CFD study to optimize the cooling performance of a narrow specialty tractor

Numerical modelling reveals design strengths and weaknesses prior to prototype testing, saving time and money.

By Leonardo Simone Pilò
Argo Tractors

Constant demand for improved engine performance has led to an increase in the heat fluxes through vehicle cooling systems, pushing their design to the limit. In agricultural vehicles, especially in specialty tractors where the hood design is very tight, this cooling problem is compounded: the low vehicle speed, higher operating temperatures and the dirty environment all have a negative impact on cooling performance. In this technical article, a CFD study of various designs for four different specialty tractors: two brands of specialty tractors for vineyard configurations, and two brands of specialty tractors for orchard configurations, are described. The analysis evaluated the heat flow rate distribution in the hood openings, the flow over the cooling packs and their heat dissipation, as well as the flows and the temperature in the underhood and over the cabin, in order to improve the overall cooling performance and operator comfort and safety.

The study revealed the strengths and weaknesses in the original design as well as in the subsequent modifications, allowing the thermal management to be improved prior to prototype testing, speeding up the design process and saving time and money.

In recent times, vehicle design has become very challenging. Increased environmental considerations, together with the constant demand for improved engine performance, have led to an increase in the heat fluxes through the coolers, pushing the design of the coolant systems to the limit. In agricultural vehicles, and in particular in specialty tractors where the hood design is very tight, this cooling problem is compounded: the low vehicle speed, higher operating temperatures and the dirty environment all have a negative impact on cooling performance.

Solving thermal management problems is essential in the early stages of design. This is where models using computational fluid dynamics (CFD) are increasingly being used to optimize thermal performance prior to production and prototype testing. The subject of this study was a new design for a narrow specialty tractor available in four combinations: a vineyard or orchard configuration under both the Landini and the McCormick brands. The objective of the analysis was to evaluate the flow rate distribution in the hood openings, the flow over the cooling packs and their heat dissipation, as well as the flows and the temperature in the underhood and over the cabin, in order to improve the overall cooling performance and operator comfort and safety.

The automotive literature includes a lot of CFD work on optimizing underhood and front-end cooling modules, but most of these focus on the overall layout or on modifications to individual components. In this study, the entire tractor is examined in order to determine the influence on the cooling performance of both the individual components and their layout under the hood.

Methodology

In this study, a CFD model of the whole tractor was built using the commercial CFD software of the ANSYS 2019R2 suite. In particular, Space Claim Direct Modeler SCDM was used to prepare the geometry from the CAD import, Fluent was used to mesh the model, and CFX to pre-process the model, run the simulation and post-process the results.

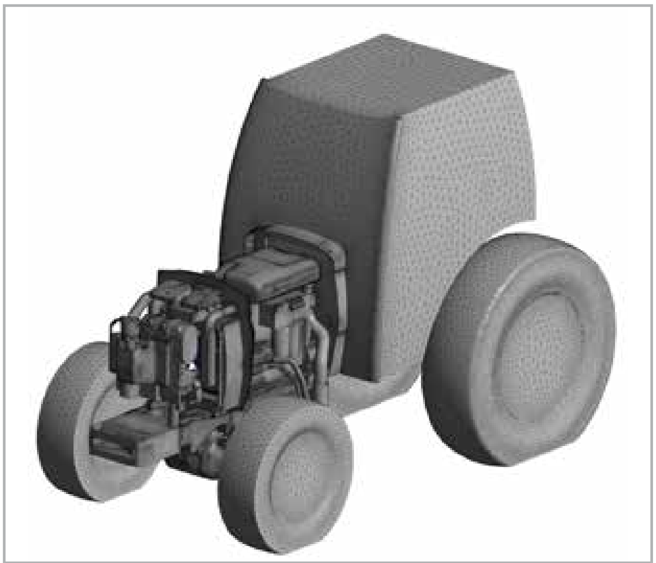


Fig. 1 - The tractor model: the whole tractor was modeled and analyzed.

System Modeling

Due to the complex geometry, some geometric simplification was applied to contain the total number of cells. The components of the under-hood compartment were all included in this study. The tractor cabin, together with the tractor axles and the wheels were also modeled. The tractor was placed inside a virtual wind tunnel box. Two different tetrahedral meshes were constructed: one for the rotating (fan) domain and one for the stationary domain, for a total of about 50 million cells.

The effect of the fan was evaluated by mean of a frozen rotor interface. This enabled the swirling effect of the fan to be considered without complicating the model. The heat exchangers were included in this study as porous media. The viscous and inertial component of the pressure drop was determined by an

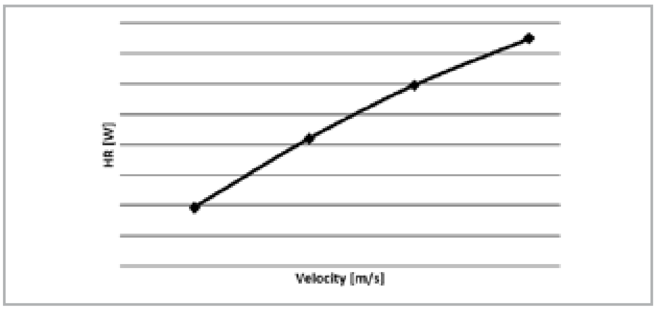


Fig. 2 - Heat Rejection curve: Heat Rejection value against mean velocity data are available for every single radiator

adaptation to the curve of the measured velocity data of the air pressure drop for the heat exchanger of interest. The thermal power exchanged was calculated for each individual radiator by multiplying the value of the Heat Rejection (HR) interpolated to the mean value of the velocity through the heat exchanger by the entering temperature difference (ETD) value calculated at the mean value of Temperature.

$$Thermal\ Power\ exchange = HR_{@mean\ velocity} * ETD$$

The ETD factor takes into consideration the reduction in the efficiency of the radiator exchange with the increase of the air temperature.

$$ETD = \frac{T_{max} - T_{averaged}}{ETD_{ref}}$$

The pressure drop, HR and ETD values have been tabulated and are available from the heat exchanger manufacturers.

The hood inlets are filled with a finely meshed metallic grill. The characteristic length of these metallic grills is smaller than the numerical mesh. The grills were therefore modelled as porous media. To evaluate their pressure drop characteristic, an auxiliary static flow simulation was performed to extract the linear and quadratic coefficients for the pressure losses.

Boundary conditions

The simulation was performed at an air temperature of 40°C. A non-slip boundary condition was applied to the lower face of the domain to represent the road surface. All the other surfaces of the

wind tunnel were modeled as opening boundaries to allow the flow to enter or exit freely from the domain. A constant pressure equal to the atmospheric pressure was applied to these faces of the computational domain.
The tractor was assumed to be stationary, the worst condition for cooling performance.

Assumptions

The model was simulated using the following assumptions:

- Steady state, incompressible flow
- Air as Ideal Gas:

$$c_p = 1004.4 \frac{JK}{kg},\ \mu = 1.83 \times 10^{-5} \frac{kg}{ms},\ K = 2.061 \times 10^{-2} \frac{W}{mK}$$

- Turbulence was modeled using Menter's shear stress transport (SST) k-ω equations
- Thermal energy exchange was modeled using total energy equilibrium equations
- Thermal radiation from the hot solid body was considered using the P1 radiation model

Simulation and analysis

The conservation equations of mass quantities, momentum, energy, and turbulence for the above model were solved in a multiple CPU cluster until the convergence standards were met. The thermal exchange in the coolant packs was automatically calculated using equations (1) and (2).

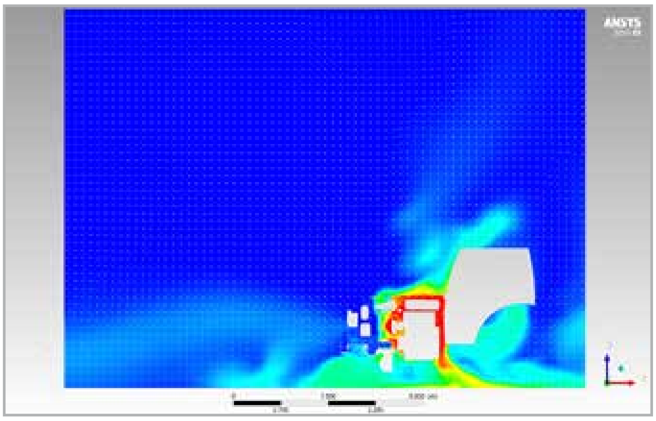


Fig. 3 - Example of results analysis: Temperature contour plot and velocity vectors projected on a longitudinal midplane.

The velocity and temperatures in the regions of interest were then analyzed.

Results

The results of the simulation are explained and analysed below, with particular attention to the area of interest to the vehicle design process.

The main topics are: distribution of mass flow between the hood inlets, distribution of mass flow between the hood outlets, recirculation of flow from the outlets to the inlets, the thermal energy exchanges and temperatures in the coolant packs, and the cabin surface temperature. These results are the most significant in terms of the thermal performance evaluation, providing

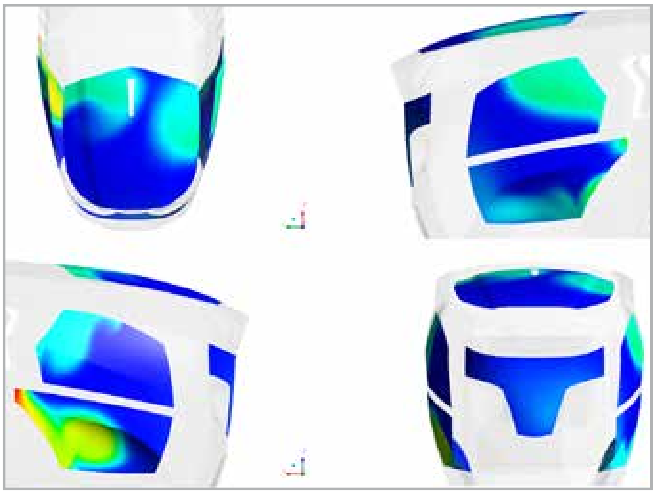


Fig. 4 - Mass flow recirculation: air temperature on the grills is highlighted. Non-blue zones are hotter than room temperature, thus indicating recirculation of hot air from the outlets

direct feedback on potential design modifications to improve performance.

The distribution of mass flow between the hood inlets or outlets, and the flow direction in these openings, provides direct feedback on the design of the openings. These openings should ensure sufficient airflow through the coolant packs to meet the objective of thermal energy exchange in the coolant packs.

The air temperature at the grills was also evaluated. The areas of the grill that are warmer than the ambient temperature indicate the recirculation of the hot air flow from the outlets. In order to maximize the performance of heat management, these areas should be kept to a minimum.

Not least, the cabin surface temperature was analysed. Hot areas here should be minimized to improve operator comfort in the cabin, which can be achieved by redirecting the hot flow from the outlets away from cabin.

The first preliminary design of the tractor hood showed several problems that could be improved. The first simulation highlighted the strengths and weaknesses of the hood design; with this in mind, four additional designs were subsequently simulated to improve the thermal management.

	Bottom Opening L	Top Opening	Bottom Opening R
2nd version	Open	Open	Open
Option 2.1	Closed	Closed	Closed
Option 2.2	Open	Closed	Closed
Option 2.3	Open	Open	Closed

Fig. 5 - Design modification: combination tested

Design modifications

The first modification of the layout introduced:

- Wider, more angulated hood outlets
- Slotted openings in the lower part
- A rectangular opening at the top

This design ("second version", below), as we will see later, aggravated the problem of recirculation, so the new upper and lower openings were selectively closed, as shown in the table below.

	Avg. Temperature@ Grills	Power @ Radiator	MFR @ Left Exits	MFR @ Right Exits	MFR @ Left Opening	MFR @ Top Opening	MFR @ Right Opening	Total MFR @ Exits
1st Design (reference)	REF	REF	52%	48%	-	-	-	100%
2nd Design	11,74%	-5,02%	84%	76%	28%	20%	20%	228%
Option 2.1	2,26%	3,49%	112%	88%	-	-	-	200%
Option 2.2	-0,90%	3,77%	100%	76%	28%	-	-	204%
Option 2.3	-4,74%	7,95%	80%	76%	28%	20%	-	204%

These new layouts were modelled and simulated as mentioned previously. The results were then analysed and discussed, following the methodology of the first design. The results are summarized in the following table.

The second version, as expected, shows a significant increase in the mass flow evacuated from the outlets (all values are percentages of the total mass flow rate from the outlets of the first design). This improvement in flow evacuation, however, had no impact on the thermal power exchanged in the water radiator; the power actually worsened by 5%.

In order to evaluate the best trade-off between flow ejection and recirculation reduction, the three options mentioned above were analysed.

An improvement in the average grill temperature shows a good correlation with the thermal power exchange performance, being the most important factor in thermal management. Option 2.3 was therefore confirmed as the best hood design.

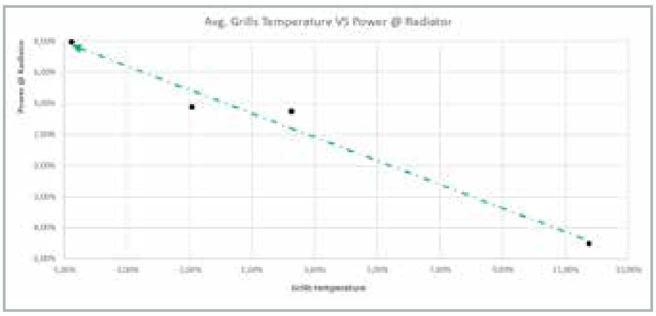


Fig. 6 - Average grill temperature vs. energy exchanged in the radiator: dotted black line shows the linear interpolation and correlation value

A close correlation between the average grill temperature (and thus the flow recirculation) and the thermal performance of the coolers is an important proof of this study. Similarly, other synthetic values can be discovered to describe the overall suitability of each design and address further performance improvements.

Conclusions

CFD simulation is an important tool for evaluating and optimizing the preliminary designs of a tractor hood to improve the performance of thermal management. In this study, the CFD analysis revealed strengths and weaknesses in the design and was used to analyze the subsequent modifications.

These modifications enabled the thermal management to be improved prior to prototype testing, speeding up the design process and saving time and money. In addition, the analysis highlighted the most impactful parameters with respect to thermal performance, assisting with future layout decisions.

As a further step, this study will be validated against real case tests as soon as the first tractor prototype is produced.

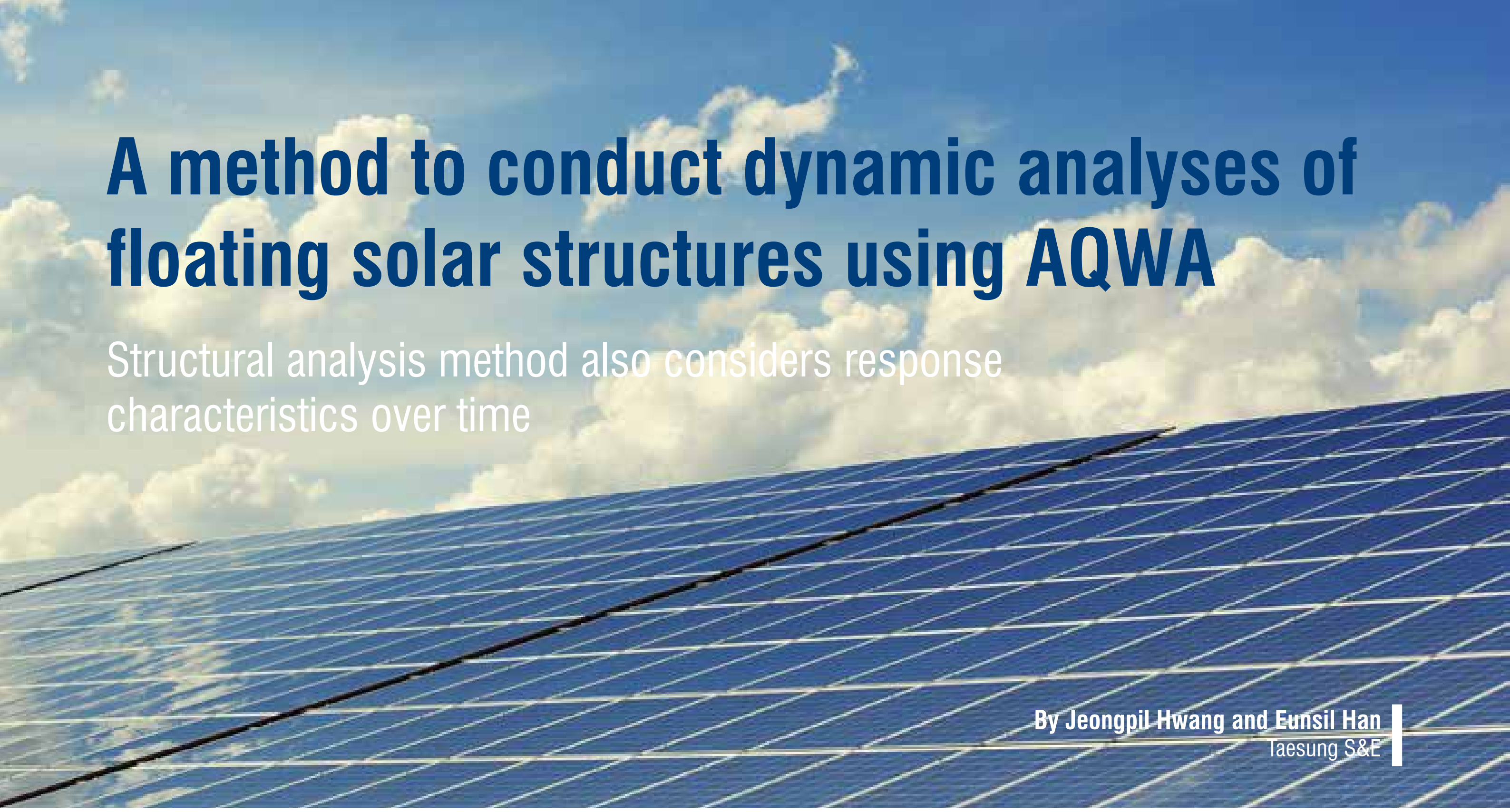
Acknowledgements

The author would like to thank the ArgoTractors' Engineering Department for their support and permission to publish this paper. The valuable contribution and collaboration of the engineer Diana Magnabosco from EnginSoft is also gratefully acknowledged.

For more information
Diana Magnabosco - EnginSoft
d.magnabosco@enginsoft.com

About Argo Tractors

Argo Tractors designs, produces and distributes a wide range of tractors under three historical brand names: Landini, McCormick and Valpadana. It is a global company with around 1800 employees that makes continuous investments in R&D, of which this work is just one result.



A method to conduct dynamic analyses of floating solar structures using AQWA

Structural analysis method also considers response characteristics over time

By Jeongpil Hwang and Eunsil Han
Taesung S&E

This technical paper presents an analysis process to accurately examine the environmental loads and structural stability of a Floating photovoltaic (PV) power plant. The method includes a hydrodynamic analysis of the Floating PV in its water-based environment as well as a structural analysis of its structural stability based on the characteristics of motion it undergoes. The method proposed used ANSYS AQWA which allows environmental conditions to be included in the analysis such as the fender, the joint, the cable winch, irregular waves, birds, etc.

A new design solution for photovoltaic (PV) power plants is the use of Floating PV systems (FPVS), which are generally installed on bodies of water such as natural lakes, dams, reservoirs, or the ocean. This market is expected to expand because floating PV systems cause less environmental pollution problems than the traditional approach to the development of solar power. This paper introduces a method to conduct the hydrodynamic analysis of floating PV structures using ANSYS AQWA as well as a structural analysis of floating PV structures that considers the response characteristics over time.

The analysis process

The installed floating photovoltaic module structure is exposed to severe environmental loads such as winds and waves. Therefore, the FPVS engineers needed to establish an analytical process for conducting a structural evaluation of the robustness of the solar structure and the solar modules. To this end, our team proposes a four-step process of numerical analysis of the FPVS to understand the hydrodynamics and structural characteristics of floating solar structures, consisting of:

1. CAD modeling using ANSYS SpaceClaim
2. Computational fluid dynamics (CFD) analysis of wind load using ANSYS AIM or ANSYS Fluent
3. Hydrodynamic analysis of wave and wind speed using ANSYS AQWA
4. Time response analysis of the overall structures using ANSYS Mechanical

To summarize, firstly we used ANSYS CFD AIM for fluid dynamics analysis on the extraction of load data. Then, we used ANSYS AQWA for hydrodynamic analysis of the aquatic conditions and environmental loads. Lastly, we used ANSYS Mechanical for the structural analysis of the Floating PV structure, including the frames and solar panels.

Model description (TSNE's Arbitrary Model)

We used three types of models for the different analyses (Fig.1)

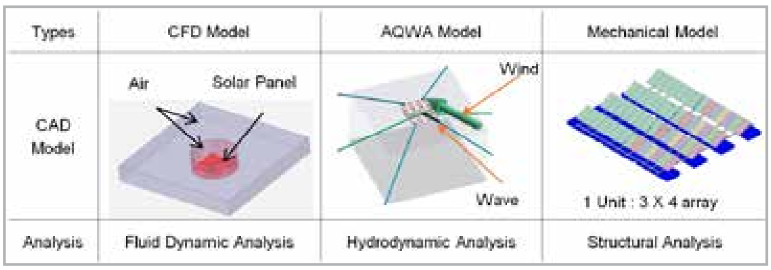


Fig. 1 - Three types of models

Hydrodynamic analysis using ANSYS AQWA

Since the use of AQWA in this analysis process is different from the general analysis method, we have explained it in more detail. The fluid dynamic analysis to derive the loads on the panels of the floating solar structure was conducted with ANSYS CFD AIM, and the hydrodynamic analysis of the water conditions and the environmental loads was conducted with ANSYS AQWA. It is important to emphasize

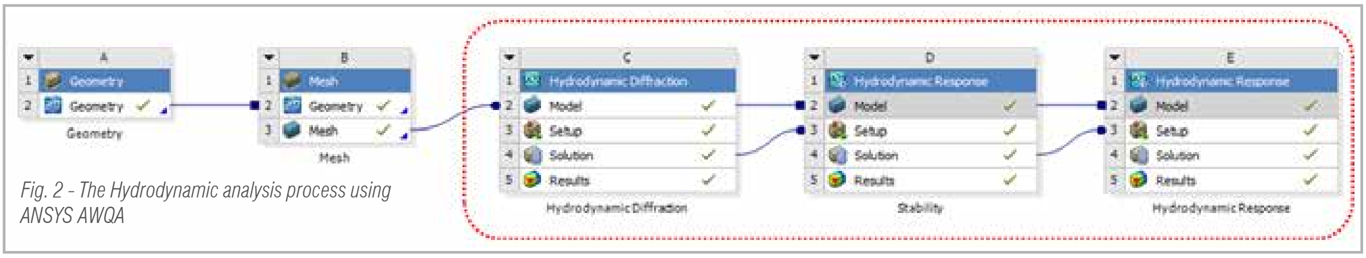


Fig. 2 - The Hydrodynamic analysis process using ANSYS AWQA

the use of ANSYS AQWA in our analysis process. Firstly, we ran a Hydrodynamic Diffraction analysis of the floating body to check its stability. Secondly, we obtained the specific behaviors of the floating body itself using hydrodynamic response analysis. Then, we conducted a simulation that considered the environmental loads created by the wind and waves to obtain the data of the structural position of the floating body. (Please see the red-dotted box in Fig. 2)

Results

Fig. 3 shows the results of the CFD analysis in terms of the total speed and pressure of wind from the direction of 90 degrees and from 180 degrees.

We extracted each X, Y, Z Force component according to the wind direction from -180 degrees to 180 degrees of total wind force

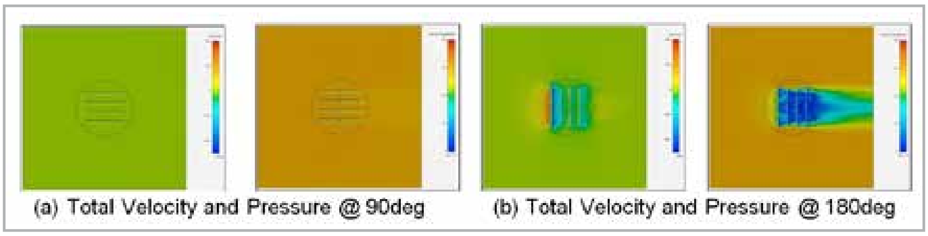


Fig. 3 - Total Velocity and Pressure with wind directions of 90 and 180 degrees

Angle	Force, X(N)	Force, Y(N)	Force, Z(N)
-180	10630.88886	5.783211957	17998.32397
-170	12418.10102	-2160.539692	21313.05518
-160	14790.7097	-5204.486723	26491.81062
-150	14974.63493	-8447.234269	29165.94937
-140	13457.93799	-15005.15894	29756.327

Total Wind Force

Angle	X(N/(m/s)^2)	Y(N/(m/s)^2)	Z(N/(m/s)^2)
-180	11.81209873	0.006425791	19.99813775
-170	13.79789009	-2.400599658	23.68117242
-160	16.43412188	-5.782763025	29.43534513
-150	16.63848326	-9.385815855	32.40661041
-140	14.95326443	-12.22795438	33.06280778

Wind Force Coefficient

Fig. 4 - Wind force and wind coefficient by wind direction

with CFD analysis using parametric variables, as shown in Fig. 4.
We transferred these wind force data to wind load coefficients by dividing by the square of the velocity. This wind load coefficient condition applies to the hydrodynamic analysis of floating structures.

Fig. 5 shows the wave surface elevation over time using the hydrodynamic diffraction with the above-mentioned wind coefficient results. This result shows the wave properties such as diffraction and radiation around the floating structure according to the wave

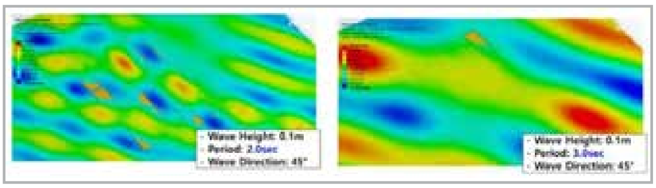


Fig.5 - The wave surface elevation

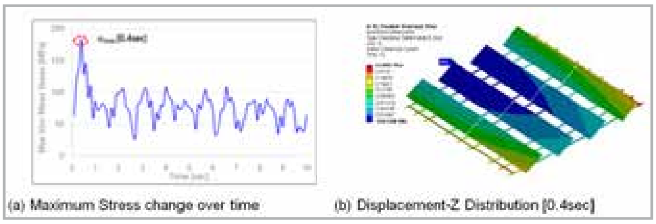


Fig.6 - Transient structural analysis

conditions such as wave direction and wave height. The highest wave height was found to be 0.26m at the 0.2sec period.

Finally, Fig. 6 shows the results of the transient structural analysis. These represent the maximum stress on the structure over time with the solar sample model installed in the water-based environment.

The analysis revealed that the sample model experienced its most unstable structural state at 0.4 sec and a maximum stress of 184 MPa.

Conclusions

Since no analysis process exists for the structural evaluation of a Floating PV installation, it was necessary to establish an analysis process to accurately examine the environmental loads and structural stability of a Floating PV. With this purpose in mind, this paper presents a method of conducting a hydrodynamic analysis of a Floating PV in its water-based environment and a structural analysis for examining its structural stability according to the characteristics of motion it undergoes. ANSYS AQWA enables various problems to be included in the analysis such as multiple environmental conditions including the fender, the joint, the cable winch, irregular waves, birds, etc. Furthermore, a fatigue analysis can be conducted to evaluate the fatigue life of the Floating PV.

References

[1] "ANSYS AQWA Reference Manual Release 2019 R2", 2019
[2] "ANSYS AIM Manual Documentation 2019 R2", 2019
[3] Yeom Duckjun, Dynamics of Marine Structure, UUP, 2010

For more information
jphwang@tsne.com or ehan@tsne.co.kr
or visit: www.tsne.com