

Using Numerical Simulation to Analyze Ship Collision

Fuqiang Wu
Energo Engineering

Robert Spong
Energo Engineering

Ge Wang
American Bureau of Shipping, Corporate Technology

ABSTRACT: Nonlinear finite element method (FEM) is a powerful tool for analyzing ship collision problem and has seen more and more applications in recent years. The reliability of the numerical simulation results largely depends on the proper definition of the problem and careful control of some critical parameters. As part of a benchmarking exercise for a ship-to-ship collision project, the work presented in this paper presents a comparison between FEM numerical results and laboratory test results of a scaled double hull structure representing ship-to-ship collision/grounding scenarios. The general structural responses (i.e., load and energy results) and major failure modes determined from the FEM compared well to the laboratory test. However, some deviations were observed and attributed to specific FEM parameters. These parameters and their associated influence on the FEM simulation results are discussed in detail. The exercise confirms the validation of the numerical simulation technique in application on the ship collision problems and provides insight and guidance into some of the key numerical modeling procedures and controls required in the simulation of these complex structural interaction problems.

1. INTRODUCTION

Of recent, there has been a tremendous drive by industry to meet the present and anticipated future energy demands of many industrial nations. One of these energy initiatives by industry to meet these demands has been in the production and distribution of liquefied natural gas (LNG). Much of the LNG energy initiatives involve development of infrastructure centered on ship-borne transportation coupled with land-based or offshore reception/storage terminals which are typically located within or around existing ports or commercial centers.

With the introduction of these new LNG terminals comes the inevitable question of what are the potential risks related to the new activities in and around these ports and commercial centers. Of particular interest is the risk of ship collisions due to the additional vessel traffic. In order to identify and address these potential risks, industry has sanctioned studies to determine both the potential collision frequencies for various port traffic patterns as well as the related consequences should such an event occur.

During the course of a risk study, collision frequencies were determined, and limitations were recognized in the damage estimates using traditional approaches. The interested parties wanted to have a better understanding of collision resistance accounting for:

- Different striking vessels
- Various collision speed and angles
- Different structural arrangement
- Different failure criteria (i.e., loss of containment, out of service, etc.)

These requirements effectively drove the consequence analysis towards applying detailed finite element analysis (FEA) for calculating structural damage. However, prior to attempting such complex analysis, there was a need to conduct benchmark studies. This need was further reinforced because of the limited publication of mechanical testing and numerical analyses on collision resistance of ship structures.

This paper presents the frame work of this benchmarking exercise used to provide the starting point for a more detailed ship-to-ship

collision analysis. The objective of the exercise was to ensure that these detailed simulations capture the structural behavior in a proper manner and provide reliable results. Specifically, the benchmarking exercise set out to accomplish the following:

- Determine appropriate finite element modeling approach
- Determine appropriate values for some of the key analysis variables
- Predict structural resistance (i.e., force vs. penetration)
- Simulate the observed behavior (i.e., tearing, bucking, etc.)

This was done by comparing the FEA results to actual scaled mechanical test results.

2. NUMERICAL SIMULATION APPROACH

The 1990's was characterized by the remarkable advances in analytical solutions of various damage mechanisms in collision and grounding accidents (ISSC 2003). A series of analytical methods was developed and applied (Wierzbicki 1992-1999, Simonsen 1997, Paik et al. 1999, Wang et al. 2000, Brown et al. 2001, Suzuki et al. 2000, Pedersen 2002). These became the main theme of 1990's.

Non-linear finite element model (FEM) simulations are reliable and provide much more detailed information than other approaches. These are especially efficient in representing large bending of local plates, multi-axial stress fields, time-dependent strain hardening and strain rate effects on material properties, etc.

The rapid advances in the computer technology make numerical simulation, a formidable task only a couple of years ago, a viable choice now. Many powerful special-purpose FEM packages, such as DYNA3D, DYTRAN, ABAQUS and PAM (ISSC 2003, 2004), are now available that can account for large deformation, contact, non-linearity in material properties and rupture. Some recent supporting literature on numerical simulations of collision and grounding include Kuroiwa (1996), Kitamura et al. (1998, 2001), Endo (2001) and Tornqvist (2003).

Since structures behave in many complex patterns, many special modeling techniques are needed. Challenges involved in analyzing such a high non-linear problem include structural contact, criteria for

material's rupture, crack propagation, among others (Wang et al. 2003).

To analyze a collision or grounding accident involving high non-linearity, contact, friction and rupture, the explicit methodology is suitable. The required calculation efforts are fewer than the commonly used implicit methods. Convergence of calculations is much easier to realize.

3. BENCHMARKING PARTICULARS

This section describes the scaled test models used to compare with the numerical model as well as the finite element tools and general parameters used in the numerical analysis.

3.1 Mechanical Test

The prototype laboratory tests reported by Wang et al. (2000) were selected for benchmarking the numerical simulation approach. This series of mechanical tests was designed to cover various collision and grounding scenarios. The tests also provided valuable data related to the major failure mechanisms. Because of these attributes, it made an excellent test bed for investigating and comparing numerical simulations of this complex structural interaction.

Of the nine tests, three (tests P-50, P-200 and C-200) were selected for numerical modeling and comparison in this study. Test P-50 stands for the case of 50 mm radius (sharp) indenter cone positioned on plate; P-200 is the case of 200 mm radius (blunt) indenter cone positioned on plate; and C-200 stands for the case of 200 mm radius (blunt) indenter cone directly on the intersection of main supporting web members. These test cases were selected since they allow comparisons between indenter profiles (sharp vs. blunt) as well as between contact on plate only (i.e., between the support structure) versus contact at an intersection of the support structure. These differences in the test cases result in different failure modes being observed as well as resistance characteristics representative of ship-to-ship collisions. Table 1 provides a summary of the different scaled test parameters used in this exercise.

Table 1. Selected Benchmark Testing

Feature	P-50	P-200	C-200
Scaled Tests			
Indenter	50mm, sharp	200mm, blunt	200mm, blunt
Impact location	Shell plate (between supports)	Shell plate (between supports)	Intersection of support member
Initial failure mechanisms	Rupture in shell plate	Rupture away from contact zone	Buckling of support member
Numerical Model			
Mesh	Shell element	Shell element	Shell element
No. elements	18,000	15,000	17,000
CPU time	60 hours	20 hours	25 hours

Figure 1 shows a schematic of the test bed (upper image), and a picture of the actual test setup (lower image). The double hull section was bolted on to strong support frames. The indenters, which are polished rigid cones, were pushed downward, penetrating the double hull section. The rigid cones were pushed very slowly so that dynamic effects were considered to be negligible. Additional details on the scaled tests can be found in the appendix.

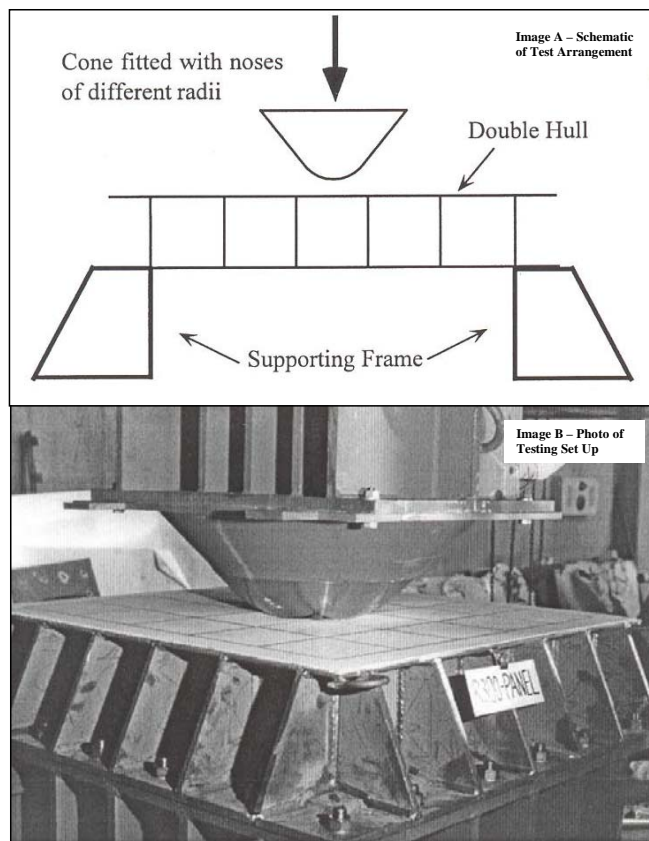


Figure 1. Test Setup Arrangement

3.2 Numerical Model

The general purpose finite element program LS-DYNA was used to recreate the tests through numerical simulation. Data pre- and post-processing was done by ANSYS. The explicit analysis solver used by LS-DYNA provided a powerful tool for a fast solution of this nonlinear collision problem.

The double hull test specimens were explicitly modeled as thin shell elements. Spacing between the web supporting members and the depth of the double hull were generally divided into eight elements. Coarser and finer element sizes were also tested in the simulation. Selection of element size is a tradeoff between accuracy and speed of calculations.

The indenter cones were modeled as rigid body. Figure 2 shows the meshed model used to represent the P-200 test case listed in Table 1.

The indenter cones were slowly pushed into the double hull test bed at a speed of 0.01 m/s to reflect the quasi-static testing condition. Attention in calculation was paid to the selection of the indenter speed to achieve a reasonable balance between calculation accuracy and cost (CPU time).

The dynamic energy was monitored and ensured that it was small and the majority of impact energy was dissipated in the deformed structures. In addition, structural damping was added to further damp out any possible dynamic vibration energy. Hence, the simulated progress was close to quasi-static results from the laboratory tests.

Vertical support boundary conditions were provided along the supporting frame locations. Bolted connection points were modeled as fixed points. The surface slip between the specimen and the test bed, which would likely occur under the large deformation, was not accounted for in the modeling. This aspect will be discussed further in the results and discussion section of this paper.

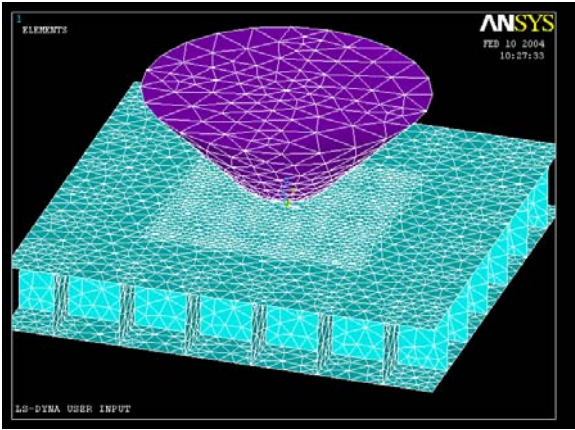


Figure 2. 3-D Image of FEM Model

Since only limited mechanical properties were available on the test set up, typical mild steel material properties were assumed based on Salmon and Johnson (1996). These properties are shown in Table 2. The stress and strain in the table are engineering stress and strain which need to be converted into true values for the simulation.

Table 2. Material properties of steel used in simulation

Yield Strength (N/mm ²)	Ultimate Strength (N/mm ²)	Rupture Strain	Young's Modulus (N/mm ²)	Poisson Ratio
282	400	0.35	200000	0.3

The material was modeled as kinematic hardening material with strain rate dependency. The strain rate effect is accounted for using the Cowper-Symonds model which scales the yield stress by the strain rate dependent factors, though this effect is not critical in a quasi-static test.

Material failure was considered in the model using strain failure criterion. If the calculated effective plastic strain for any element exceeds the predefined value, the element will be removed from the model and the simulation continues with the eroded model.

Under normal dry surface conditions, the friction coefficient on mild-steel-on-mild-steel surface is 0.74 for static friction and 0.57 for sliding friction. However, since the indenter had a polished surface, these values were reduced by about 25%. Therefore, the friction coefficient used in the simulation was 0.55 for static friction and 0.43 for sliding friction.

4. RESULTS COMPARISON

Three main parameters were used to compare the simulation to the test results. These parameters included:

1. Energy Absorption – This is one of the most commonly used parameters used in verification of these types of structural interactions problems. Specifically, it is the energy being absorbed as the indenter is pressed into the double hull test bed.
2. Applied Load – This is often used in more accurate and advanced calculation approaches, and provides a better indication of different failure events and the resulting load and deformation changes throughout the entire loading process.
3. Progressive Damage – This is the overall observed behavior of the structural interaction (i.e., rupture, buckling, large deformations, etc.).

To allow comparison between the test and numerical simulation results, the first two parameters, load and energy, are plotted with indenter displacement. To compare the progressive damage behavior, images of the double hull structure from the test and numerical simulation are presented.

4.1 P-50 Test Results

The P-50 test represents a case with a relatively sharp faced indenter driven between support members (i.e., on the plate). The load-indentation curves and absorbed energy-indentation curves are presented in Figure 3. For this case, initial failure is in the form of rupture of the outer shell due to sharp indenter. Once occurred, ruptures begin to tear toward the adjacent support structure. As the indenter is driven further into the structure the load begins to increase again as the indenter comes in contact with the adjacent support structure. Some initial buckling of the support members occurs resulting in some reduced load, but the indenter is now in contact with a large surface and the friction forces gradually increase as the plate and support structure is forced down and outward. The load continues to increase until the support member intersections eventually buckle and there is a notable reduction in load. Note that the indenter is driven to a depth 0.2 meter, which is depth of the double hull structure.

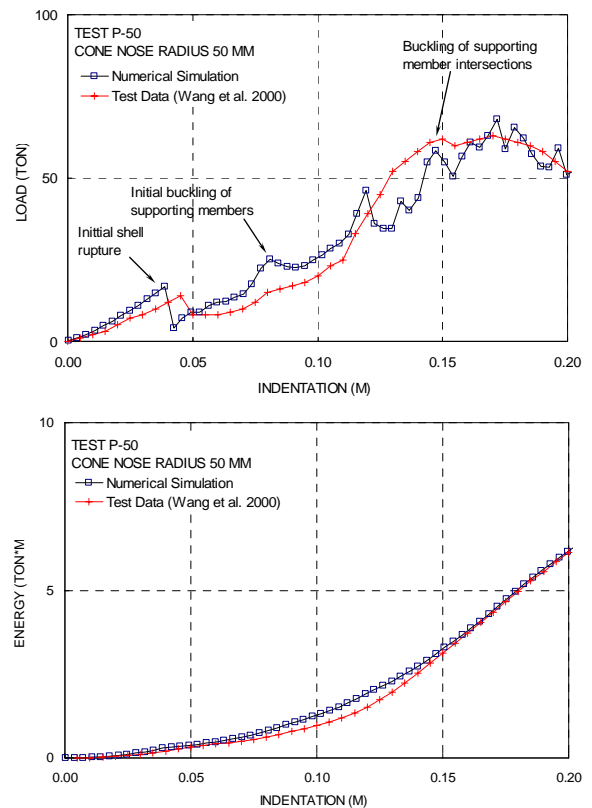


Figure 3. Load and Absorbed Energy Curves, test P-50

When reviewing the plots, the numerical simulation results compare well both with regards to load and energy absorption as the indenter is pushed further into the double hull structure. The results are a good indication of the rupture and buckling modeling parameters used in the numerical model are representative of the actual behavior observed in the test. Additionally, it is important to note that the load indentation curves for the simulation are relatively smooth with moderate jumps at particular failure points indicating both the time steps and the mesh size are adequate for the simulation.

Further comparison between the numerical and test results is in the form of the observed damage behavior. Figure 4 shows an image of

the simulation (top) and a picture of the damage in the test specimen. In this case it is noted that although the load and energy results matched, the rupture behavior of the outer shell, the rupture lines did not tear in the same direction. Specifically, the simulation predicted tearing of the plate toward the support member intersections and the test results tore toward the center of the support members. For this case, the rupture behavior of the outer plate was found to be driven by the element size and orientation at the initial contact point of the indenter. This is discussed in more detail in Section 5.3.

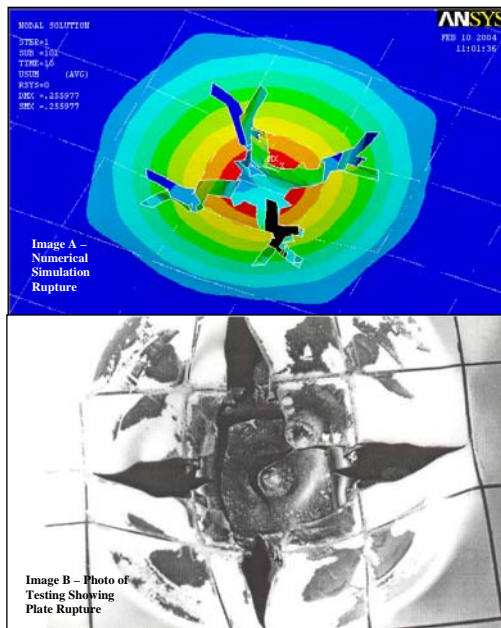


Figure 4. Progressive Damage Comparison, Test P-50

4.2 P-200 Test Results

The P-200 test represents a case in which a relatively blunt faced indenter is driven between support members (i.e., on the plate). The load-indentation curves and absorbed energy-indentation curves are presented in Figure 5. For this case, initial failure is in the form of buckling of the adjacent support members due to the large contact region of the indenter. Although buckling occurs, the load continues to increase due to the large contact area and load transfer within the double hull structure.

When reviewing the plots, the numerical simulation results compare well both with regards to load and energy absorption for the initial indentation (i.e., up to 0.12 m). However, there is a notable deviation in load as the indenter exceeds 0.12 m, with the numerical simulation predicting higher overall load and energy absorption than the test results.

Further investigation into this indicated that this is caused by the modeled boundary conditions. The numerical model was found to have more rigid boundary conditions than the bolted connections used in the tests. This was not observed to be an influencing factor for the P-50 test case (See Figure 3). However, unlike the P-50 test case where the sharp indenter tends to rupture the shell plating with only localized deformation of the test bed, the P-200 blunt faced indenter comes in contact with the adjacent supports which tends to cause a more global response (i.e., spreading deformation beyond the contact region) in the test bed. Hence the P-200 results are more dependent on the boundary conditions restraining the overall test bed particularly as the indenter passes midway through the double hull structure. This is also observed by the larger loads and energy absorption in the P-200 cases as compared to the P-50 case. This boundary condition problem will be discussed in more detail in Section 5.1.

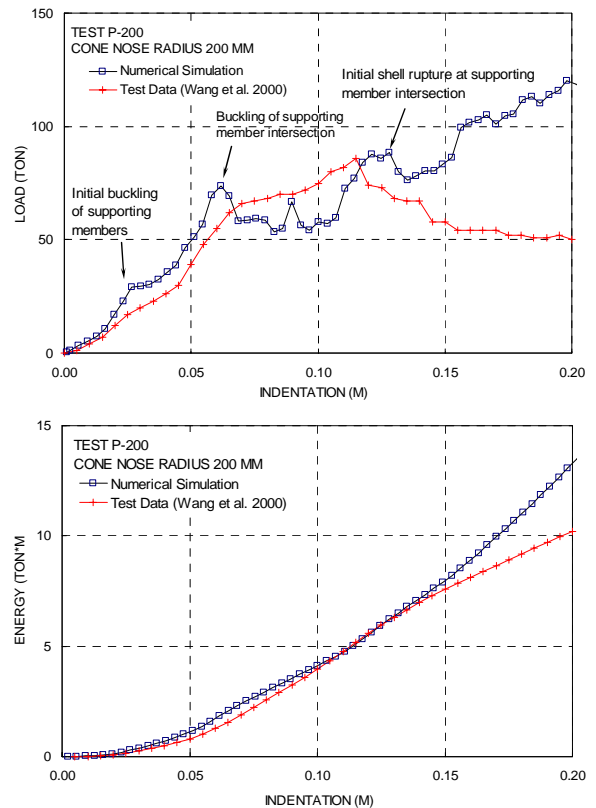


Figure 5. Load and Absorbed Energy Curves, Test P-200

Comparisons between the observed structural behaviors of the double hull were found to be similar. Figure 6 shows an image of the simulation (top) and a picture of the damage in the test specimen.

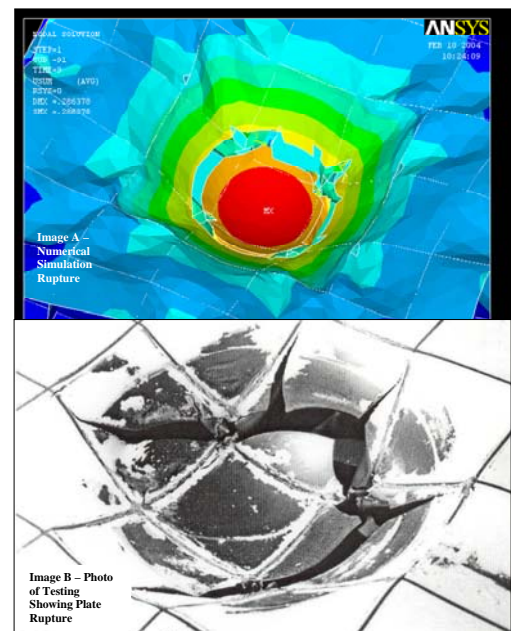


Figure 6. Progressive Damage Comparison, Test P-200

Unlike the P-50 case, the outer shell region at the center of the indenter is deformed and no rupture occurs. Instead rupture occurs along the outer regions outside of the adjacent support structure. In this case, the location of rupture and the general buckling and crushing behavior observed in the numerical model are very similar to the behavior observed in the test.

4.3 C-200 Test Results

The C-200 test represents a case in which a relatively blunt faced indenter is driven on top of an intersection of two support members (i.e., cruciform). The load-indentation curves and absorbed energy-indentation curves are presented in Figure 7. For this case, initial failure is in the form of local buckling of the support members beneath the indenter. Although buckling occurs, the load continues to increase due to the large contact area and load transfer within the double hull structure. Eventually, as the indenter is driven over 0.13m. At this point buckling of adjacent support intersections occurs, and there is a noted spread of the outer shell rupture and reduction in load.

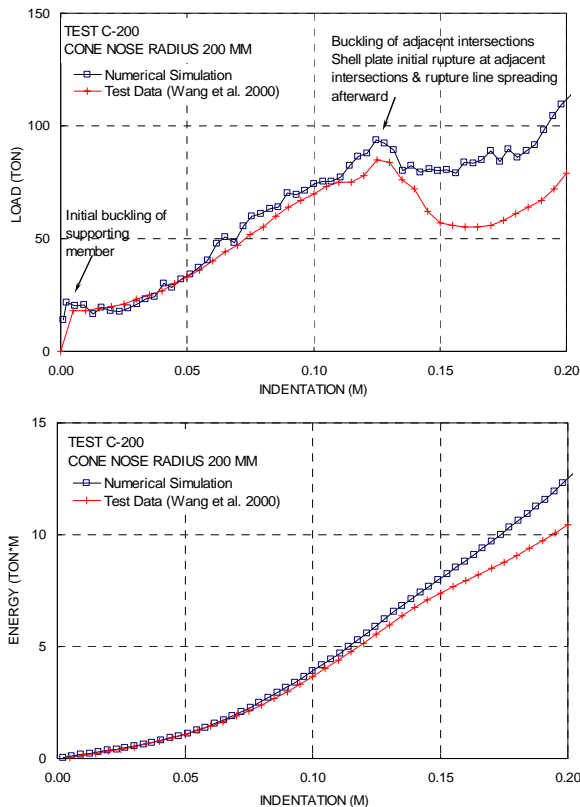


Figure 7. Load and Absorbed Energy Curves, Test C-200

After this point, there is a noted deviation between the test and the numerical results. The numerical simulation tends to predict higher load and energy absorption. This is similar to the behavior observed in the P-200 case and attributed to the modeling boundary conditions. This is discussed in more detail in Section 5.1.

Comparisons between the observed structural behaviors of the double hull were found to be similar. Figure 8 shows an image of the simulation (top) and a picture of the damage in the test specimen.

The outer shell region at the center of the indenter is deformed and no rupture occurs. Rupture occurs along the outer regions just inside of adjacent support structure. In this case, the location of rupture and the general buckling and crushing behavior observed in the numerical model are very similar to the behavior observed in the test.

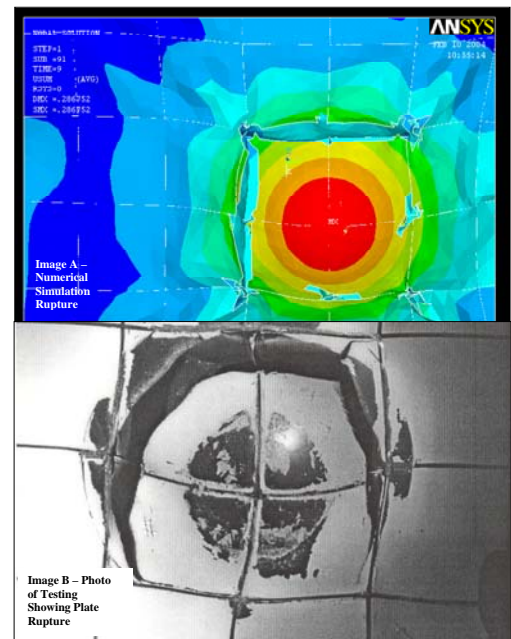


Figure 8. Progressive Damage Comparison, Test C-200

4.4 Summary

The comparison indicates the numerical simulation provides very similar results and is capable of capturing the distinctly different failure mechanisms: buckling and rupture. Noted deviations from the test results were investigated further to ensure adequate understanding on what the influential factors are.

5. DISCUSSIONS

As noted in the results section, some deviations between the numerical simulations and the laboratory test results were observed. Some of the key FEM parameters were found to influence the results, especially in the regions where behavior and load/energy results varied.

5.1 Boundary Conditions

Simulations deviate somewhat from test results for relatively large indentation. One example is the load-indentation curve of the P-200 test. After about 0.12 m indentation, loads dropped gradually in the test but continued building up in the numerical simulation. This is mainly because beyond this point, the boundary conditions assumed in the simulation have increasing influence on the analysis results.

Color-coded deformation contours as shown in Figure 6 clearly indicates the occurrence of large area deformation beyond the indenter cone contact area. Thus, very high membrane stresses developed near the support boundary, which in the mechanical test were bolt connection points.

In the FEM model, the bolt connections were modeled as fixed points at corresponding locations similar to the actual test set up. The gaps between bolts and holes and the possible slip between two bolted surfaces were not explicitly modeled due to the complexity of the connection. This difference does not have much effect in the initial loading stage when small to moderate hull membrane stress starts to build up. However, close to the final loading stage, as the base hull plate was extensively stretched, very large hull membrane stresses occurred. This in turn may overcome the friction resistance between double hull structure and supporting frame, resulting in gap closing and bolt stretching. These factors in the laboratory test will tend to reduce the boundary rigidity and result in an unloading trend. Since this physical situation was not explicitly modeled in the

numerical simulation, some deviations from the test results are anticipated. These influences were confirmed by conducting analysis runs with progressively less rigid model boundary conditions.

5.2 Mesh Size

When compared to typical FEM analysis for design purposes, non-linear simulations of a collision event use very fine mesh. To properly capture the local large deformation around a plastic hinge, a non-linear FEM simulation of collision may place 16 elements in one stiffener spacing. Using about 4 elements in one stiffener spacing is generally sufficient for a buckling or ultimate strength analysis. A conventional elastic FEM analysis for design verifications often uses 1 element for the same stiffener spacing. Alternatively, analytical formulae derived for evaluating structural damage characteristics (e.g., failure patterns) such as those summarized by Wierbicki (1992-1999) and Wang (2002) may be used to determine relevant mesh size.

In this study, different element sizes were tested for both accuracy and efficiency. When four elements were used for the depth of the double hull, the predicted load-indentation curve became unstable and fluctuating. The general buckling failure mode is still captured, but the damage progress is not very smooth. Eventually in the depth direction, 8 (triangular) elements were chosen because of the good predictions of load-indentation curves and the reasonable cost (CPU time) as listed in Table 1. More refined elements can also be used but this comes at the expense of increased CPU time.

5.3 Element Shape and Orientation

Element shape has some effects on the FEM results depending on the general mesh feature of the model. For shell type elements, quadrilateral elements are generally preferred over triangular elements because the former usually generates fewer elements with the same mesh size.

In this study, both quadrilateral and triangular elements were used in different test cases for comparison purposes. It was found that for the same mesh size, quadrilateral element model tends to rupture and buckle along the predefined node lines since these lines are in regular patterns and well defined as the plastic hinge lines. Conversely, triangular element models had a more irregular mesh pattern that is difficult to lead cracks to some specific direction. The general rupture and buckling behavior of the triangular element model tends to match the experiment test results better than the quadrilateral element model. It is important to note that this observation is only valid for the mesh size we used in the simulation. The difference between element shapes will tend to be reduced if a very fine mesh is applied, but again this comes at the expense of CPU time.

Element orientation also plays a role in determining where the tearing lines go once initiated. For example in Figure 4 of the sharp indenter case P-50, the simulated tearing lines do not fully follow those observed in the test. In the simulation, tearing lines have a tendency to run towards specific directions as a result of localized high stress caused by a combined effect of relatively coarser element sizes and element orientation. This can lead to distortion in tearing lines, especially for the sharp indenter P-50 case for which the element mesh size is relatively large comparing to the indenter cone size. Additional simulations were run and confirmed that a finer mesh with properly selected element orientation tended to result in tearing directions similar to those observed in the laboratory tests.

5.4 Rupture Strain Used in Analysis

A major challenge in nonlinear finite element analysis is the prediction and simulation of initiation and propagation of fracture (ISSC 2003). This is essential for members subject to extensive membrane stretching, while it is usually less important for axial crushing.

The most commonly used assumption is that fracture occurs when the "equivalent strain" reaches a critical value. Efforts have been devoted to calibration of this critical value from large-scale tests, real-life observations or tensile tests (i.e., Kitamura 1996, Wang et al. 2002). Element size has been viewed as perhaps the most critical parameters for determining critical rupture strain, and there are some studies addressing this topic (Simonsen et al. 2000, ISSC 2003).

It is noted that this assumption is convenient for both nonlinear finite element analysis and analytical solutions. However, it is not fully justified theoretically. More refined models that consider material behavior around the crack tip are being developed (Simonsen and Tornqvist 2004). It is expected that more rational rupture criteria will emerge that more properly reflect the material behavior and also are easy to be incorporated into a numerical simulation scheme.

The rupture strain used in this study has been listed in Table 2 with other material properties. Over the course of the exercise, different rupture strains were run to investigate sensitivities. Obviously, lower rupture strain results in lower resistance and lower energy absorption capacity. The percentage reduction, however, depends on the general failure mode and damage sequence involving the steel rupture and/or buckling. The rupture strain mainly controls the steel rupture failure. For the tested cases in this exercise, we found that a 30% reduction of rupture strain generally causes 20-25% reduction of energy absorption capacity near the final loading stage. The load-indentation history curve generally resulted in more spikes and sharp drops after the occurrence of first rupture.

5.5 Friction

As part of the exercise, different friction coefficients were evaluated. It was concluded that friction coefficients have only some, but not significant, influence on the general results of this benchmarking exercise. However, for real ship collision and grounding simulation, the friction effect may become quite significant when the contacts take place in a wide area and/or last long.

6. CONCLUSION

The recent industry needs for advanced numerical analysis tools have been driving application of nonlinear FEM for analysis of ship collisions.

This paper presented a study that aims to verify and benchmark numerical simulation approach. Due to the complex nature of the collision problem, a significant effort was made to ensure adequate understanding and proper selection of the many variables involved in this finite element analysis.

Comparisons were made between FEM numerical results and laboratory test results of a scaled double hull structure representing ship-to-ship collision/grounding scenarios. The general structural responses (i.e., load and energy results) and major failure modes determined from the FEM compared well with the laboratory tests. Some specific FEM parameters were discussed, including boundary conditions, mesh size, element shape and orientation, rupture strain and friction. The focus is to best match FEM simulation to the best possible with test observations.

This exercise validates the application of numerical simulation techniques to ship collision problems and provides insight and guidance into some of the key numerical modeling procedures and controls required in the simulation of these complex structural interaction problems.

ACKNOWLEDGEMENTS

This paper reflects work experiences the first two authors have had while working for ABS Consulting. The authors would like to express their gratitude to ABS Consulting and American Bureau of Shipping for supporting them in the development of this paper.

REFERENCES

- Brown, A. and Chen, D., 2001. Probabilistic Method for Predicting Ship Collision Damage, Oceanic Engineering International.
- Endo, H., and Yamada, Y., 2001. The Performance of Buffer Bow Structures against Collision (1st Report: Collapse Strength of the Simplified Structure Models). Journal of the Society of Naval Architects of Japan, Vol. 189, 209-217.
- ISSC, 2003. Committee V.3 Collision and Grounding. 15th International Ship and Offshore Structures Congress (ISSC), San Diego, August 11-15.
- ISSC, 2004. ISSC V.1 home page, www.issc-collisiongrounding.org, ISSC Committee V.1 Collision and Grounding, International Ship and Offshore Structures Congress.
- Kitamura O., 2001. FEM approach to the simulation of collision and grounding damage. The second International Conference on Collision and Grounding of ships. Copenhagen, Denmark, July 1-3.
- Kitamura, O., Kuroiwa, T., Kawamoto, Y. and Kaneko, E., 1998. A Study on the Improved Tanker Structure against Collision and Grounding Damage. Proceedings of the 7th PRADS, 173-179.
- Kuroiwa T. 1996. Numerical simulation of actual collision and grounding experiments. International Conference on Design and Methodologies for Collision and Grounding Protection of Ships, San Francisco.
- Paik J.K., Chung J.Y., Choe I.H., Thayamballi A.K., Pedersen P.T., Wang G., 1999. On rational design of double hull tanker structures against collision. SNAME annual meeting, Baltimore MD.
- Pedersen, P.T., 2002. Collision risk for offshore structure, Journal of Engineering for the Maritime Environment, Proceedings of the Institution of Mechanical Engineers, 216:M1, 29-44.
- Salmon, C.G., Johnson, J.E., 1996. Steel Structures, Design and Behavior. Fourth Edition, Harper Collins.
- Simonsen, B.C., 1997 Ship grounding on rock: I & II., Marine Structures, 10:519-84.
- Simonsen, B.C., Lauridsen, L.P., 2000. Energy absorption and ductile fracture in metal sheets under lateral indentation by a sphere, International Journal of Impact Engineering, 24, 1017-1039.
- Simonsen, B.C., Tornqvist, R., 2004. Experimental and numerical modeling of ductile crack propagation in large-scale shell element, Marine Structures, 17, 1-27.
- Suzuki, K., Ohtsubo, H., Sajit, C., 2000. Evaluation method of absorbed energy in collision of ships with anti-collision structure. Ship Structure Symposium on "Ship Structures for the New Millennium: Supporting Quality in Shipbuilding, Arlington, VA, 13-14 June.
- Tornqvist, R., 2003. Design of crashworthiness ship structures, Technical University of Denmark, Ph.D. thesis.
- Wang, G., Atita, K., Liu, D., 2000. Behavior of a Double Hull in a Variety of Stranding or Collision Scenarios. Marine Structures, Vol. 13 (2000), pp.147-187.
- Wang G., Spencer J., Chen, Y.J., 2002. Assessment of ship's performance in accidents. Marine Structures, 15, 313-333.
- Wang G., 2002. Some recent studies on plastic behavior of plates subjected to very large load. Journal of Ocean Mechanics and Arctic Engineering, ASME, 124, 3, 125-131.
- Wang G., Jiang DJ, Shin Y., 2003. Consideration of collision and contact damage risks in FPSO structural designs. OTC-15316. Offshore Technology Conference (OTC'03), 5-8 May 2003, Houston, TX.
- Wierzbicki, T., 1992-1999. Reports, joint MIT-industry program on tanker safety.

APPENDIX

The test series of Wang et al. (2000) includes nine quasi-static tests to investigate the behavior of a scaled double hull structure. These tests were designed to combine the pressing indenter cones of different nose radii with three major contact locations representing different collision scenarios.

Figure 1 shows a schematic of the test bed (upper image), and a picture of the actual test setup. The overall dimensions are shown in Figure A.1.

The double hull section was bolted on to strong support frames. The test pieces were constructed of 2.3 mm thick mild steel (yield point = 282 N/mm²). This includes both the two plates representing the inner and outer shell plating of a ship and the main support structures, which represent the web transverses and horizontal stringers. The depth of the double hull was 200 mm. The main support structure is constructed in a grid, evenly spaced at 200 mm in the transverse and longitudinal directions.

The indenters were pushed slowly downward and penetrated the double hull section. Five different indenters with spherical nose radii of 300, 200, 100, 50, and 10 mm were used to simulate various striking vessels.

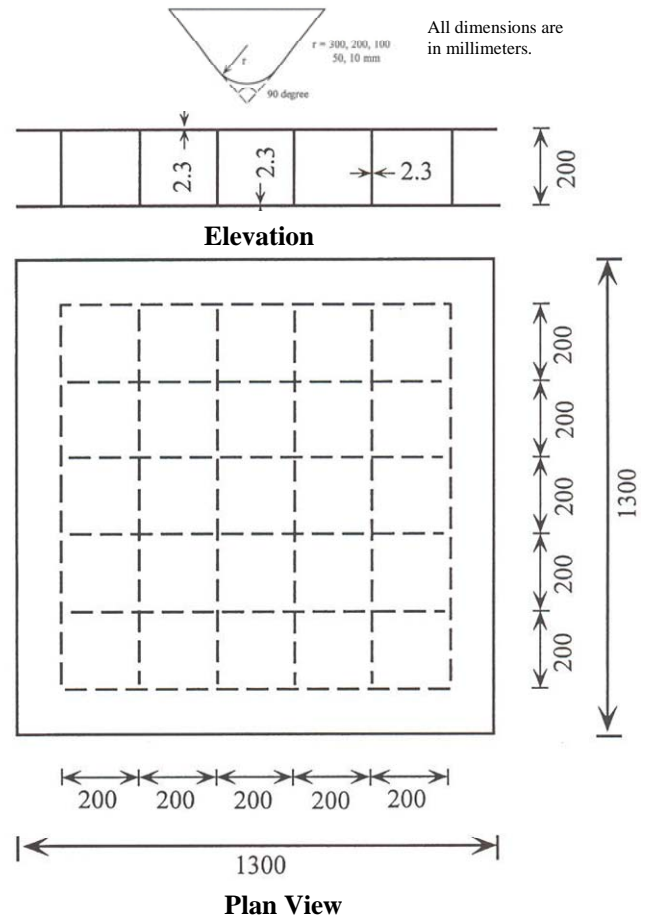


Figure A.1. Details of tested double hull