Fender System Selection Using ANSYS

Dr. Moujalli Hourani
Manhattan College
Civil Engineering Department
Riverdale, NY 10471

Mr. Reeves Whitney
Research Assistant
Manhattan College
Civil Engineering Department
Riverdale, NY 10471

Mr. Raymond Gizzi P.E
ECCO III Enterprises, INC
Yonkers, NY 10701

Abstract

This paper will describe complete investigations of the analysis and design of a proposed fender system. Finite elements analysis using ANSYS were performed on a wood fender system and a steel fender system subjected to impact load caused by a vessel collision. The impact load was converted into equivalent static load. The selection of the most efficient fender system is based on the principle on energy absorption of the system to the impact load. The ANSYS program was chosen to perform the analysis because it has an excellent feature that calculates the potential energy of the material. The wood system consists of 153 vertical rows of piles, a nose pile, 76 rows on the front face, 76 rows on the back face and walers spanning between the piles. The wood system was designed according to the NDS specifications. The steel system consists of 29 vertical rows of piles, a nose pile, 14 rows on the front face, 14 rows on the back face and steel beams (walers) spanning between the piles. The steel system was design according to the AISC specifications. The paper will present detailed procedure of the analyses and present comparative summary of the analyses. The paper will also describe the process used in choosing to adopt to construct the steel fender system.

Introduction

The proposed fender system was designed to replace an existing wood system. The main focus of our investigation is the selection of a fender system to resist an impact load caused by a ship collision. The system will be selected based on strength, stiffness and energy absorption. The maximum impact force caused by a ship striking the dock when berthing is based upon certain assumptions as the ship’s operation with respect to the angle and speed with which it approaches the dock. The analysis of the fender system presented here follows a customary assumption that the ship is fully loaded (displaced tonnage) and approaches at an angle of 10° to the face of the dock, as shown in Figure 1. Displaced tonnage is defined as the actual weight of the ship, or the weight of water she displaces when afloat, and may be either “loaded” or “light”. Displacement loaded is the weight, in long tons, of the ship and its contents, when fully loaded with cargo to the load line. (Ref.1.) Displacement light is the weight, in long tons of the ship without cargo, fuel, and stores. As shown in Fig.1.it is typical to assume that the bow of the ship will strike the fender, and only approximately one-half the tonnage will be effective in creating energy of impact to be absorbed by the fender parts.

The speed of approach will have to be assumed and it is here that the greatest uncertainty exists, particularly, since its effect on the energy varies as the square of the velocity (Ref.2.). The speed of the ship must be converted into component normal to the dock. Experience has indicated that the velocity of
approach will be between .5 to 3.5 knots at an angle of 10°. 1 knot = 1.689 ft/sec. In our investigation we used a vessel 360’X40’X12’, with an effective draw of 7 ft.

The total displaced weight is equal to: 360’X40’X 7’X 62.4 lb/ft³ = 6300 kips.

The kinetic energy of impact is \( E = \frac{1}{2} Mv^2 \) = \( \frac{1}{2} (W/g)v^2 \).

Where \( E \) = Energy, ft-kips
\( W \) = displaced weight of the ship, long tons
\( v \) = velocity of ship normal to dock, ft/sec
\( v = 3 \) knots is used in this analysis
\( g = \) acceleration due to gravity, 32.2ft/sec²
\( E = \frac{1}{2} Mv^2 = \frac{1}{2} (W/g)v^2 = \frac{1}{2} \times 6300/32.2 \times (3 \times 1.689 \times \sin(10°))^2 = 75.74 \) k.ft.

The energy to be absorbed by the fender system and dock is usually taken to be 1/2E, as the remaining one-half is assumed to be absorbed by the ship and water, because of the rotation of the center of mass of the ship around the point contact of the bow with the fender, which is assumed to be at one-fourth point of the length of the ship (Ref.3.)

**Analysis**

In this section of the paper complete analysis and design recommendation of the fender system will be presented. Based on a first order analysis, one wood fender system, and one steel fender system with comparable energy absorption will be investigated. A structural analysis and design computer program, RISA was used for the preliminary analysis and design of the fender system (Ref.4.) The program has the capabilities to design steel and wood systems according to the AISC (Ref.5.) and the NDS (Ref.6.) specifications respectively. The selected members by RISA were used in the ANSYS analysis to perform stress analysis on all the members and to calculate the potential energy of the systems. Two different types of analysis were performed on the steel and the wood fender. The first analysis dealt with the equivalent static load applied as a nodal load. In the second analysis, the load was applied as a patch static load acting on the waler as a load per unit length. The equivalent static load was calculated using the force acceleration principle by Derucher (Ref.2.) The descriptions and the results of the second solution of the wood and steel systems are presented next.
**A- Wood System**

The fender system under investigation consists of 153 vertical rows of piles, 76 rows on the front face and 76 rows on the back face. The pile placement and the spacing between the piles are as follows: 16 rows of piles at 1 ft spacing, 15 rows spaced at 1.166 ft, 15 rows spaced at 1.333 ft, 15 rows spaced at 1.5 ft, and 15 rows spaced at 1.666 ft. Each pile is divided into 6 3-D beam elements BEAM 4 as shown in Figure 2.

The point of fixity is assumed at 4 ft. below the mud-line.

---

![Figure 2. Mesh of a Typical Pile](image)

The material and section properties of the nose, pile and waler elements are shown below.

<table>
<thead>
<tr>
<th>Label</th>
<th>Material</th>
<th>Shape</th>
<th>Area (ft²)</th>
<th>Iyy (ft⁴)</th>
<th>Izz (ft⁴)</th>
<th>J (ft⁴)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PILE</td>
<td>WOOD1</td>
<td>SOPINDSS_R10.0</td>
<td>.5454</td>
<td>.02479</td>
<td>.02479</td>
<td>.04189</td>
</tr>
<tr>
<td>BEAM</td>
<td>WOOD2</td>
<td>SOPINDSS-9X12</td>
<td>.6788</td>
<td>.02838</td>
<td>.05195</td>
<td>.06197</td>
</tr>
<tr>
<td>NOSE</td>
<td>WOOD3</td>
<td>SOPINDSS_R15.0</td>
<td>1.227</td>
<td>.1255</td>
<td>.1255</td>
<td>.21201</td>
</tr>
<tr>
<td>PLATE</td>
<td>WOOD</td>
<td>SOPINDSS_5X10</td>
<td>.2968</td>
<td>.00348</td>
<td>.01550</td>
<td>.00978</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Label</th>
<th>E (k/ft²)</th>
<th>G (k/ft²)</th>
<th>N</th>
<th>Thermal</th>
<th>Density (k/ft³)</th>
<th>Fy (k/ft²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>W</td>
<td>230400</td>
<td>88617.6</td>
<td>0.3</td>
<td>.2E⁻³</td>
<td>0.035</td>
<td>648</td>
</tr>
</tbody>
</table>

SOPINDSS = Southern pine dense structural section
R10. = Radius of the cross section

SHELL 63 was used to model the 5 inch thick plate element representing the sheathing spanning between the walers. The allowable stresses of the wood elements are multiplied by 1.5 in accordance with the AREA specifications (chapter 8, section 32). (Ref.7.) The ANSYS Model of the wood system is shown in...
Figure 3. The model consists of 918 pile elements and 836 waler elements. The loads on the system consist of its self weight and a 44 kips equivalent static load of impact between the ship and the fender. The 44 kips load was applied as uniform load per unit length of 1.83 k/ft on 18 elements located between (X= 40.5, Y = 29.25, Z = 10.125) and (X= 38.5, Y = 33.75, Z = 11.25). The first order analysis of the system produced the following results:

a) The maximum displacement in the Z direction is equal to 21.45 inches at node 961 (X= 47.167 ft, Y = 39 ft, Z = 11.132 ft). See Figure 4.

b) The maximum bending stress, \( f_y = 2.22 \) ksi occurred in the waler element 1262 which spans between nodes 805 and 806.

c) The maximum bending stress, \( S_x = 3.57 \) ksi occurred in the pile element 48.

d) The weight of the materials needed for the design = 143.4 kips

e) The stress distribution \( S_x \) in the plate elements is shown in Figure 5.

f) The potential energy of the system = 36 k-ft. Figure 6.

g) The energy input = \( 1/2 E = 37.87 \) k-ft.

Figure 3. Wood model
Figure 4. Wood deflection

Figure 5. Wood stress
B- Steel System

The model for the steel system is shown in Figure 7. The material and the section properties are shown below:

<table>
<thead>
<tr>
<th>Member</th>
<th>Material</th>
<th>Section</th>
<th>Area</th>
<th>I_y</th>
<th>I_z</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>PILE</td>
<td>STL</td>
<td>W10X68</td>
<td>0.1389</td>
<td>0.00646</td>
<td>0.01900</td>
<td>0.00017</td>
</tr>
<tr>
<td>BEAM</td>
<td>STL</td>
<td>MC13X35</td>
<td>0.07153</td>
<td>0.00059</td>
<td>0.01215</td>
<td>0.00005</td>
</tr>
<tr>
<td>NOSE</td>
<td>STL</td>
<td>PI30X.5</td>
<td>0.32179</td>
<td>0.24316</td>
<td>0.24316</td>
<td>0.48632</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Label</th>
<th>E (k/ft^2)</th>
<th>G (k/ft^2)</th>
<th>ν</th>
<th>Thermal</th>
<th>Density (k/ft^3)</th>
<th>F_y (k/ft^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>STL</td>
<td>4176000</td>
<td>1606176</td>
<td>0.3</td>
<td>6.50E-04</td>
<td>0.49</td>
<td>7200</td>
</tr>
</tbody>
</table>
The allowable stresses of the steel elements are multiplied by 1.5 in accordance with the AREA specifications (chapter 8, section 32). The steel consisted of 29 piles, 14 piles in the front, and 14 piles in the back. The system was analyzed for the loading conditions that produced comparable kinetic energy with the wood system. The ANSYS model was analyzed due to an applied 65 kips. The 65 kips was applied as uniform load per unit length of 2.72 k/ft. on 6 elements spanning between (X = 40, Y = 29.25, Z = 10.125) and (X= 48, Y = 33.75, Z= 11.25). The analysis produced the following results:

h) The maximum displacement in the Z direction is equal to 19.2 inches at node 184 (X= 56 ft, Y = 39 ft, Z = 12.375 ft). See Figure 8.

i) The maximum bending stress, $f_y = 35$ ksi occurred in the pile element 55 which spans between nodes 9 and 38.

j) The stress distribution $S_x$ in the plate elements is shown in Figure 9.

k) The weight of the materials needed for the design = 124.5 kips

l) The potential energy of the system = 38 k-ft. Figure 10.

m) The energy input = 37.87 k-ft.
Conclusion

The design of a fender system is a very complicated and interesting problem for structural and geotechnical engineers. The design of a typical fender must satisfy strength, stiffness and energy absorption. This paper presented the detailed analysis of a proposed fender system using ANSYS. ANSYS provided us with a unique capability to determine the potential energy of the system in addition to its outstanding stress analysis capabilities. Based on the results concerning the potential energy of the two system obtained by the
ANSYS analysis, we selected the steel system to be constructed. The wood system consisted of 153 piles, maximum load capacity of 44 kips, maximum deflection in the Z direction of 21.45 inches, and total weight of materials = 144 kips. The steel system consisted of 29 piles, maximum load capacity of 66 kips, maximum deflection in the Z direction of 19.2 inches, and total weight of the materials = 124 kips. In addition to all the advantages of the steel system, the major factors in our decision were the constructability and the cost issues.

References
2- Bridge and Pier Protective Systems and Devices. Kenneth Derucher & Conrad Heins, Marcel Dekker, 1979
4- RISA 4.1, RISA Technologies, Foothill Ranch, CA 92610
7- American Railway Engineering Association, Manual of Railway Engineering, 1997
8- ANSYS 7.1, Educational Version