Drop Test into Water and Wave Impact Simulations of a Novel 7-Meter Plastic Boat with LS-DYNA®

Martin Vézina
Arash Firoozrai
SimuTech Group Inc.
550 Chemin du Golf, suite 100
Verdun (Québec), Canada H3E 1A8
www.SimuTechGroup.com
mvezina@SimuTechGroup.ca

Abstract

The US Congress, in its desire for a safer boat for the US Navy, contracted Stanley Widmer Associates Inc. to design and build a novel rotationally molded 7-meter boat using a patented "kiss-off" design. A first ever made prototype will be ready for trial in 2010. The material used for most of the boat, high density cross linked polyethylene, and the double hull with "kiss-off" design should, among other advantages, increase the boat's impact resistance compared to fiberglass and aluminum boats.

This paper presents numerical fluid structure interaction simulations done on the 7-meter boat, utilizing LS-DYNA, in order to investigate its structural integrity under water impact. Drop test into water simulations were done in the first phase of this numerical investigation. The boat was dropped at different heights and angles. A modal analysis was also done in Phase 1. A procedure to generate realistic waves was developed in the second phase. Explicit simulations with the boat going through waves at different velocities were done. A similar fiberglass boat was finally compared to the plastic design.

Results from the drop test simulations and the modal analysis in Phase 1 clearly showed a critical flexural area. A modified design was run in Phase 2. LS-DYNA analysis results from Phase 2 predict that the plastic boat can go through 3-foot waves at 40 knots without damage while the fiberglass one has predicted damage at 30 knots. Overall, the numerical simulations show that high density cross linked polyethylene could be the new age of boating material.

Introduction

The US Navy has different types of vessels from big ships to small boats. The US Navy uses boats for patrolling rivers, countering pirates on the high seas, clear mines, etc. The boats need to withstand blast, impact and\or structural loads when they are used to blow up mines, when moving through rough seas at high speed and\or when they are hoisted aboard. Standard hull designs currently used by the US Navy are made of aluminum or fiberglass. Aluminum hulls do not withstand successive blasts well and fiberglass hulls break when moving through rough seas at high speed.

The goal of this work was to build a virtual prototype of a novel rotationally molded 7-meter boat being designed by Stanley Widmer Associates Inc. (SWA). A first ever made prototype will be ready for trial in 2010. The characteristics of SWA's design are a double hull with patented "kiss-off" connection between the two hulls and the material used for most of the boat i.e. a High Density Cross Linked PolyEthylene (HDXLPE). Those characteristics should make SWA's boat more flexible (less breakable) and thus more blast/wave impact resistant, unsinkable, resilience to small arms fire, cost effective, etc. Figures 1-3 show the design, molding tools and the first molded hull respectively.

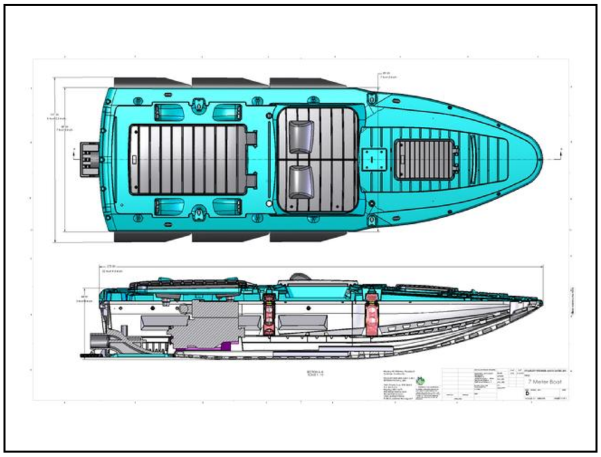


Figure 1: 7-meter boat design

Figure 2: 7-meter boat molding tools

Figure 3: First molded hull and cut-away showing the patented "kiss-off" connection

Several numerical simulation types were done on the 7-meter boat virtual prototype i.e. static, modal and dynamic. The numerical simulations were done with either the implicit or the explicit finite element LS-DYNA solver.

The numerical investigation started by simulating drop tests into water in order to assess the structural integrity of the boat. The intention was to mimic a wave impact. The decision to do drop tests was made because (1) we were not able to generate realistic waves in LS-DYNA at that time and (2) we knew that drop tests into a water tank is currently used in the industry for testing new designs. A modal analysis was also done in parallel. This work will be referred as Phase 1.

Phase 2 was started several months later when a comparison between the plastic boat and a similar fiberglass boat was required by the US Navy. A procedure to generate realistic waves and a fiberglass boat virtual prototype were then developed.

Numerical model development in Phase 1

The set up of the plastic boat was mainly carried out in Phase 1. The original solid CAD model was read in, simplified, repaired and converted to a surface model with ANSYS® DesignModeler™. ANSYS® ICEM CFD® was used to generate a shell/solid mesh. LS-PrePost® was then used to finalize the numerical model.

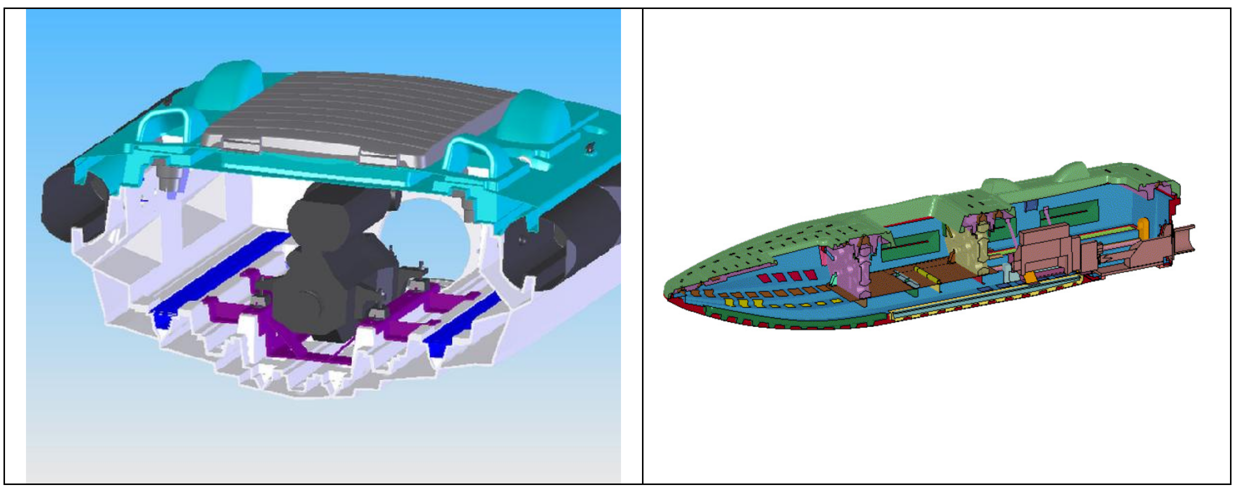


Figure 4: CAD model, Simplified model

Figure 4 shows a section of the solid CAD model and the FEA model. The boat is made in two parts i.e. the hull and the deck. Constant thickness was assumed for all parts. The propulsion system was meshed with solid elements and considered rigid. Airbags on the side of the boat and the deck's hatches were not included in the model. Other details were also neglected e.g. seats, etc. Figure 5 shows the mesh of the hull with the rear tank, supports, propulsion system and struts running the length of the hull up to the front tank.

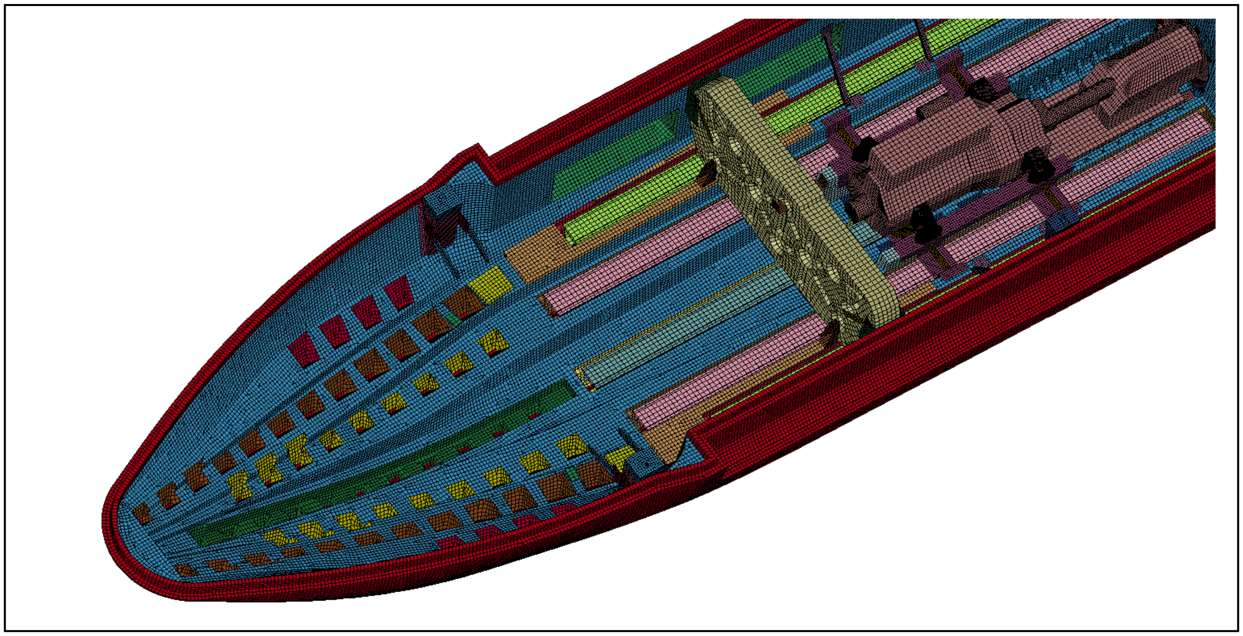


Figure 5: Boat mesh

Most of the connections in the assembly were simplified by rigidly grouping and sharing a set of nodes. Some connections in the assembly were simplified by sharing a set of nodes to the rigid propulsion system.

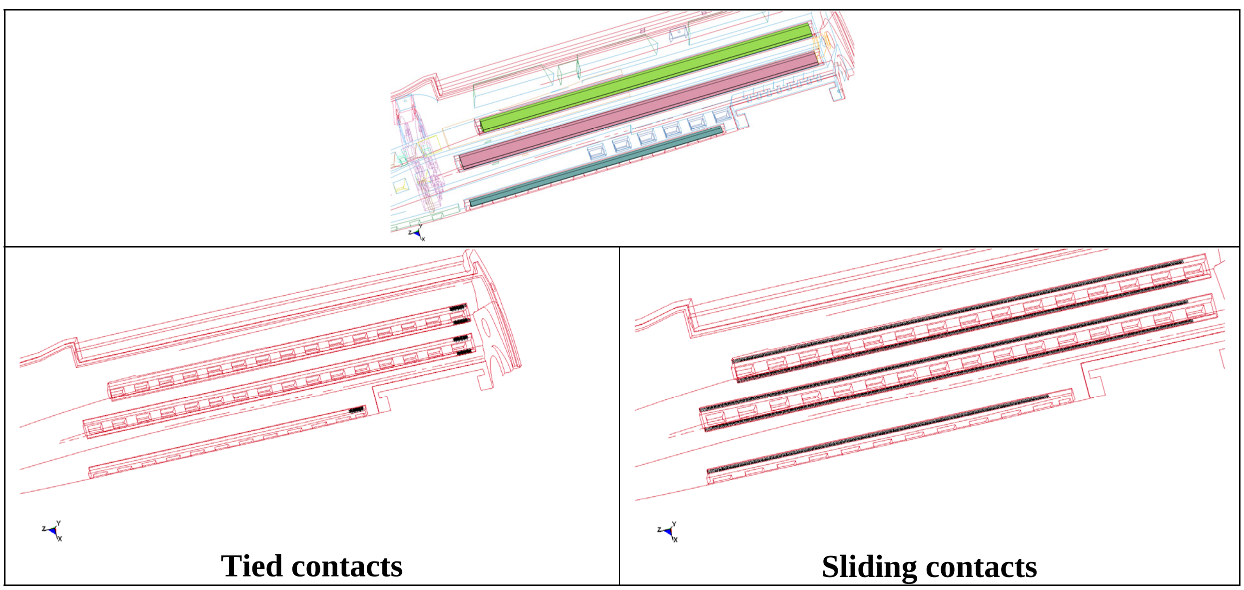


Figure 6: Connections between the aluminum struts and the hull

The connections between the boat and the struts were modeled with contacts as showed in Figure 6. The struts are fixed at one end and are free to slide along their length elsewhere. Fully tied and sliding connections were thus used. The connections between the hull and the deck were modeled with tied contacts. Finally, frictionless contacts between all boat's parts and themselves were taken into account.

Most of the boat is made of plastic i.e. HDXLPE. Supports are in stainless steel; struts are in aluminum. A simple isotropic elastic material model was used for the supports and the struts (*MAT_001). An elasto-plastic material model was used for the plastic (*MAT_024).

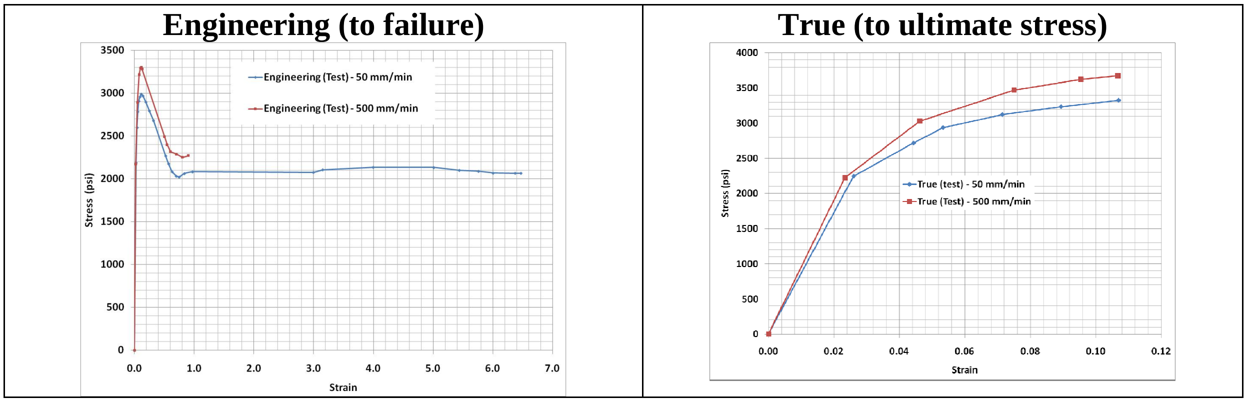


Figure 7: HDXLPE quasi-static stress strain curves

Only limited material data was available for the plastic. Assumptions and simplifications were thus inevitable. Figure 7 shows the quasi-static stress strain curves obtained from the supplier. The nominal strain rates in the tests were low (max of 0.333/s). Nevertheless, the tests show significant effects on the behavior especially on the failure strain. It was thus decided to use the true stress strain curve up to the ultimate stress and a failure strain equal to the ultimate strain since the goal of this work is to make sure the boat won't fail in service.

Drop test simulations and modal analysis in Phase 1

Boat makers are usually doing field tests on their prototypes. But they can also do controlled drop tests into a water tank. The prototype is then equipped with accelerometers, pressure and strain gauges. A high speed camera can also be used to record the event. The numerical investigation started by simulating drop tests into water. The boat was dropped at different heights and angles.

Water impacts were simulated using the Arbitrary Lagrangian Eulerian (ALE) approach in LS-DYNA. The ALE approach allows us to do Fluid Structure Interaction (FSI) simulations where the structures are modeled with a purely Lagrangian mesh and the fluids are modeled with an Eulerian mesh in which the fluids flow. A motion and/or a deformation can be prescribed to the Eulerian mesh.

Figure 8 shows the drop test simulations set up. The ALE domain was filled with water (70%) and air. Air must be included in the domain in order to simulate water motion and splash. The ALE domain was neither allowed to move nor deformed. Linear Equations of State (EOS) were used for water and air. An initial atmospheric pressure was set in water and air through the EOS initial internal energy or initial volume variable. Normal velocity of the fluids was constrained at all free surfaces of the ALE domain except on the top surface (a water tank). A reference atmospheric pressure was applied to all free surfaces of the ALE domain. The coupling mechanism for modeling FSI was set using *CONSTRAINED_LAGRANGE_IN_SOLID.

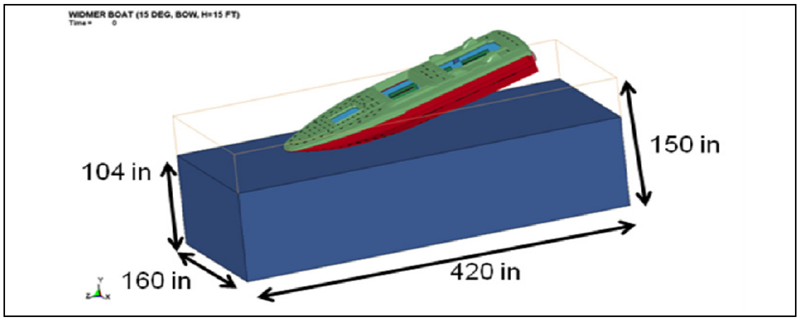


Figure 8: Drop test simulations set up

Results from the drop test simulations revealed a critical weaker section on the boat. A free-free modal analysis done on the boat with the implicit solver of LS-DYNA showed that the first flexural and torsional modes are around this critical section.

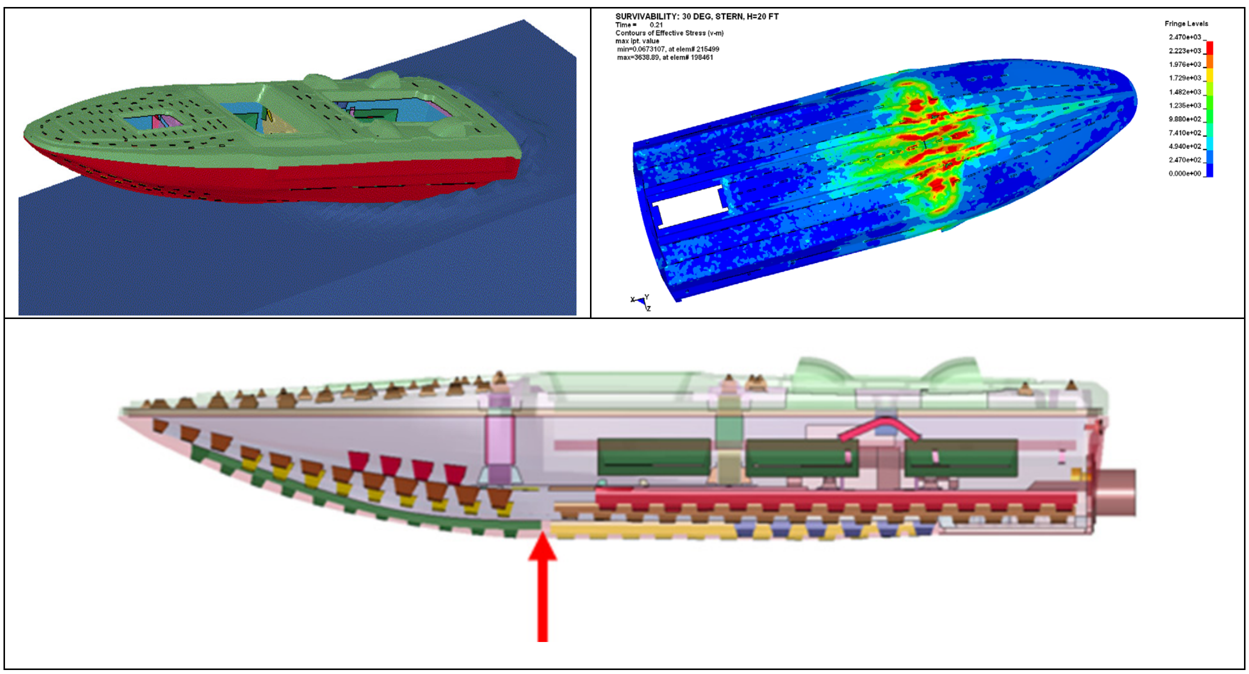


Figure 9: Results from drop test simulations in Phase 1

Figure 9 shows the interaction between the boat and the water and the equivalent stress in the external hull. The critical section is pointed out by the red arrow in the bottom image. The critical section is located at a cross section change, where the hull changes from a nearly flat to a curved shape, the struts end and where there was a discontinuity between the "kiss-off" structures. The length of the discontinuity was only about 65 mm long over 7 meters. The results also indicated that the flexibility of the deck was too high.

Numerical model development in Phase 2

Modifications to the design of the boat were made based on results in Phase 1. The critical section is stiffened by an extra aluminum strut supported by a plastic mount and attached to the existing strut. The plastic mount adds extra “kiss-off” to the structure. Two “kiss-off” were also added on each side. Figure 10 shows the design modifications.

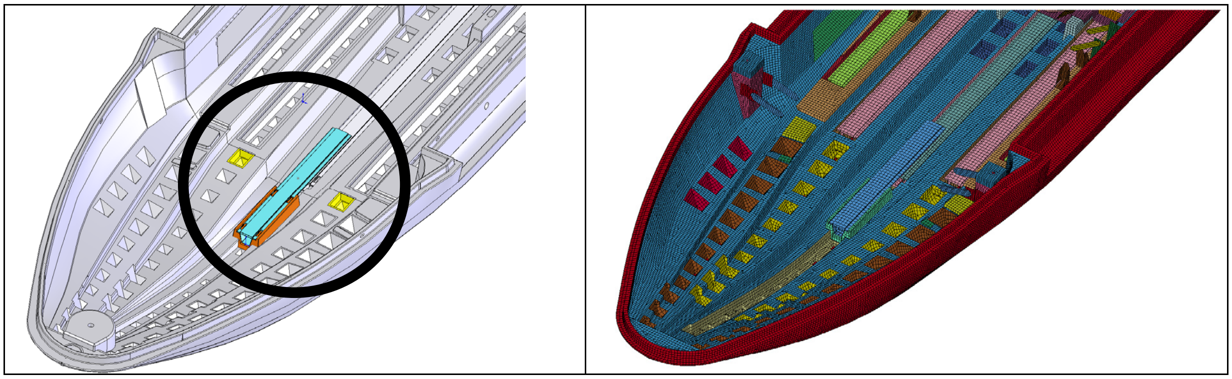
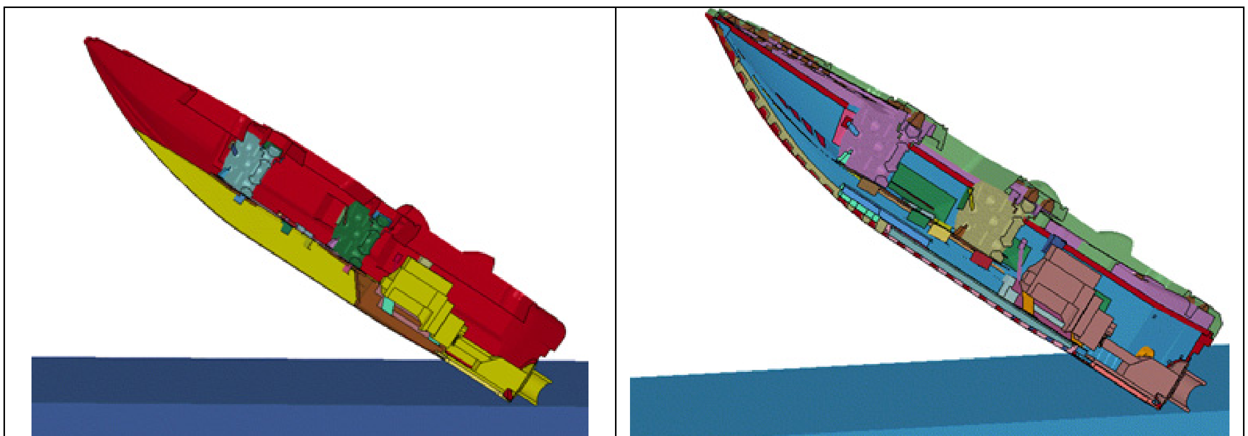


Figure 10: Design modifications

A similar fiberglass model was created in Phase 2 in order to compare its performance to the plastic boat. Figure 11 shows the fiberglass and the plastic boat models. The fiberglass boat was made of Chopped Strand Mat (CSM) and Woven Roving (WR) fiberglass. An orthotropic material model for composites with brittle failure was used (*MAT_022).



Fiberglass **Plastic**

Figure 11: Fiberglass and plastic boat

Drop test simulations in Phase 2

Figure 12 shows the stress distribution before and after design modifications for the plastic boat. It indicates that the stress after the design modifications is more evenly distributed along the “kiss-off”. It also shows that hot spots on each side are removed by the addition of an extra “kiss-off”.

The design modifications for the plastic boat do not change radically the stress path i.e. there is still a cross section change at this location. Nevertheless, the modifications are significant enough to reduce damageable localized effects without reducing too much the flexibility of the boat. The forces are in fact redistributed in a larger area but also in the aluminum struts.

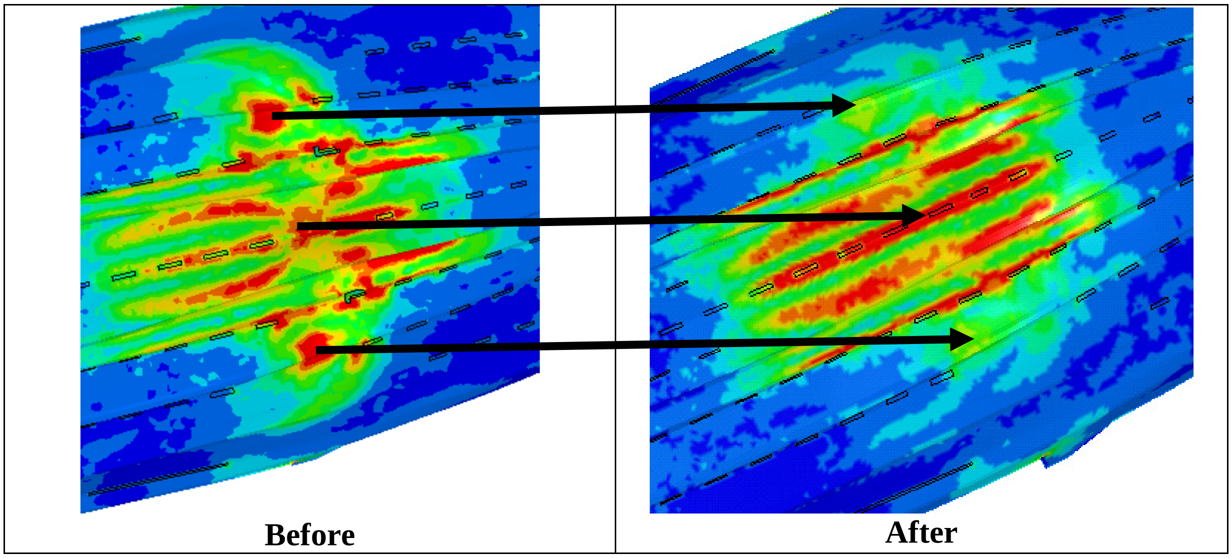


Figure 12: Stress distribution before and after design modifications for the plastic boat

The drop test results for the fiberglass boat did not show catastrophic failure even if some of the WR layers failed close to the critical region. The plastic boat is 26% heavier than the fiberglass boat but still the fiberglass boat is much less flexible than the plastic boat. Figure 13 gives the filtered acceleration at the Center of Mass (COM) of the rigid propulsion system. The fiberglass boat peak (20 g) is 54% higher than the plastic boat peak (13 g). This is a critical factor considering the level of g a Marine can sustain. Figure 13 also gives the internal energy. The plastic boat absorbs more impact energy due to its flexibility.

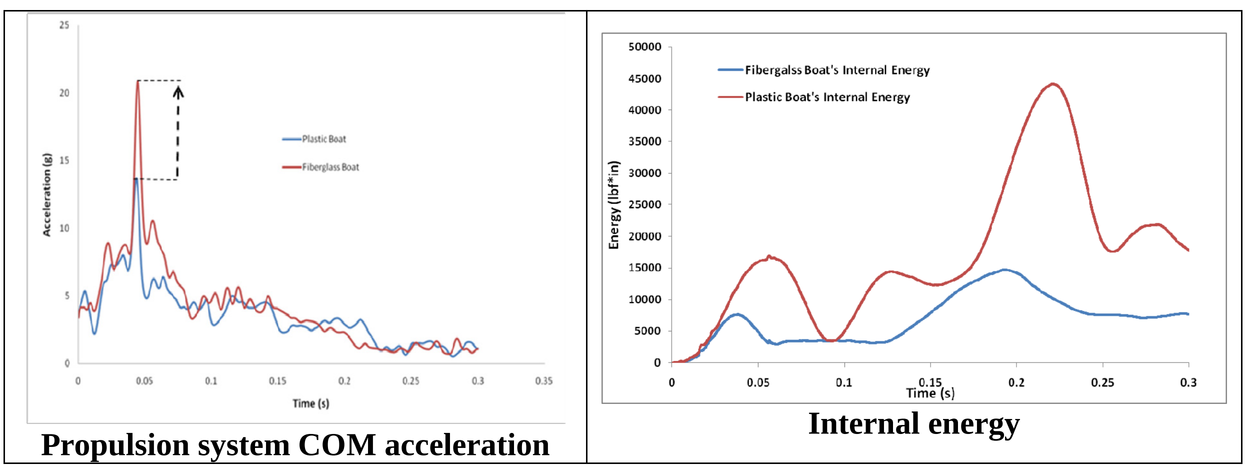


Figure 13: Acceleration and energy comparison between plastic and fiberglass boats

Wave generation

A procedure to generate realistic waves was developed in Phase 2. Several iterations were done before getting them in LS-DYNA. Modifications to the ALE domain needed to be done. New boundary conditions for the ALE domain were also required. Figure 14 shows the plastic boat through 3-foot waves.

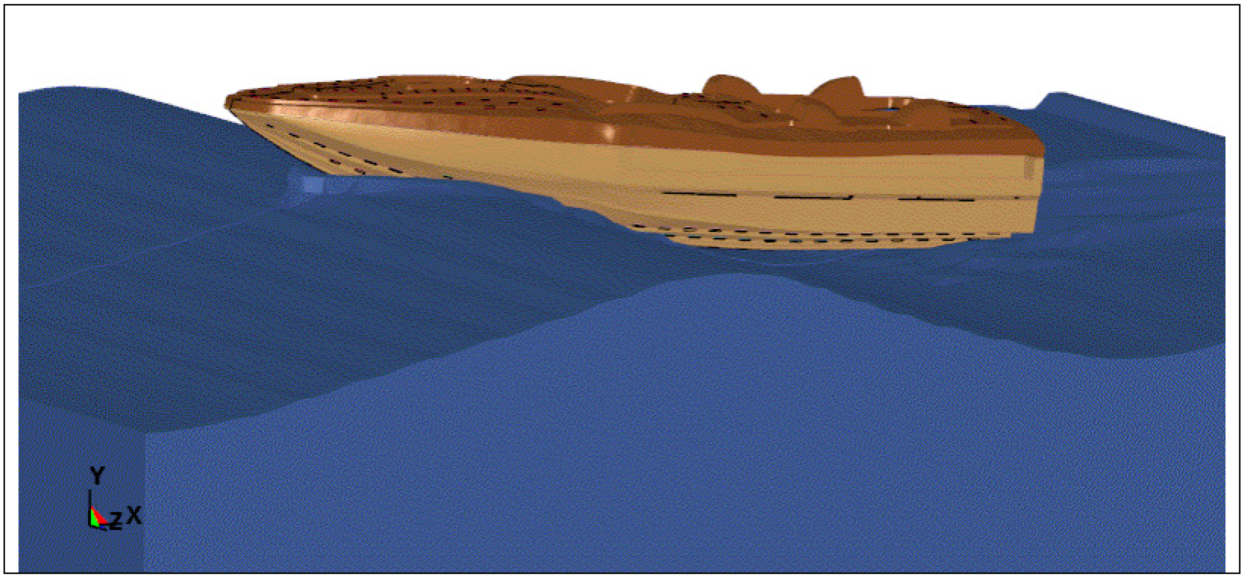


Figure 14: Plastic boat through 3-foot waves

A layer of ambient elements were added to the initial ALE domain at the inlet, outlet and at the bed (bottom). Hydrostatic pressure was applied by using *ALE_AMBIENT_HYDROSTATIC commands and by applying a constant base acceleration (gravity) to the ALE domain. Constant velocities were set at the outlet and bed ambient elements. A constant atmospheric pressure was applied on the top surface.

The velocities at the inlet were generated by using the Open Channel Wave Boundary Conditions equations found in ANSYS® FLUENT®. According to ANSYS FLUENT help, shallow waves are defined as:

$$\begin{pmatrix} u \\ v \end{pmatrix} = \frac{gkA}{\omega}\frac{\cosh[k(z+h)]}{\cosh(kh)}\begin{pmatrix} \cos\theta \\ \sin\theta \end{pmatrix}\cos(k_x x + k_y y - \omega_e t + \epsilon)$$

$$w = \frac{gkA}{\omega}\frac{\sinh[k(z+h)]}{\cosh(kh)}\sin(k_x x + k_y y - \omega_e t + \epsilon)$$

Figure 15: Equations used to generate shallow waves in ANSYS FLUENT [2]

where u is the component in the flow direction, w is the component in the gravity direction and v is the component in the cross direction of u and w.

The nominal wave speed was set to the velocity of the boat. The amplitude, wavelength and the shallowness variables were adjusted in order to get the appropriate wave configurations. There are of course limitations i.e. it is not possible to generate any types of waves but the current implementation was within the limits of this project.

Wave impact simulations

Both plastic and fiberglass boats were subjected to a wave impact. The boat did not have a forward motion i.e. it was rather the water that was flowing at the boat velocity. The boat was constrained at its COM. The boat was only free to translate along the vertical y axis (heave) and to rotate about the transverse z axis (pitch). All other degrees of freedom (DOF) at its COM were fixed. The water flow was along the x axis as showed on Figure 16.

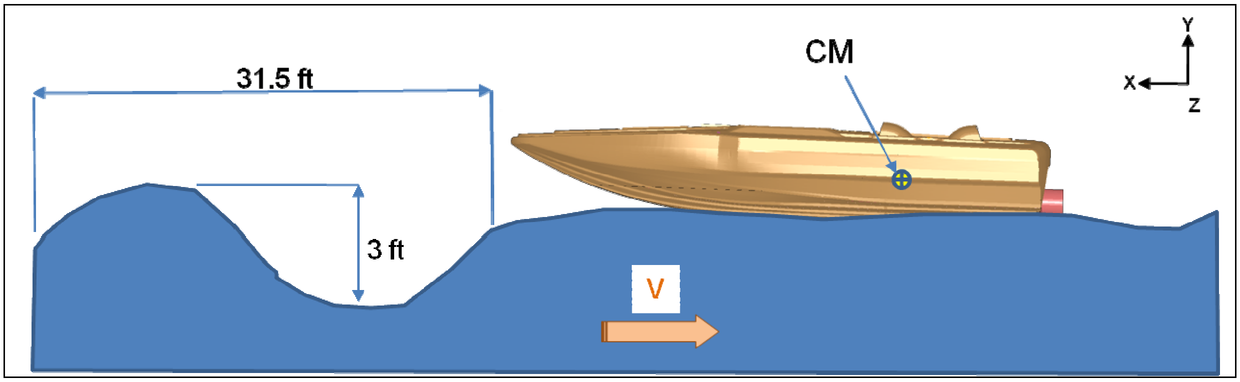


Figure 16: Wave impact set up

The wave impact simulation was divided in several phases. A special procedure was developed in order to switch all boat's deformable bodies to rigid bodies at the first iteration. The newly rigid bodies were switched back to deformable bodies once the forces and moments were equilibrated at the boat's COM. The waves were generated after the last switch. The goal was to speed up the process.

The configuration in Figure 16 was a severe case. The wave started with a valley. The wavelength was deliberately set in order to have most of the impact load transfer to the boat i.e. the boat was almost not supported by water when it hits the wave. All the constraints were at the boat's COM and none at the thrust. The constraints at the boat's COM were fixed in time. It is thus like the boat was going through the waves with a constant velocity.

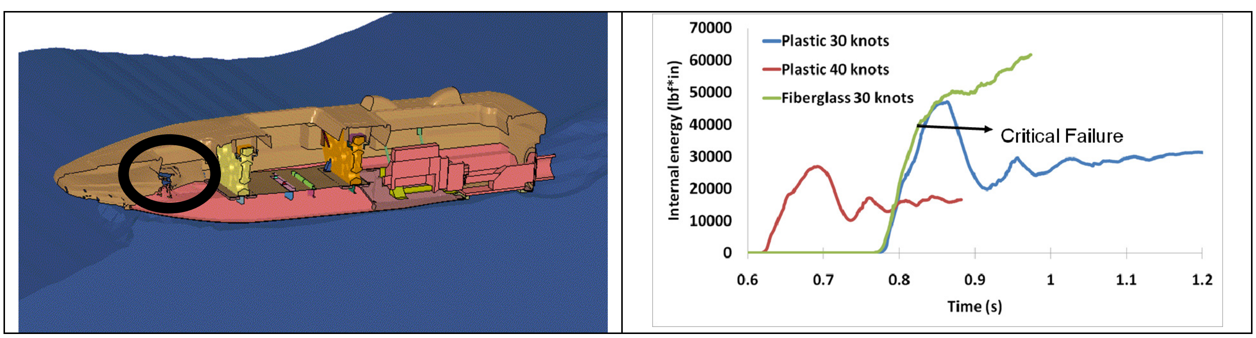


Figure 17: Results from the wave impact simulations

Figure 17 reveals a hull failure in the fiberglass boat at 30 knots. Internal energy comparison between plastic and fiberglass boats indicates the capability of the plastic boat to accumulate and restore impact energy in contrast to the fiberglass one which catastrophically failed. The plastic boat was simulated at 30 and 40 knots.

The first peak in the plastic boat curves is the strain energy accumulated during the bow impact. The plastic boat is flexing and it is returning to its original shape after the bow impact. The fiberglass boat is too rigid to flex, it fails and the forces are continuously transferred to non failed regions. Plastic boat travelling at 40 knots hits the wave closer to the crest than at 30 knots, causing less internal energy.

Conclusions

ANSYS tools were successfully used to generate a finite element model (FEM) of a complex assembly i.e. a novel 7-meter plastic boat. LS-PrePost was used to finalize the FEM and for post processing the results. LS-DYNA is a powerful numerical tool that was used for static, modal and dynamic analyses on the same model. Furthermore, it was demonstrated that the ALE approach in LS-DYNA can be used to model drop test into water but most interestingly, it is possible to generate realistic waves in order to simulate wave impacts.

Results from the drop test simulations and the modal analysis in Phase 1 clearly showed a critical flexural area. A modified design was run in Phase 2. LS-DYNA analysis results from Phase 2 predict that the plastic boat can go through 3-foot waves at 40 knots without damage while the fiberglass one has predicted damage at 30 knots. Overall, the numerical simulations show that high density cross linked polyethylene could be the new age of boating material.

Acknowledgements

The authors would like to thank Ian Do at LSTC for his very helpful suggestions and examples regarding the ALE/FSI capabilities in LS-DYNA. Grateful acknowledgements are given to Matthew Wadham Gagnon at Gurit Ltd for supplying material data.

References

1. LSTC (2007). LS-DYNA® Keyword User's Manual, Livermore Software Technology Corporation, CA, US.

2. ANSYS (2009). ANSYS FLUENT 12.0 Documentation, ANSYS, Inc., PA, US.

3. ANSYS (2009). ANSYS 12.0 Help, ANSYS, Inc., PA, US.