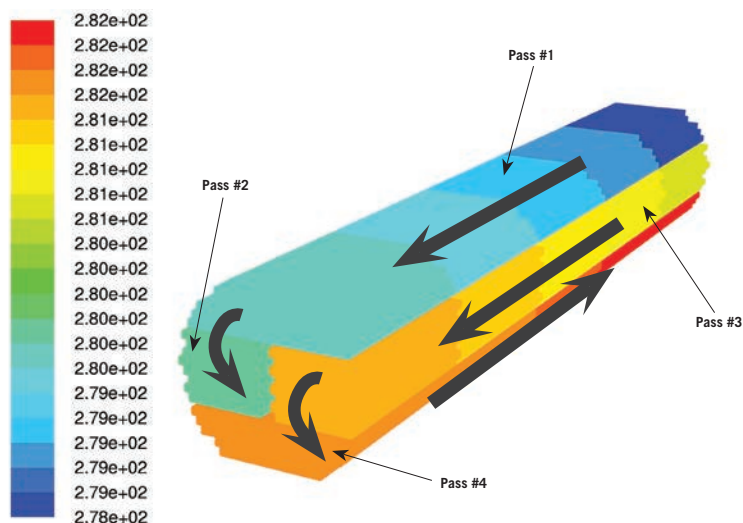
▲ Three-dimensional simulation of OTEC prototype

film. The shape of the film evolves in the tube assembly based on the surrounding gas velocity and gravity. Condensate falls from the upper tubes to the lower tubes in the bundle (inundation). Depending on the rate of condensation, the condensate may fall as droplets, columns or a continuous sheet. Increasing gas velocity agitates the film and intensifies the thermal exchange, while inundation increases the thickness of the film, reducing thermal exchanges. The goal in condenser design typically is to maximize thermal exchange to reduce the size of the tube bundle, which, in turn, reduces the size and cost of the condenser.

Designers usually develop condensers using analytical methods to create an initial design. The weakness of this approach is that analytical methods do not take the geometry of the condenser into account and are not capable of predicting local values in the heat exchanger. Therefore, a vital element of the condenser — the arrangement of the tube bundle — cannot be optimized. Physical tests normally use laboratory-scale prototypes with

hydrodynamic performance that often does not accurately scale up to production-size units.

SIMULATING THE CONDENSER

For years, DCNS has had great success with ANSYS CFD solutions for troubleshooting and investigating new designs. But to migrate simulation workflow from research to design engineering, the organization needed to develop reliable models that can provide answers in minutes, rather than days, without compromising the fidelity of results. The ANSYS Customer Excellence team in France developed simplified but reliable and comprehensive models for DCNS. These models provided a good prediction of condenser behavior

almost instantly. The collaboration was smooth and results matched expectations. DCNS engineers then developed additional models with a new approach to simulate inundation in three dimensions. This made it possible to put powerful simulation tools, normally used only by researchers, into the hands of engineers responsible for product design and development.

DCNS is improving the condenser design process using ANSYS Fluent. The condenser model depicts the tube bundle as an equivalent domain with porous media. This method enables DCNS to considerably reduce the number of mesh elements. Nevertheless, it is still necessary to complete the resolution of Navier–Stokes equations by adding source terms to correctly model all hydraulic and thermal phenomena.

The model assumes only gas phase flow; condensate is not represented. The proportion of liquid is very low in the condenser, typically just a thin layer around the tubes. However, this thin liquid film plays an important role in thermal exchange between the inner and the outer walls of the tubes. An algorithm determines the flow rate for each tube and calculates inundation of the film liquid in the tube bundle. This approach makes it possible to calculate thermal resistance caused by inundation and integrate it into the calculation of the condensation flow rate, which is based on the heat balance between the inside and outside of each tube. Thermal exchanges are determined based on an analytical correlation drawn from the literature.

The result will be a reduction in size and cost of future OTEC and heat-engine power plant installations and ship propulsion systems.

The flow rate of condensation is then represented by a term in the mass conservation equation. A source term representing gas pressure losses of the bundle is added to the momentum conservation equation.

User-defined functions calculate these two source terms: pressure drop and the rate of condensation flow. A custom user interface simplifies the process of setting up the simulation and reduces the risk of input errors. ANSYS Meshing automatically generates the mesh with tetrahedral and hexahedral elements.

The team successfully validated this simulation methodology on an OTEC condenser prototype. DCNS engineers compared the vapor mass flow rate and outlet temperature inside the tubes. CFD results agreed with experimental data.

DCNS engineers also simulated the condenser used in the steam-operated heat engine in the Coleson Cove power generating plant in Saint John, Canada. For the Coleson Cove plant, engineers compared CFD results to experimental data, including temperature at six different points. CFD results were in good agreement with experimental data, as shown in the figures.

CFD simulation makes it possible, for the first time, to accurately predict local values in the heat exchanger prior to construction of large-scale prototypes. This, in turn, enables product design engineers to optimize the arrangement of the tube bundle and improve the performance of heat exchangers much earlier in the design process.

In its next program, DCNS plans to use CFD simulation to optimize the tube bundle geometry and the inlet position to improve vapor velocity and limit inundation. The CFD model will be used to visualize these and other phenomena and optimize the design prior to building the first prototype. DCNS expects that simulation will provide a lighter, smaller and less expensive compressor design that will meet or exceed performance requirements. The customized process allows design engineers to use simulation tools to speed the design process. The result will be a reduction in size and cost of future OTEC and heat-engine power plant installations and ship propulsion systems. ∧
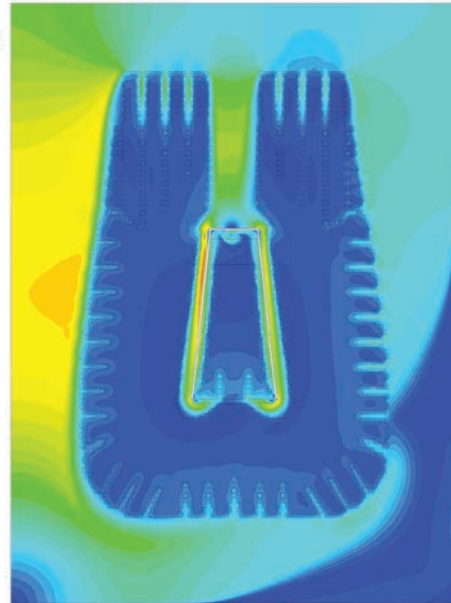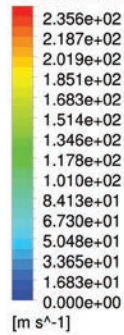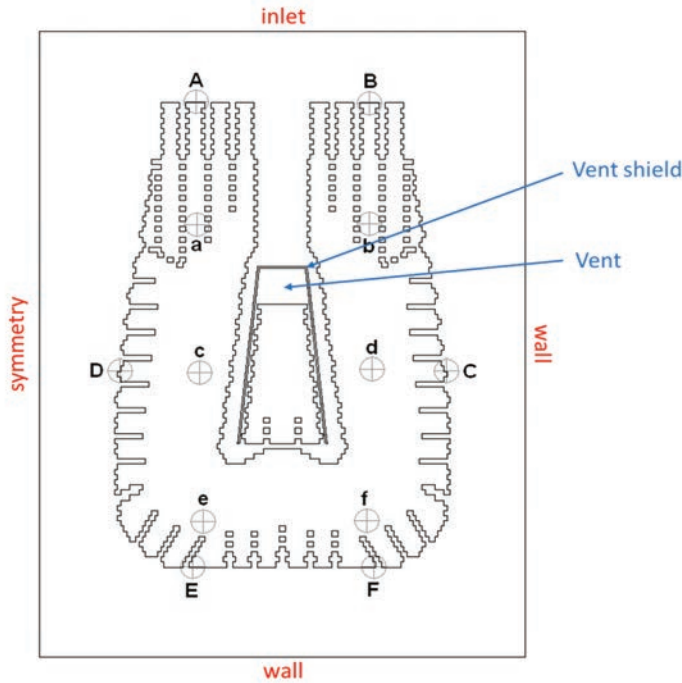
## Condensation rate
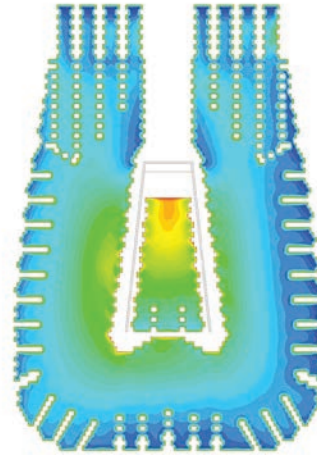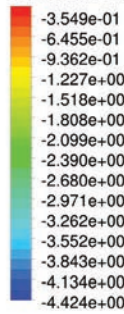
Velocity

▲ CFD results from Coleson Cove condenser

|	Outlet temperature inside tubes Absolute deviation in C	Mass flow rate shell side Relative deviation in percent
Case 1	0.18	2.1
Case 2	0.21	3.2
Case 3	0.26	5.1
Case 4	0.07	0.3
Case 5	0.01	0.9
Case 6	0.02	-0.4
Case 7	0.06	0.8

▲ Comparison between experimental data and CFD results for OTEC condenser prototype

▲ Measurement points at Coleson Cove plant
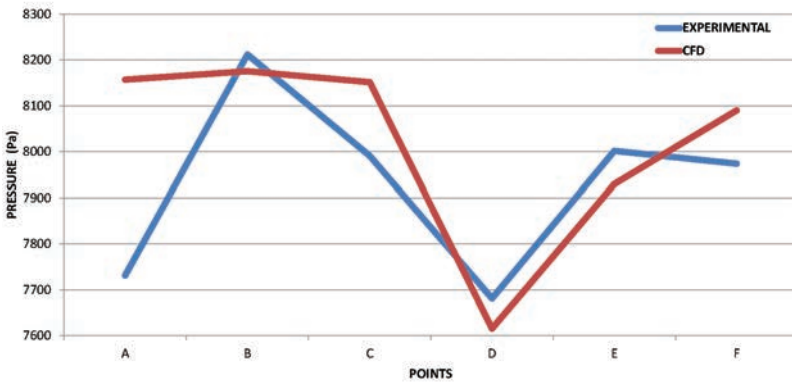
▲ Comparison of CFD and experimental pressure results for power plant at Coleson Cove
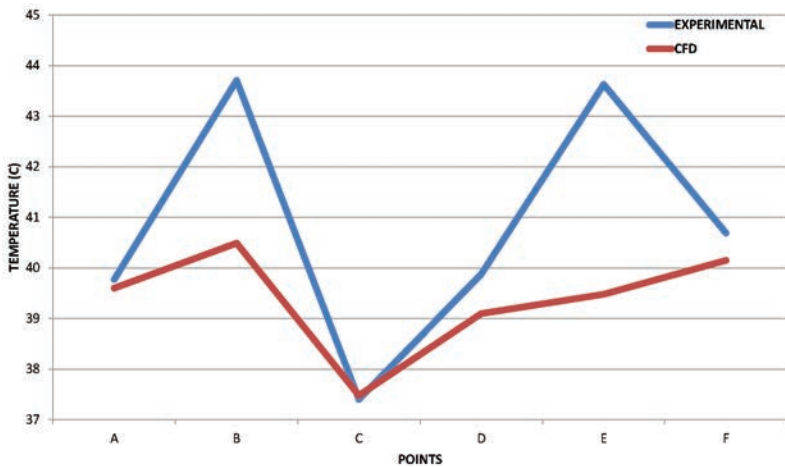
▲ Comparison of CFD and experimental temperature results for power plant at Coleson Cove