range of well conditions can be covered by the existing program. From the analysis of the mechanical requirements of the wire it is, however, evident that there is a need for a high-strength alloy.

Material from a standard Sanicro 28 was remelted in the lab furnace with varying N-content to improve the strength. Based on positive results from the investigations, a heat was ordered with 0.125%N. The material was hot rolled into wire rod and then wirelines were drawn.

The wire was investigated both with regard to surfaces and metallographic structure. It was then tested in a rig simulating conditions during a jarring operation. As referenced, the standard version of Sanicro 28 was tested, together with the newly developed Sanicro 26 Mo. A load of 193 kg was applied, and the wirelines were pulled over a wheel diameter of 17.8 cm. From the results, it is clear that the modern alloys have excellent properties from both mechanical and corrosion points of view. It is worthwhile to note the drastic reduction on the number of cycles to failure when increasing wire dimension to 0.125 in. from 0.092 in.

In conclusion, Sandvik has made thorough material investigations to simulate its products intended for oil / gas service. It has been shown that the materials available are expected to perform reliably in field service. WO

Literature Cited

1. Sato et al., *Proceedings of the 2nd Asian-Pacific Corrosion Control Conference,* 1981.
2. Oredsson, J. et al., *Materials Performance,* Vol. 22, No. 1, 1983.
3. Dundas, H. I., *Climax Report RP-33-80-13,* Nov. 29, 1982.
4. Vaughn et al., NACE Corrosion Conference, Paper 182, 1982.

Back to Top

Computer simulations analyze wave damage to offloading vessels

Charles Ortloff, Manager, Numerical Simulation Department, CTC/United Defense, Santa Clara, California

Computer simulation is beginning to play a major role in analyzing floating production / storage / offloading (FPSO) vessel damage in typhoons and other storms. Prediction of transient loads resulting from green-water impact on FPSO hull and deck structures has traditionally been determined solely by scale-model tests. However, physical tests are expensive, and methods currently used to scale up test results leave a lot to be desired, particularly for deepwater applications. That is the main reason why computational fluid dynamics (CFD), a numerical method that solves fluid-flow equations, is increasingly being used to simulate green-water impact.

CFD has already been demonstrated to be capable of accurately predicting transient loads on FPSO hull and deck structures. Advantages include the fact that it is less expensive, and it eliminates scaling difficulties associated with physical tests. Additionally, wave speed / length / period relationships for deep- and shallow-water applications, as well as wave shapes, can be prescribed as input.

FPSOs hold their position through use of computer-controlled thrusters or anchoring, and can operate in various water depths. Currently, there are about 85 FPSOs in the worldwide fleet, either operating or under construction - up from about one-third of that only five years ago.

Green-water impact. One of the most critical issues in FPSO design is their ability to withstand green-water impact without damage. Green water, defined as water coming onto the deck of a ship in large waves, is the most common cause of damage to vessels in the U.S. Navy. On an FPSO, green water can damage sensitive on-deck equipment such as piping, the turret structure and hatch structures structure and hatch structures.

While FPSOs are mobile enough to move out of the path of a storm, if given enough warning, typhoons can rise up suddenly. For example, in 1996, Typhoon Sally passed within 10 nautical miles of the *Liuhua* FPSO, generating waves 88-ft high and winds of 112 mph - conditions that are associated with a 200-yr event. *Liuhua* successfully weathered this super typhoon - with only minimal damage to pipe insulation, antennas, windows, etc. - due to its conservative design.

Physical testing has long been the only method available to determine loadings resulting from green-water impacts. Test tanks are constructed, and artificial waves are crashed against a small model structure representing the hull. A typical test tank might be one-quarter acre in size, resulting in a substantial construction expense. Testing a different hull design in waters of different depths would require expensive modifications due to depth-related wave-speed and wave-length relationships.

But the biggest problem with physical testing is the issue of scaling. It is generally accepted that scale effects in the absence of waves follow Froude scaling. However, on-deck flows following the moment of high-velocity impact and post-impact flows - which contain sprays and thin sheets of runoff - are likely to be affected by different scale factors, or governed by empirical correlations to relate models to full-scale transient forces. Concern exists that scaling effects may render many existing physical-model test results inaccurate. To compensate for this uncertainty, vessels are often constructed with larger-than-necessary margins of safety, which increases their cost.

Usefulness of CFD. In an effort to overcome these problems, researchers at CTC / United Defense have, over the last several years, worked to develop a capability to simulate green-water impact on an FPSO with CFD analysis methods. This involves solution of the governing equations for fluid flow at millions of discrete points within a computational grid in the 3-D flow domain. For this work, it was central to have a method that could model the free-surface development of a green-water wave in its fullest complexity.

FLOW-3D ® from Flow Science, Inc., Los Alamos, New Mexico - which uses the volume-of-fluid (VOF) method to predict free-surface fluid motions - incorporates surface tension, turbulence models and other flow complexities necessary to describe the free-surface-flow pattern development. In particular, this code provides algorithms that track sharp liquid interfaces through large deformations and applies the correct normal and tangential-stress boundary conditions - a feature that distinguishes it from other CFD programs. This makes it possible to track separated "blobs" of fluid, as well as air bubble entrainment effects - generated after a wave impact - that have an important effect on total pressure / viscous forces exerted on the vessel.

When properly validated, a CFD analysis allows engineers to obtain transient flow velocities and pressures at any location in the problem domain. This is far more information than can be obtained from test tanks, and it brings far more useful information to bear on the design process. The geometry of the vessel, depth of water, size of wave, wave speed and period, or any other condition can be changed quickly on the computer and re-analyzed to determine effect of the change. Most important, scaling is not an issue with CFD, because the simulation can easily be performed at actual size. Of course, simulation is not a substitute for testing, but rather a useful tool that, once validated, can reproduce conditions that would be impossible or impractical to duplicate in the test tank.

Simulation example. In one recent example, CTC researchers constructed a solid model of a 160,000-dwt FPSO hull and transferred it to FLOW-3D as a fluid-flow obstacle. To ensure a hydrodynamically smooth outer surface, the hull was created using ANSYS methods with 60,000 tetrahedral elements. The entire fluid-plus-solid domain consisted of a 600,000-active-cell, 3-D mesh. A scaled-geometry rectangular structure was created on top of the hull with height, width and location similar to an instrumented test wall used in a corresponding physical-model test.

Code was developed to simulate hull pitching frequency / amplitude that

Code was developed to simulate hull pitching frequency / amplitude that match the motion derived from test instruments. Heave and surge effects were incorporated into the code but not applied to the current calculations. The phase angle between wave and hull was set so that maximum up-pitch corresponded to the crest impacting the bow. Near the tanker bow, the computational grid size was refined to more accurately capture fluid interactions with the hull.

CFD simulations were performed to study green-water behavior in three basic stages: 1) a large increase in relative wave elevation at the bow; 2) a large volume of water separating from the wave and flowing aftward on the foredeck with increasing speed due to bow-up pitch accelerations; and 3) high-velocity flow impacting the deck bridge structure. There were striking similarities between flow behavior seen in the simulation and that shown in a photo sequence from the physical-model test.

Example simulation. Incipient deck-overflow, free surface wave pattern for a 10-sec. wave period and wave amplitude of 15 m for the hull starting its pitch-up mode. Note presence of Vedernikov waves in the overfall flows onto the foredeck. Deepwater conditions apply.

Free surface shape for wave impact with the vertical bridge structure on the FPSO hull. Transient forces on the vertical bridge are a combination of impact, hydrostatic and viscous loads.

Free surface shape of water as it streams off the foredeck, for the hull at maximum pitch-up angle about its center of gravity. Loading on the foredeck is primarily from hydrostatic loads resulting from accumulated water.

At the start of the simulation, some water remains on the deck from the previous wave cycle, which also transmitted minor disturbances to the next incoming wave. At one-sec. time, the wave crest approaches above the deck; between the 3- and 4-sec. mark, pitch motion is near a maximum and the wall of water begins to overflow the deck. At 4 sec. into the simulation, a large volume of water is shooting down the foredeck. Both simulation and test photos show a distinct, three-tiered surface overflow side pattern just ahead of the bridge between 3.5 and 4.5 sec.

Near the 4.5-sec. point, water has contacted the deck structure while the reflected wave pattern propagates radially outward from the bow. By 6 sec. into the simulation, the deck is completely enveloped in water. In general, there is a somewhat larger quantity of on-deck water in the simulation than in the tests. The exclusion of heave in the simulation, as well as size differences between model and full-size hull, may explain this difference.

Between 7 and 8.5 sec., deflected water is flowing off the sides of the up-pitched deck, which is consistent with test photos. After maximum water height on the deck structure is reached, runoff of the accumulated deck water begins. From 13 to 17 sec., the disturbances created by deck runoff can be seen, as well as buildup of the next crest wave.

CFD was also used to predict lateral-force coefficients for a 1:82.5-scale model of a 200,000-dwt tanker. Computations were performed for a fully loaded vessel, with a conventional bow shape, at various angles of attack and water depths. In general, there was a good agreement between computations and test data.

The technique shows great promise for improving detailed knowledge of design loads for FPSOs and other structures subject to severe wave and current loading, and it will prove valuable for fatigue / stress-analysis safety qualification of FPSOs in the future. Results of sample analyses indicate CFD methods can provide information that cannot be obtained from scale-model testing. In some cases, the computations provide insight into details of the flow field not available from tests due to technical limitations of test probes and the limited number of probes that can be placed in a flow field.

CFD also has the potential to identify scale parameters for conversion of test data to prototype results by providing solutions for both model and full-scale structures. In the future, while testing will continue to play an important role, there will be an increasing shift to the use of test-verified CFD runs as a design tool (particularly as computer speed and capacity increase) resulting in more-accurate simulations. Flow Science, Inc.'s website is: www.flow3d.com. WO

---

* Trademark of Flow Science.

Marine News, Apr 30, 2001/p. 22-23

Back to Top

Stripper well consortium established

E. Lance Cole, Petroleum Technology Transfer Council; and **Gary Covatch,** DOE National Energy Technology Lab

DOE's Strategic Center for Natural Gas and the National Petroleum Technology Office at the National Energy Technology Laboratory have partnered with The Pennsylvania State University and the University of Tulsa to establish a national, industry-driven consortium that is focused on improving the production performance of domestic petroleum and gas stripper wells.

The aim is to achieve improved results through collaborative efforts in identifying and funding small R&D projects. DOE will provide up to $3.0 million over three years - to be matched with a 30% cost share, or $1.3 million from industry - to fund projects selected by the consortium. One