International Journal of Engineering Science Invention (IJESI)
ISSN (Online): 2319-6734, ISSN (Print): 2319-6726
www.ijesi.org ||Volume 9 Issue 11 Series II || November 2020 || PP 07-15

A Case Study on construction repair of Syncrolift from Ship Fallen Accident

Walid H. Shalaby
Department of Civil Engineering, Higher Institute of Engineering and Technology, King Marriott, Alexandria3135, Egypt

ABSTRACT *: During Ship landing from Syncrolift, an accident has occurred due to wire broken, ship has been fallen into water from small height, broken wires are two from thirty-two wires. Although no damage has occurred to the ship (weight 3500 ton) and no injury to the ship operating group, the steel deck has been partially damaged. This study is addressed to evaluate the existing situation and choose the optimum repairing method for the system. Instantly, survey impact has been performed to record the generated deformations and discrepancies from the original pattern. Applying FE Method by Abaqus software, the fallen action has been simulated, upon which the damaged parts have been identified. Approximate analysis shows one of the main girders needs full jacketing (for the web and flanges). To optimize the real needed work another exact analysis has been performed showing the exact location of failed areas and damaged stiffeners. Partially removal of the girder top flange has been performed, new elements have been welded, and deflected stiffeners have been replaced.*

KEYWORDS –*Abaqus, Finite Element, Steel structure, Strengthening, Syncrolift*

Date of Submission: 08-11-2020 Date of Acceptance: 23-11-2020

I. INTRODUCTION

A unique idea that raises a ship out of the water to make required repairs to the underwater portion of the hull of the ship is the Syncrolift System. The underwater part of the ship was revealed prior to its construction through the use of floating dry docks, graving docks or marine railways as shown in Figure 1, all of which were more costly to construct and service[1].

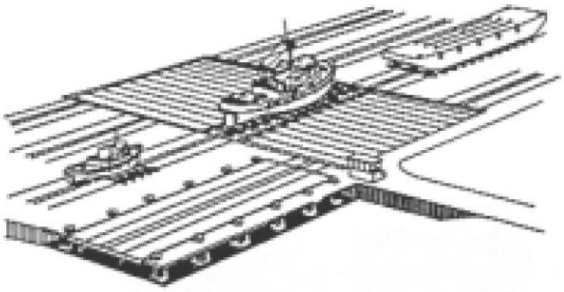


Figure 1: Transfer rails system

The mechanism essentially raises a ship out of the water, moves the ship to shore, and then returns the repaired ship to sea. The device consists of a large elevator that can be lowered into the water, mounted above it by a ship, and then the elevator and the ship can be raised vertically to the shipyard's ground level as shown in Figure 2.

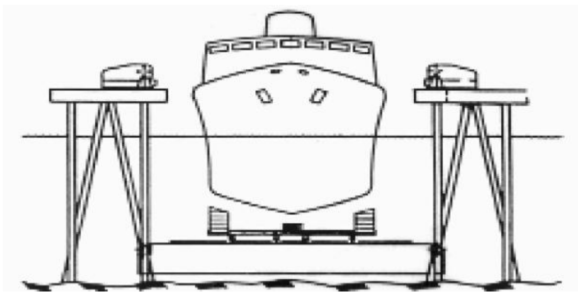


Figure 2: System main components

A structural steel platform, a selected number of electrically driven wire rope hoists to lift and lower the frame and a central motor control center to operate the system are three main components. The hoists are supported by fixed structures, most of which are normal marine piling systems.

During returning ship to sea two wire robs have been broken, cause a fallen secondary unit of the system into sea, the other main unit has been deflected and twisted. While Ship has been laid safely on the water, the system deck and the electro-mechanic units have malfunctioned due to the damage.

II. STRUCTURAL EVALUATION

The evaluation of the degree of damage to the system is done through evaluation of the structural arts affected, followed by FE modeling for accident simulation. which is an essential stage before taking the decision on the possibility of repair or even partial reconstruction of the steel deck or whether to resort to demolition, and it should stand on some important details.

2.1 A general description of the Steel Deck

It consists of 31 adjacent panels, 16 is rigid (fixed) panel connected directly to four lifting motors, two motors at each side through powered wires. Other 15 panels are articulated (secondary movable) type, rested simply on the rigid units. All the deck is covered with clamped wooden boards.

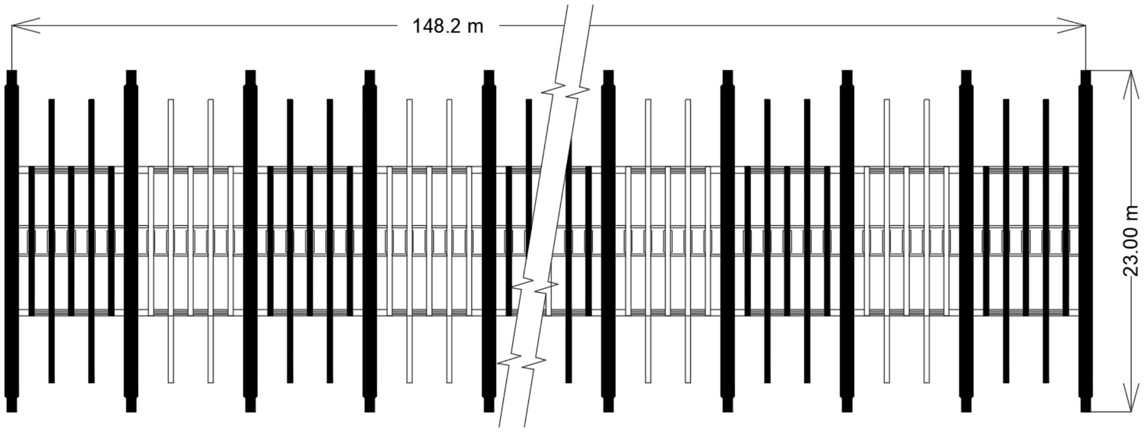


Figure 3 : Platform Assembly

Rigid unit consists of two main transverse girders (1B, 1C) “I” section with depth 2.2 m, connected directly to powered wires with rigid part containing coil with multi lanes for the wires. The two main transverse girders are connected to each other by two longitudinal beams (2B, 2C) “I” section, with depth 1.0 m. These beams support five transverse beams, (two 3B, three 3C) as shown in Table 1.

Table 1: Steel Deck Cross section dimensions of members

ID	Length (mm)	Web-Depth (mm)	Web-thickness (mm)	Flange-width (mm)	Flange-thickness (mm)
1B, 1C	23,100	2,190	25	820	45
2B, 2C	8,400	1,000	20	382	40
3B	20,000	359	6.25	120	9.375
3C	10,500	359	6.25	120	9.375

2.2 Survey Impact on the Deformed unit

After lifting the articulated unit from the sea ground, the supported adjacent panel has been lifted to maintenance levels and supported by four steel blocks. Visual check shows deformations in the main side girder (just beside the articulated fallen unit), other deformations are shown in the transverse bracing elements as shown in Figure 4.

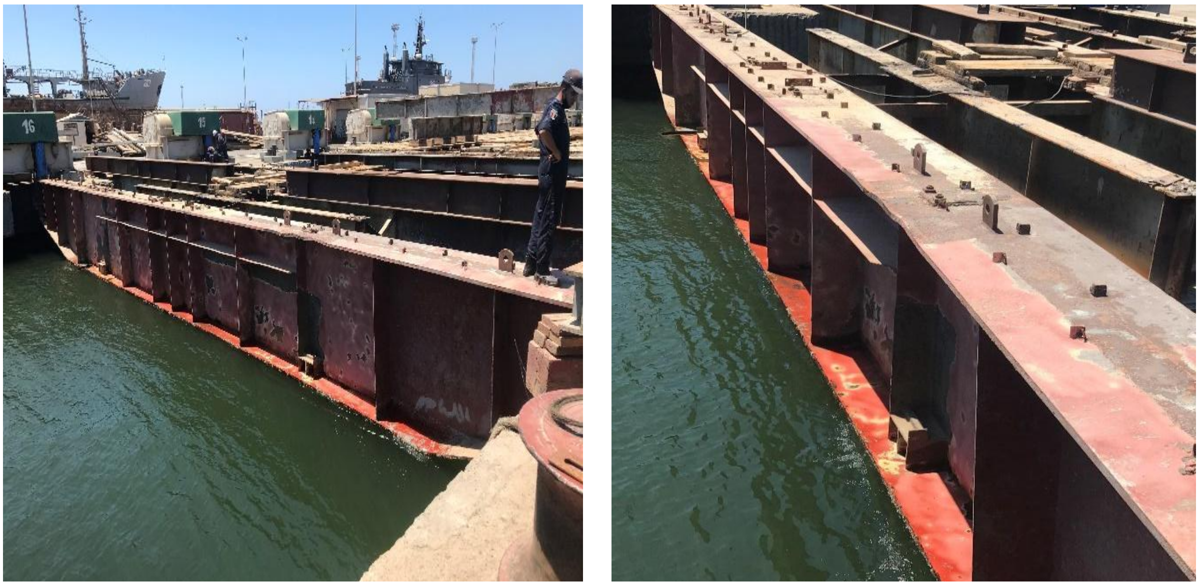
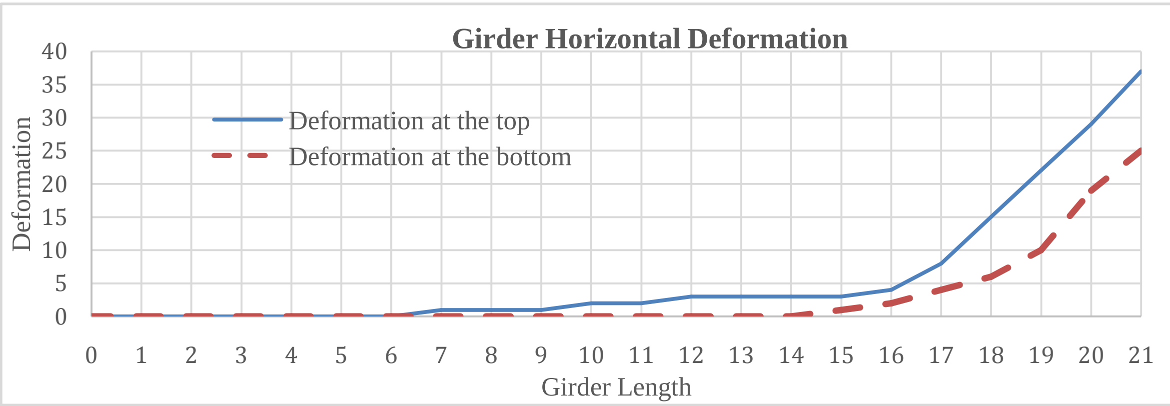


Figure 4: Deformation in the main girder

Detailed measurements have been recorded using total station devices for all the unit members. Main concern is to determine exactly the developed deformation at each region of each member. The records show severe bending in the main girder, resulting deformation at the girder end with 40 cm horizontally and 7 cm vertically, associated with obvious buckling at the top flange, and slightly buckling in the lower flange. Web stiffeners are buckled in this region. It is nothing to see the deformation in the girder occurred as it has been bent around inclined axis, due to the lower connection with (2C) girder as shown inFigure 5.



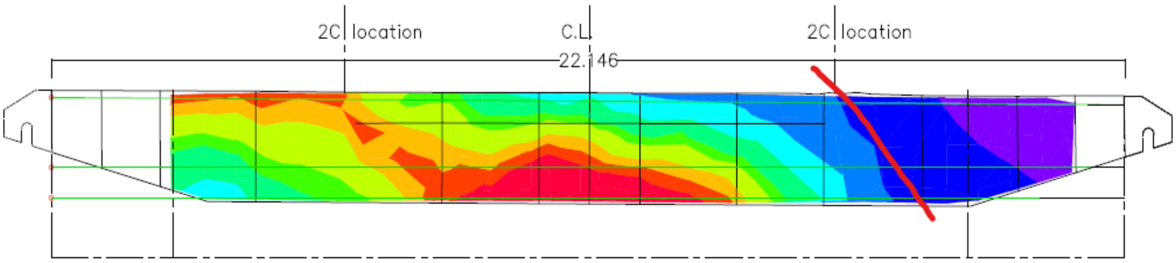


Figure 5 : Deformation records in the girder's web

This proof that plastic hinge has been developed along the inclined axis. The intensity is significantly high at the top and decreases gradually towered the bottom flange. It can be seen clearly the top flange is severed buckled, while the bottom flange is slightly affected as shown in Figure 6

Figure 6 : Top and Bottom Flanges buckling pattern

The vertical stiffener and the attached horizontal stiffener have been dramatically deformed, which require removing it completely during the repairing process. In the same manner, the transverse bracing above the longitudinal beam (2C) have been completely buckled.

III. STRUCTURAL ANALYSIS OF THE AFFECTED UNIT

1.1.1 Global and Discrete FE Model

Design available data has been inspected at site, and used to assemble generic FE Models. One commercial FE program has been used. Utilized material is A572[2], all mechanical properties of the material have been introduced in the model. No loading has been identified, occurred deformation from the accident at the (1B) end (40 cm horizontal and 7 cm vertical) have been applied to the model. Linear analysis has been applied. Analysis results show no compatibility with the site, either the deflection shape, or the developed straining actions. Because linear analysis cannot present cumulative yielding of material, and present plastic hinge developing.In the same manner the large deformation has significant effect on the geometry of the model. Analysis has been repeated as nonlinear analysis, with including material and geometrical nonlinearity[3], [4].Analysis results present the corresponding straining actions, and show considerable matching with the site investigations. Analysis output data have been used to perform prompted design check;design has been performed to all elements of the damaged unit. Check results show the mentioned adjacent (1B) girder is yielded to some extent in the determined region, the stresses in the section are nearly double the design limit. The analysis determines exactly the yielded part in the mentioned region for the whole girder section. According to design output, repairing choices are either to remove the whole region, or conduct complete jacketing. First choice requires moving the unit onshore to perform the required working shop activity from cutting and welding in quality. Although second choice can be performed in place, still a difficult task and because of the extra used material it can cause lack of stability to the unit center of gravity.

1.1.2 Microscopic FE Model

To achieve exact determination of the yielded portions, extremely detailed analysis has been performed using Abaqus platform[5], [6], [7], this type of analysis meshing all the model members to finite elements. Wherever any stresses above the accepted one, region can be identified. It helps to optimize the required repairing work from one side, and present any veiled elements require strengthening or removing such as stiffeners,tie elements, holes, and any additional pieces linked to unit members. Assembling such FE model requires sequential steps have been determined by the program methodology.

1.1.2.1 Modeling Part

While Abaqus is capable of performing a complete analysis of the interaction between unit elements under the dynamic loading, this level of complication and expense was not deemed necessary in this case. The analysis assumed that the damaged rigid unit would conform to the existing shape and, therefore, did not have to be explicitly modeled. A reasonable assumption given that the rigid unit unseats from the adjacent girder rim and deflates under the applied load. Also, because the exact dynamics of the impact to the unit were not well defined, a static equivalent loading approach was used.

1.1.2.2 Material Part

Exact material properties for the subject wheel are available through the designed drawings. The material is compiled with ASTM type A572- Gr50 steel. The relevant stress strain curve for it was used, with a yield strength of 361 MPa and an ultimate strength of 488 MPa at 19% elongation. Multiple points were defined in the plastic card between the yield and ultimate strength to give reasonable fidelity in the post-yield material definition.

1.1.2.3 Element types and meshing

All of the unit elements are modeled with solid elements, the type is C3D8R. Details of the mesh are shown inFigure 7, it meshed in structural technique.

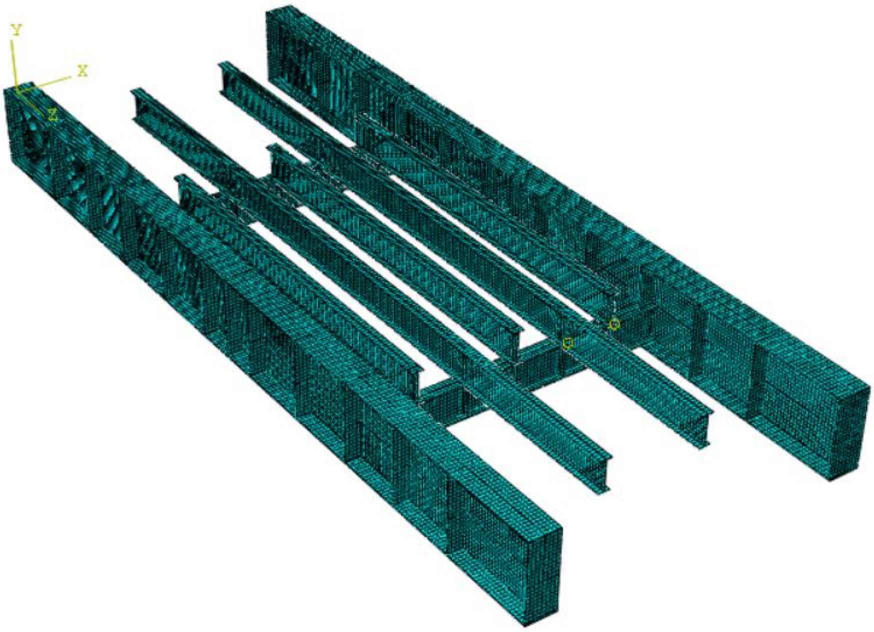


Figure 7: Damaged unit Assembly with meshing

FE model had roughly 45,456 elements with 530,592 degrees of freedom and the analysis has been run on a PC with five cores processor.

1.1.2.4 Boundary conditions and loading

A Cartesian coordinate system is defined for the definition of boundary conditions. The main two girders end are held fixed. Loading was accomplished with displacement control which is, of course, much more stable for nonlinear analysis. The deflected end has been moved and rotated with MPC constraint to simulate the accident. The loading steps are two, first to simulate the accident by moving and rotating the girder end to the existing values. Second step can be considered as a part of the repairing stage, it presents the girder straightening to original shape. This process has been presented using hypothesis time, the first unit is the accident stage, which is the executed deformations period. Next unit is the repairing stage which girder is straightened.

1.1.3 Results and Discussion

Two-point series have been chosen on the lower and upper flange, to track the displacements and stresses. The time-deflection curves for the top and bottom flanges chosen points are shown in Figure 8 and Figure 9 respectively. It is obvious to see that, first repairing step has achieved straightening with tolerance less than three millimeter, which is accepted.

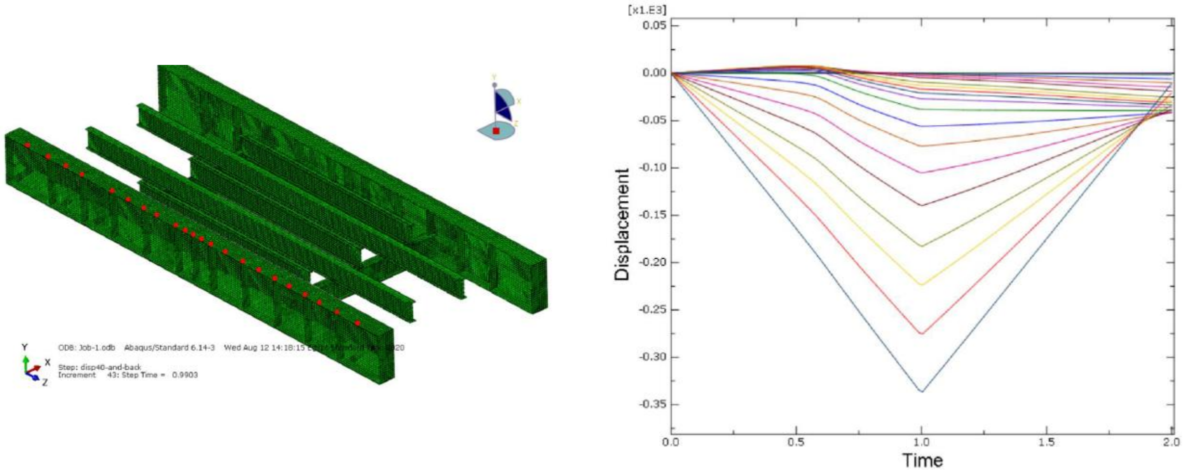


Figure 8: Deformations at the top flange

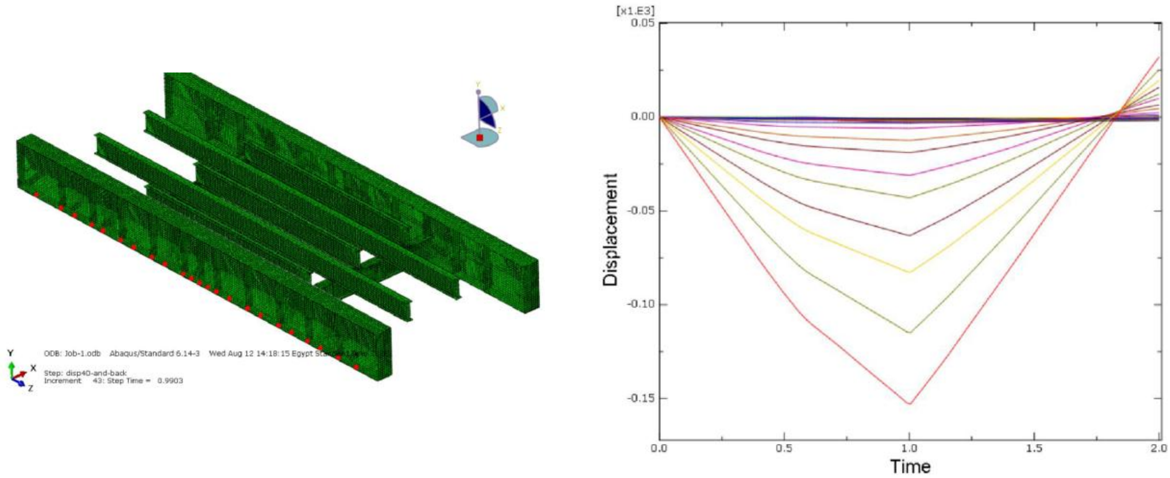
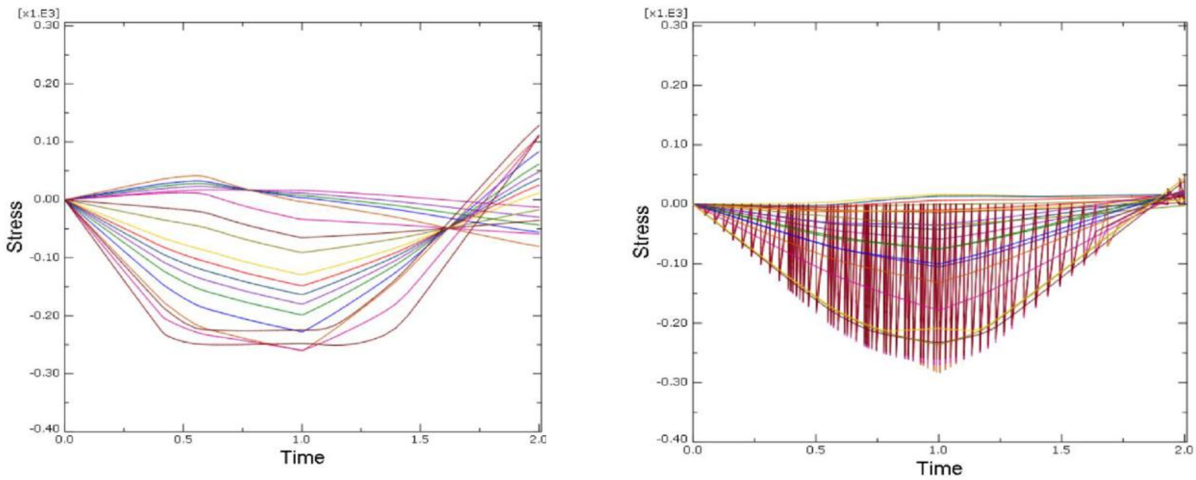


Figure 9: Deformations at the bottom flange



a- Stresses at the top flange **b- Stresses at the bottom flange**

Figure 10 : Developed and residual stresses at the top and bottom flanges

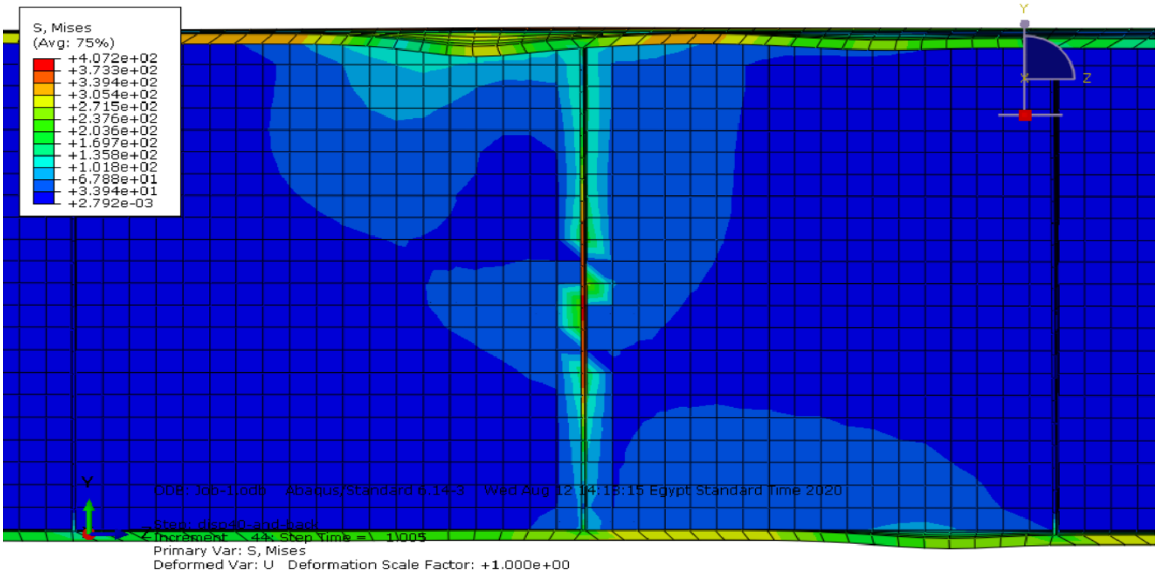


Figure 11: Residual stresses at the damaged region

The von Mises stress result under the occurred displacement is shown inFigure 10. The stress-timecurves show obviously when the girder is straightened with residual stresses in the top flange exceed 20% of the material yield stress while it is less than 5% in the lower flange. The top flange is significantly yielded, and it is required to be replaced while the bottom flange contains slightly residual stresses, which is within the design margin. The residual stresses in the web is less than 3 % and all the straining actions have been grabbed by the buckled stiffeners as shown in the detailed view in Figure 11.

IV. STAGES FOR CONSTRUCTION REPAIRING

FE Model analysis gives forensic simulation for the accident and straightening process[8], [9], upon that repairing stages have been identified as shown in Figure 12. Starting with removal of the bucked region on the top flange with length 2.0 m. Followed by static straightening with using chains hoists to return the girder tip to original position.

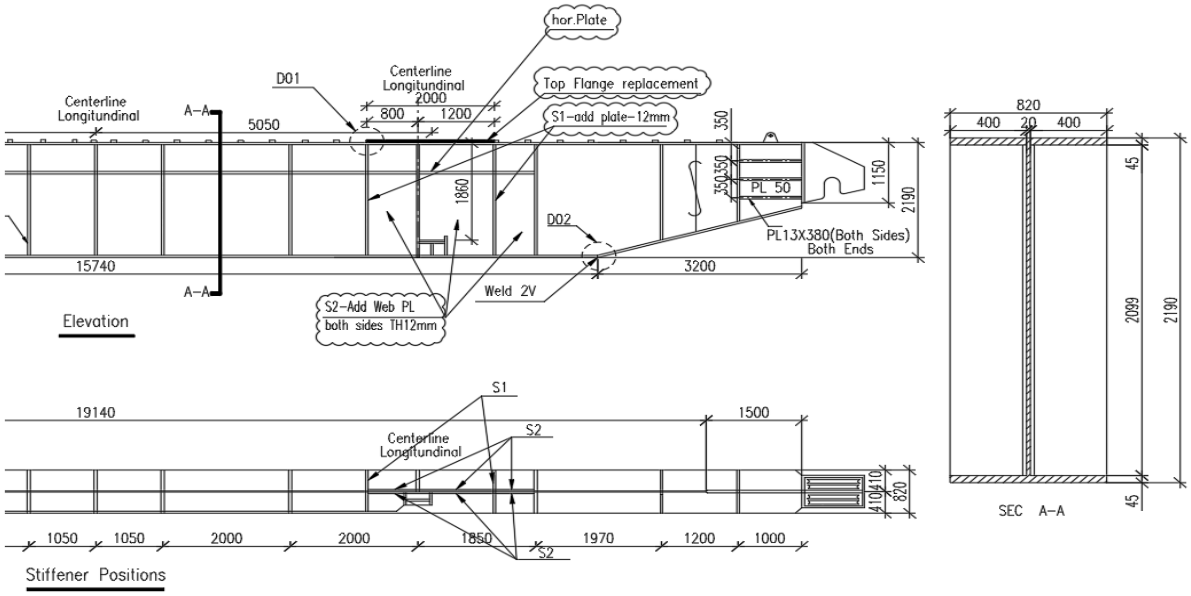


Figure 12: Repairing details

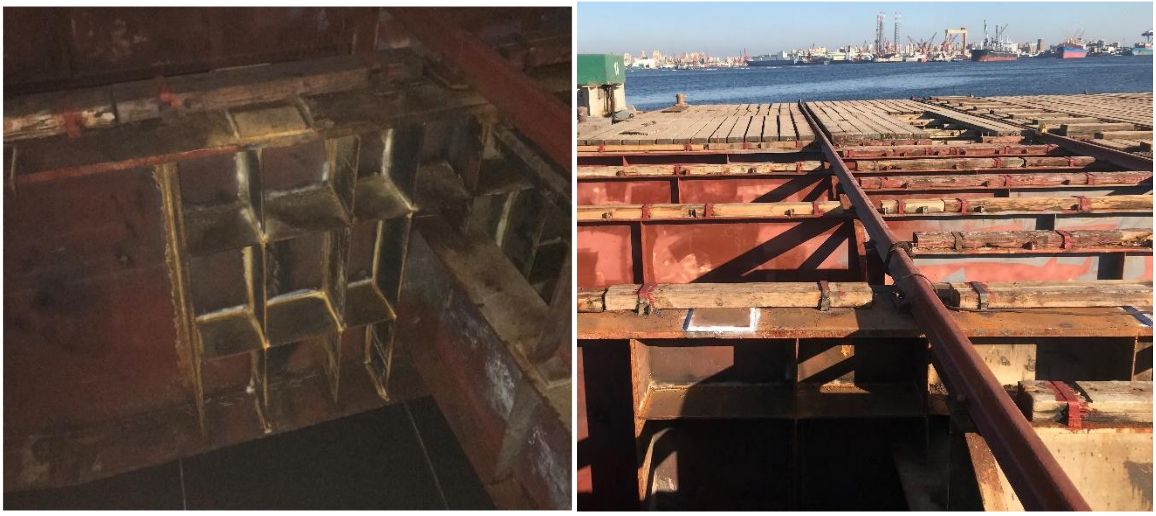


Figure 13 : Additional stiffeners

Heat treatment is totally forbidden and that was controlled by the QC sector. Grove type weld has been used to connect the new part of the top flange and the girder's web; this avoids adding a Doubler plate to the girder's web. To overcome this difficulty, stiffeners have been added in grid shape as shown in Figure 13, this decreases significantly the expected stresses in the girder's web.

a- antiquated ship

b- Ship with total weight 3500 ton

Figure 14 : Ships Loading

Loading tests have been done, especially on the repaired unit, by loading an antiquated ship as shown in Figure 14-a. Deflection has been recorded in loading and unloading stages, and compared with the permissible limits. After getting confidence from the behavior of the system as all, the normal ship has been loaded and moved to inshore with a total weight of 3500 ton as shown in Figure 14-b. Deflection values have been recorded during lifting the ship and moved to the working ground. All values are within the limits as before the accident.

V. CONCLUSIONS AND RECOMMENDATIONS

The structure system of the steel deck has a unique feature, which is the free connection between the rigid units and articulated units. The type of connections prevents catenary action during the accident. Using survey data in such accuracy, has guided the performed analyses in many stages to suit the existing situation. The repairing work could be tremendously increased if the microzoning FE model is neglected. Although linear analysis is fast track one, its result is diverted from the right solution. The nonlinear analysis is significantly reflect the existing condition. Work duration has been shortened with 50% without any impact on the efficiency and performance of the system.

REFERENCES

[1] T. E. A. S. Award, "Basic Principles Of The Syncrolift," 2019. [Online]. Available: Https://Shiplift.Com/Brochures/.

[2] A. A. State, Standard Specification For High-Strength Low-Alloy Columbium-Vanadium Structural Steel, West Conshohocken: Astm International, 2003.

[3] M. M. V.-B. M. D. M. A. C. J. G. Georgios E. Mageirou*, "Comparison Of Linear And Nonlinear Analysis Methods For Steel Columns," In *4th Gracm Congress On Computational Mechanics*, Patras, Greece, 2002.

[4] P. W. D. Authors: Kythe, An Introduction To Linear And Nonlinear Finite Element Analysis, Birkhauser, 2004.

[5] Simulia, Getting Started With Abaqus, 6.14 Ed., Ri, Usa: Copyright Dassault Systèmes, 2014.

[6] Simulia, Analysis User's Guide, Providence, Ri, Usa: Dassault Systèmes, 2014.

[7] Simulia, Analysis User's Guide Volume Iii: Materials, Providence, Ri, Usa: Dassault Systèmes, 2014.

[8] I. Aisc, Specification For Structural Steel Buildings, Chicago, Illinois, 2010.

[9] C.-M. U. B. Chi, "Effect Of Straightening Method On The Cyclic Behavior Of K Area In Steel Rolled Shapes," La Jolla, California 92093-0085, California,, 2001.

Walid H. Shalaby. "A Case Study on construction repair of Syncrolift from Ship Fallen Accident." *International Journal of Engineering Science Invention (IJESI)*, Vol. 09(11), 2020, PP 07-15. Journal DOI- 10.35629/6734