

Ship-Shape Simulation

Designers use structural and hydrodynamic analysis to ensure working vessels meet challenging demands in harbor and at sea.

By Kuno van den Berg, Project Manager, Gijsbert Jacobse, Marine Engineer and Michiel Verdult, Marine Engineer, Vuyk Engineering Rotterdam b.v., The Netherlands

Designing ships is a demanding process, since vessels are among the most massive and mechanically complex moving structures in the world. Ships must operate reliably in harsh environments and meet stringent standards. Engineering demands are particularly challenging in designing work vessels for harbor and open-water applications, such as hoisting, dredging, construction, pipe-laying and other marine operations. The hull and internal structural members must be seaworthy and stable. In addition, topside mechanical assemblies, such as cranes, must provide sufficient strength and reliability to work efficiently even while waves excite the ship.

In meeting these demanding requirements, engineers at Vuyk Engineering Rotterdam (VER) in The Netherlands rely heavily on engineering simulation to develop designs and upgrades, ensure that government and industry standards are met, and resolve field problems that may arise. VER serves the maritime industry worldwide by providing consultancy

and engineering services for ship design, equipment design, marine operations and building supervision. The company has used ANSYS Mechanical software exclusively since 2002 for structural analysis in determining stress distributions, elastic deformation, reaction forces and component fatigue. Engineers utilize the software for other types of detailed analysis such as calculation of structural vibration and impact loads of one structure colliding with another. Vuyk chose ANSYS after an exhaustive evaluation of competitive packages based on the flexibility of the code for a range of applications, recognition of the software globally as best-in-class in numerous industries, and depth of the technology for a broad range of features.

In 2007, VER implemented ANSYS AQWA software for computing hydrodynamic motions and loads on vessel hulls for strength and fatigue analyses. This

software can be used to determine vessel response to wave environmental conditions. Such capabilities are required to study critical operational details such as cargo swaying as it is lifted, relative movement of a moored vessel, and interaction of adjacent ships as well as the ability of the ship to hold a given position in heavy seas. The capability to smoothly exchange data between ANSYS AQWA and ANSYS Mechanical products is critical in performing analysis in applications in which

Matador 3 in action on open-water construction site



structural behavior is closely related to hydrodynamic effects.

Prior to licensing ANSYS AQWA, VER employed rudimentary 2-D strip theory software. This calculation method had limitations, as it was only applicable for a narrow range of traditional ship hulls and was not suitable for pure wave-load calculations or multibody motion analysis. In these cases, VER outsourced work to an outside research center or university.

Bringing the work in-house with ANSYS AQWA technology gives engineers more control over the hydrodynamic analysis and greater insight into vessel behavior. The design team also can iterate much faster by varying parameters to compare alternatives and optimize designs. Performing time-domain dynamic motion analysis in-house positions VER among the leaders in the marine industry and strengthens the company's competitive value as an engineering services provider.

Upgrading Lifting Capacity

In one recent project, VER engineers used the ANSYS tools to upgrade the lifting capacity of the Matador 3 sheerleg — a self-propelled floating crane used for lifting heavy loads at the Rotterdam seaport docks as well as for offshore construction projects, open-water wreck removal operations, and bridge and lock construction along inland rivers and canals.

The Matador unit consists of two hinged, adjustable A-frame structures with a hoisting jib held in place by a network of cables looped through deck sheaves and controlled by main power winches on the base of a pontoon platform. One of the larger floating sheerlegs in the world, the Matador 3, which is owned and operated by Bonn & Mees, has a maximum height and reach of 70 meters. Two lifting blocks at the top-most point of each structure are raised and lowered by cables and winches to lift



Finite element analysis calculated stress distribution in the overall Matador 3 lifting structure (top) as well as individual critical components such as the sheave support plates (bottom)

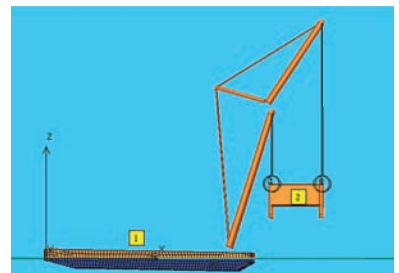
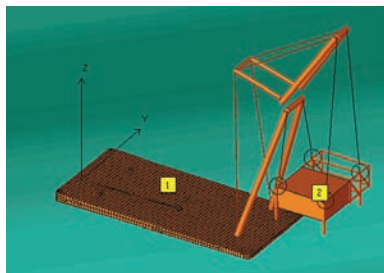
cargo. Alternatively, the A-frame can be used with four blocks for lifting.

The aim of the redesign project was to increase the jib lifting capacity from 600 to 900 tonnes. Engineers accomplished this by increasing the number of sheaves (pulleys) used on the ship for the cables to the two jib hoists as well as sheaves for holding the A-frames in place. Engineers used ANSYS Mechanical software to optimize the jib load capacity by modeling the lifting frame with beam elements and calculating reaction forces at the hinge points and stresses across all

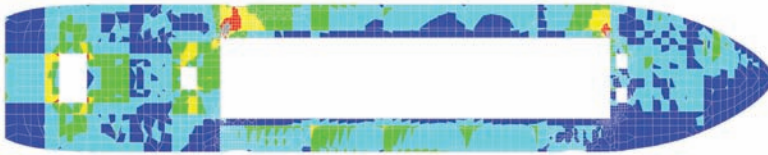
structural members. By parameterizing the model, the team was able to quickly enter different angular variables to generate lift, reach and lifting capacity curves for the Matador.

In separate analyses, ANSYS AQWA was used to study the motions and loads of the lifting structure as waves of various heights and frequencies impact the vessel at different angles. In these studies, the vessel and crane structure were modeled separately from the lifted structure and then combined with representations of the connecting cables into a multibody hydrodynamic model. The engineering team determined the working range of the structure with respect to the sheerleg capacity and positioning accuracy.

VER also performed studies for individual projects in open waters, including a motion analysis of the Matador during lifting, transporting and installing a wind-turbine high-voltage station in the North Sea for various wave scenarios. ANSYS AQWA software was used to determine the maximum wave height allowable for various wave periods (time between wave crests) and headings. Using the simulation from the study, operators were able to match the ship's work schedule to weather forecasts for the 12-hour trip from harbor to open-water location to safely transport the sections of the wind turbine, position the vessel and lift the foundation into place with the platform on top. To describe the rigorous method of the study to



Diffraction analysis calculates the sway and stability of hoisted loads.



Joint structural and hydrodynamic analysis determined maximum combined torsion and bending aft of the hopper.

the client, VER cited ANSYS AQWA capabilities and included the graphical and tabular output from the software in the client report.

Upfront Simulation

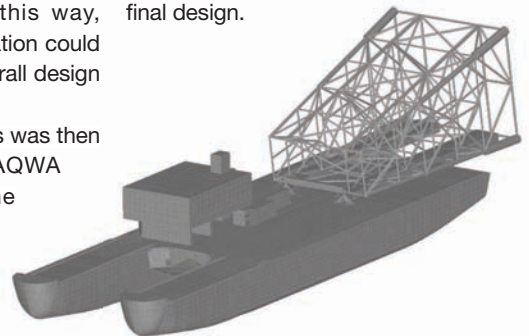
In another study, engineers used ANSYS Mechanical and ANSYS AQWA software in a one-way coupled simulation in which hydrodynamic pressure loads against the outside of the vessel hull calculated by ANSYS AQWA software were transferred directly into ANSYS Mechanical to determine the structural behavior of a trailing suction hopper dredging vessel. In particular, the study was intended to check longitudinal bending of the critical midship region of the hull, calculate overall hull girder effects at the aft and fore hopper ends, and provide a detailed stress analysis for evaluating girder fatigue.

Initially, engineers created a finite element model for ANSYS Mechanical using shell elements. The model included all the major parts of the ship, such as the outer hull, girders and basic topside structure. This represented only the basic geometry and mass distribution so analysis could be done in the early stages of basic vessel design, which takes about five months to complete. In this way, results of this upfront simulation could be used as input for the overall design of the ship.

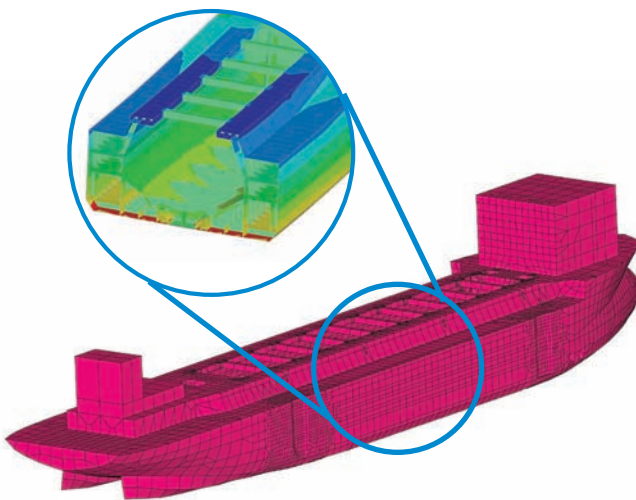
A 3-D diffraction analysis was then performed with ANSYS AQWA software to determine the pressure distribution around the complete perimeter of the hull from loads generated by waves on the side of the vessel and associated vessel motions. The finite element hull

geometry served as the basis for the diffraction model, ensuring compatibility between the finite element and diffraction analyses.

VER engineers combined still-water and wave load sets from ANSYS AQWA into a load set representing total water pressure against the hull. This data was then used in ANSYS Mechanical to determine stress and buckling of the structural girders. In this simulation, engineers found a stress concentration in the main deck aft of the hopper. Structural strengthening in this region was achieved by adding thicker deck plates and additional girders. A subsequent fatigue analysis on the modified structure ensured the validity of the final design.



Girder analysis of the twin-hull Pieter Schelte pipelaying vessel, whose unique design created a number of engineering challenges



ANSYS Mechanical model of the trailing suction hopper dredger and longitudinal stresses in the dredger's midship section

In a separate project, coupled simulation was used in the analysis of the hull girder design of a unique and very large twin-bow vessel named Pieter Schelte, currently in final design and planned to be the world's largest pipelaying vessel. The ship will be about 1,250 feet long and 380 feet wide. Due to the significant height-to-length ratio of each bow, traditional rules for ship design are not applicable for this twin-bow vessel. Thus, the use of ANSYS AQWA technology was beneficial because the flexibility of the software allowed it to be customized. ■