Test based finite element analysis of wire meshes

J Straub[1], L Bošković[1,2], T Bogatzky[2] and U Dornbierer[3]

[1] Materials Engineering Laboratory, HTWG Konstanz University of Applied Sciences, Alfred-Wachtel-Str. 8, D-78462 Konstanz, Germany
[2] WITg – Institute for Material Systems Technology Thurgau, Konstanzerstrasse 19, CH-8274 Tägerwilen, Switzerland
[3] Industrial and Safety Solutions, Geobrugg AG, Aachstrasse 11, CH-8590 Romanshorn, Switzerland

Corresponding author e-mail: joerg.der.straub@googlemail.com

Abstract. As fish farming is becoming more and more important worldwide, this ongoing project aims at the simulation and test-based analysis of highly stressed wire contacts, as they are found in off-shore fish farm cages in order to make them more reliable. The quasi-static tensile test of a wire mesh provides data for the construction of a finite element model to get a better understanding of the behavior of high-strength stainless steel from which the cages are made. Fatigue tests provide new insights that are used for an adjustment of the finite element model in order to predict the probability of possible damage caused by heavy mechanical loads (waves, storms, predators (sharks)).

1. Introduction

In view of climate change, overfished wild fish populations and the rapid growth of the world population, which is expected to reach almost 10 billion people in 2050, the issue of nutrition is increasingly becoming the focus of future challenges [1, 2]. One way to continue the supply of important proteins is aquaculture, which refers to the growth of aquatic organisms (fish, crustaceans, shellfish, etc.) and now accounts for around 50% of global production. The per capita consumption of e.g. edible fish in the early 1960s was 9.0 kg and rose to 20.2 kg, more than twice as much in 2015. On the one hand, this makes the urgency for solutions tangible and, on the other hand, it strengthens the need to promote aquaculture [3]. Since the area on the mainland and near the coast lines is limited and the number of fish produced in aquaculture is increasing, breeding will have to be moved to deeper waters in the future, where extreme environmental conditions may exist [4, 5]. Due to high mechanical loads (for example waves, storms, predators (sharks, seals)), the fish farms and thus the cages used are subject to high stress - often beyond the limits of their requirements. Damage, i.e. wire breaks, can occur and thus fish can escape. In the present work a simulation model was developed, which can partly represent real processes and provides a basis for further simulation models and experimental tests. The challenge in this project was not only to create a suitable numerical model, but also to collect data using quasi-static and dynamic tensile tests in order to understand the mechanics in the small wire contact area of cages. The focus of this work is to complete the database of the simulation software with important insights from experimental data in order to refine the virtual model and to adapt it to the available observations. Finally, this package will provide the basis for the combination of non-linear plastic and linear elastic calculations in order to be able to carry out further FE investigations on larger wire-mesh-simulation-models in short computing times in the future.

2. State of the Art

Finite element analysis is a widely used method to investigate linear and non-linear problems in advance by simplified reconstruction of reality. Simulations of fish farming were performed in several different projects. Frequently, these studies involved the consideration of entire systems, i.e. a cage and its mooring lines to a platform. The numerical models that are analyzed can be cylindrical, but also angular, rigid, semi-rigid or flexible [6, 7]. Furthermore, the current trend is to replace conventional open cages with closed cages [8]. With this wide variety of cage designs, the movement of the cage caused by waves and currents, its deformation, and the resulting tension in the mooring lines from net to platform are investigated. Often a comparison is made between experimental data and the predictions made by simulation models [7]. Another often investigated and not to be neglected topic, is biofouling, which takes on different dimensions depending on wave and flow conditions and accordingly has an effect on the mechanical stress of a cage [9]. In general, the simulation of fish farms is a complex challenge, for which most commercially available software does not provide sufficient capabilities to map the present situation. Especially in the field of flow simulation this leads to the in-house development of specially tailored programs or finite element codes [7]. These studies usually have a strong theoretical approach. In this respect, the focus of this study is the pragmatic use of simple methods and tools in order to finally make statements about the lifetime of the components using a design-Wöhler line.

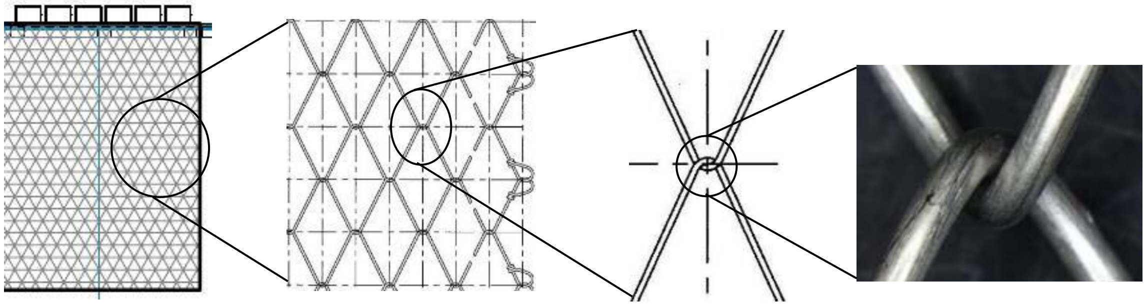
3. Tensile test data acquisition

3.1. Quasi-static tensile test – straight wire

In order to generate a reasonable virtual image of reality, it is necessary to record experimental data. The basis for the mesh is the corrosion-resistant duplex stainless steel 1.4462, which was declared in a previous study to be particularly suitable for a fish farming application in comparison with other steels [9]. Depending on the respective wire supplier, the wire material quality differs. For this reason, wire tensile tests are carried out in order to compare them with the manufacturer's specifications and to carry out simulations based on the own data. For this purpose, wire samples are taken from a wire coil and tested on a tensile testing machine in accordance with the standard. Important parameters are the yield strength, the tensile strength and the young’s modulus (Table 1). The breaking load resulting from the tensile test corresponds to that one calculated by a simulation model.

3.2. Quasi-static tensile test – One-Link

The properties of the wire change when it is formed into a triangular component of a mesh segment by cold forming [10]. For this reason, quasi-static tensile tests are also carried out for a very small section of such a mesh segment, the so-called One-Link (Figure 1), in order to create a data basis for comparing and refining the numerical model.



(a) (b) (c) (d)

Figure 1. The One-Link (c, d) is a small part of a net segment (a) of a whole cage, which consists of thousands of One-Links (b).

For the tensile test of such a One-Link, a special clamping device developed by Geobrugg AG is required with which it is possible to simulate the conditions under pure tensile load. This device enables a uniform testing (no bending moments). An overview of the average results of the tensile test with straight wire and the One-Link is shown in Table 1.

Table 1. Geometric, material and physical properties of straight and One-Link wires

Component	Parameter	Value
Straight wire	Material	1.4462; X2CrNiMoN 22-5-3
	Effective density	7,8g/cm³
	Diameter	2mm
	Yield strength	1467MPa
	Tensile strength	1712MPa
	Youngs modulus	202,5GPa
One-Link wire	Material	1.4462; X2CrNiMoN 22-5-3
	Effective density	7,8g/cm³
	Diameter	2mm
	Braking force	4332N
	Youngs modulus	202,5GPa

4. Simulation Setup

In this study, a numerical model is built in a non-linear structural-mechanical system based on CAD data. By applying different approaches, the behavior during the practical tensile test is simulated. The simulation was performed using the finite element software ANSYS. A short description of the numerical model is given in this chapter.

4.1. Design and boundary conditions

The CAD data are prepared with the CAD program PTC Creo 4.0 and then embedded in the simulation environment. By selective division of the geometry surface, areas with contact conditions are defined. These are the contacts between clamping device and wire as well as wire to wire (Figure 2).

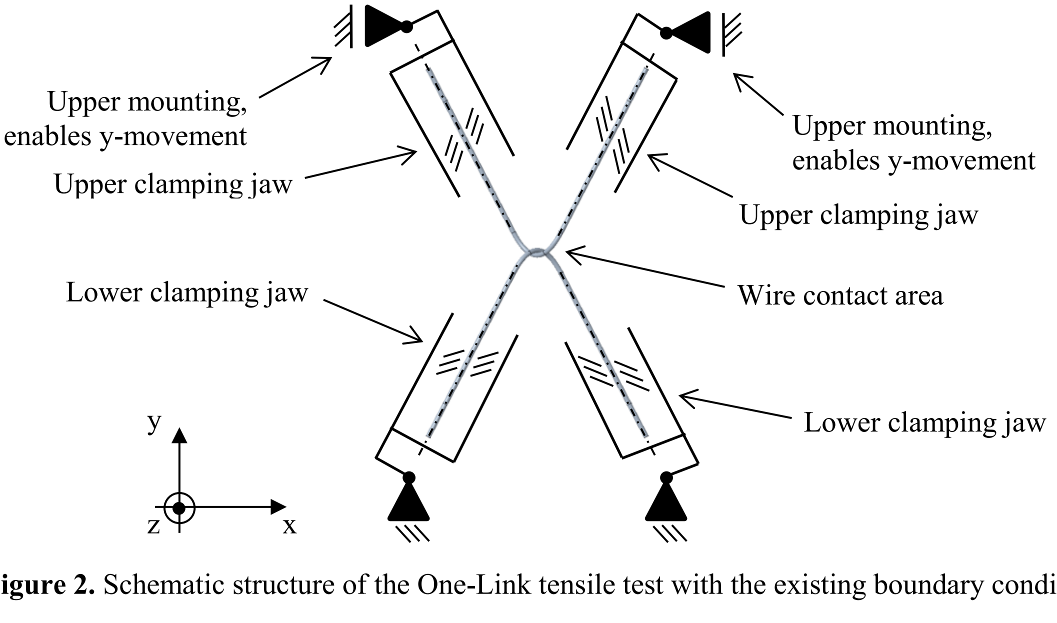


Figure 2. Schematic structure of the One-Link tensile test with the existing boundary conditions.

The numerical model of the One-Link-Test was built and completely defined within a static mechanical analysis system. The material data as well as the geometrical and physical properties are listed in Table 1. The lower clamping jaws (structural steel) of the clamping device are fixed in x, y and z direction as well as rotationally around the x and y axis, the rotation around the z axis is free, so that a pivoting movement is possible. The same applies to the upper clamping jaws with the exception that the tensile movement of the tensile testing machine is carried out in the y-direction, which during the tensile test led to breakage after an average $y_{total} = 2.85$ mm. The y-movement is applied in load steps. Due to the plastic deformations observed in the tensile tests at the bending radii of the two wire meshes of the One-Link, all calculations are non-linear. Iterative solution progresses take different amounts of time depending on the number of nodes and elements. The elements for three-dimensional FE-models proposed by the FKM guideline Nonlinear are used. This is the SOLID186 - a 20-node hexahedron with square displacement approach and reduced integration order. The reason for using this type is the high quality of calculation results [11]. The bilinear isotropic strain hardening model is used to represent the stress-strain behavior of the material. The wire contact area is subject to friction and CONTA174 as well as TARGE170 elements were used. By comparing all available contact algorithms, the Penalty Method produced comparable results in a very short time. The asymmetrical wire contact is detected at the Gauss integration point and its stiffness is updated in each calculation iteration. Parameter studies showed only slight influences on the results by changing the coefficient of friction, so it was set to $\mu = 0.2$. The wire ends, which are clamped in the upper and lower jaws, are defined by a bonded contact condition, so that they cannot move.

4.2. Simulation procedure

In the course of many calculation iterations, a procedure for the evaluation of the failure range has been developed through the trial and error principle, which is based on the sub-model technique. A sub-model is a small section of the initial overall structure at its interfaces the boundary conditions prevailing there are transferred (here the overall deformation) [12]. First the entire structure (clamping device and One-Link) is calculated with a rough FE-mesh (Figure 3). Then, over a total of four stages, increasingly smaller sub-models of the original model are extracted and calculated, whereby a convergence of the stresses prevailing in the wire contact area is achieved. The number of elements and nodes of the FE-mesh is continuously increased in order to finely resolve the stress curves and to detect the failure areas. Multizone, inflation and a face mesh are used to ensure a uniform mesh in the sub-models.

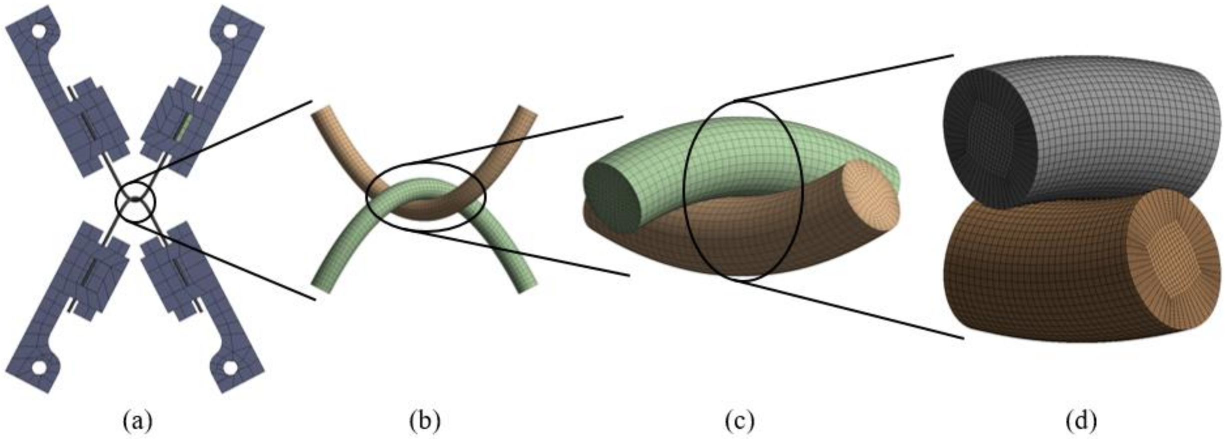


Figure 3. Initial overall model (a) and three sub-models derived from it (b, c, d).

In order to be able to perform both meaningful and fast calculations, some settings such as the contact stiffness, the contact detection method, the solution algorithm, etc. were changed from model to model on the basis of this four-step approach, so that the relationship between calculation time and result quality was satisfactory.

5. Setup of dynamic tensile test

The testing machine used for the dynamic tensile tests resembles a rocker with different lever lengths. The maximum test frequency of f = 1 Hz is generated by a pneumatic system. The force amplitude and the force range in which the test is performed can be set using adjusting valves. Adapters are used to mount the clamping device of the One-Link to the testing machine (Figure 4). The output of a load cell is computer monitored. During the tests, only the currently acting force is recorded in a graph as a sinusoidal curve. The data can be exported as an Excel sheet, evaluated and fed into the numerical model.

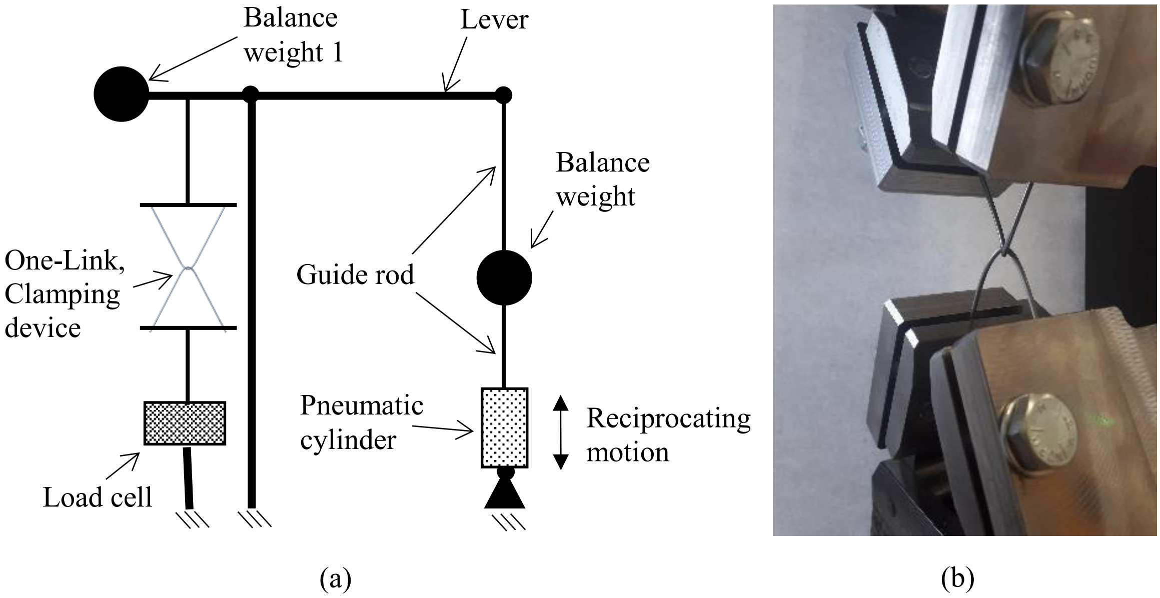


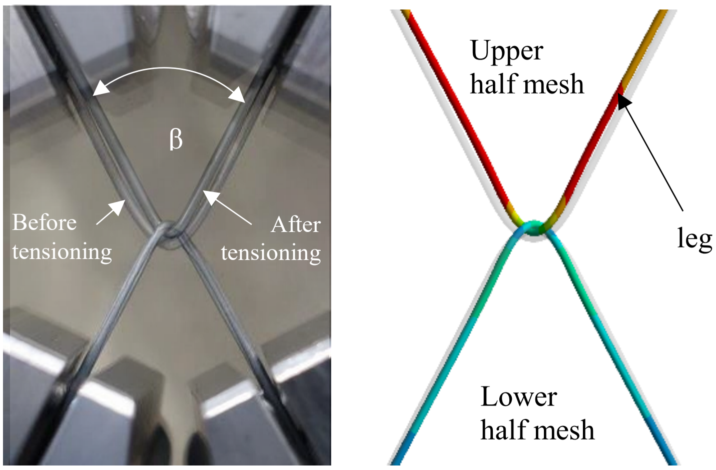
Figure 4. The schematic layout of the dynamic tensile testing machine (a) and the One-Link clamped in the clamping device (b).

One difficulty in the entire test setup is to make the conditions exactly the same for each test run. There are many mechanical adjustment possibilities. Starting with the use of the correct balance weights, through the alignment of the rocker lever, to the clamping of the One-Link in the clamping device.

6. Results

6.1. Comparison of tensile test deformation and numerical model

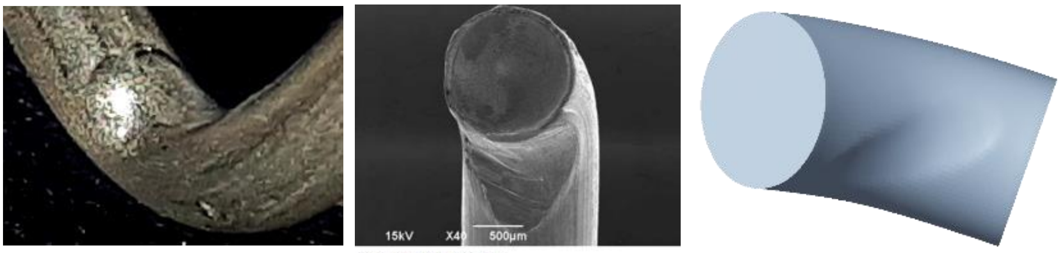
If the results of the One-Link tensile tests are compared with those of the numerical model, several criteria can be used to show that the comparison of experiment and simulation has been successful. When looking at Figure 5 (a) it becomes apparent that the bending radii at the beginning of the tensile test are slightly larger than just before the break of the upper wire. Due to the clamping device, the leg angle $\beta = 55°$ does not change. Figure 5 (b) shows the numerical model of the One-Link without clamping device. The initial state is shown in light grey and the state after the entire travel distance $y_{total} = 2.85$ mm is shown in color. As in the tensile test, the bending radii taper and the leg angles β remain constant.



(a) (b)

Figure 5. Before/after: One-Link tensile test (a) and the initial overall model (b), which represents the basis for the sub-models

After one of the two One-Link half meshes breaks at a tensile force of $F_{max} = 4332$ N, plastic deformation (notches) remains in the wire contact area of both, the intact as well as the broken half mesh (Figure 6 (a, b)). Shortly before breaking, the material begins to flow strongly. Therefore, the notch at the broken half mesh is more pronounced. The intact half mesh wire diameter is only reduced at this point. A measurement results in a difference of $\Delta = 0.12$ mm, which is also shown by the numerical model (Figure 6 (c)). These notches define the oval-shaped area where the two half meshes are in contact. Just below the wire surface of the notch transition, stress peaks (Hertzian pressure) are formed which finally cause the wire to break when the maximum tensile force is exceeded. The breakage is therefore always to be expected off-center of the bending radius. A look at Figure 7 shows the failure critical stress peaks. The maximum equivalent stress of Mises is $\sigma_{Mises} = 1784 N/mm^2$.



(a) (b) (c)

Figure 6. The notch on an unbroken (a), on a broken half mesh (b) and the notch on the numerical model (c)

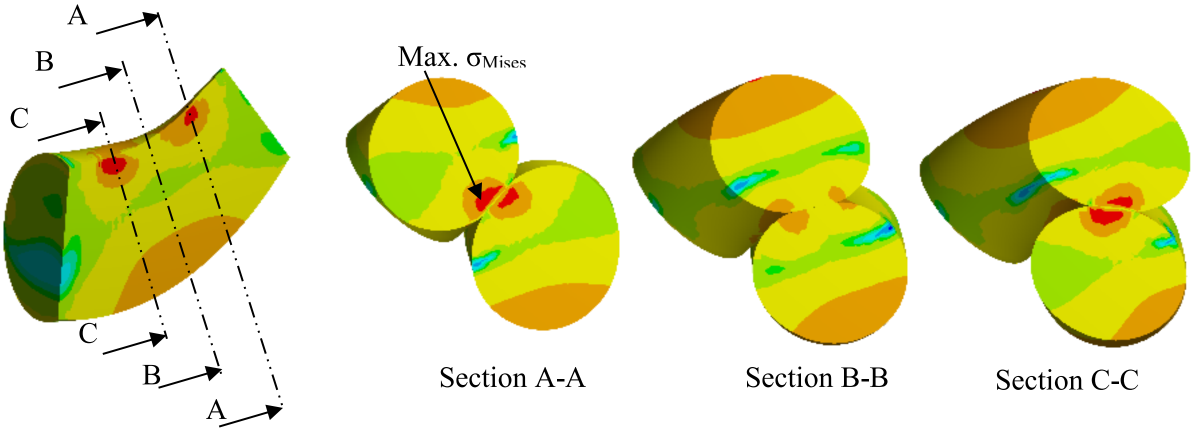


Figure 7. The numerical model shows the stress peaks just below the wire surface that cause the wire to fail left or right of the bending radius center.

This comparison shows that the numerical model can simulate the tensile test with the One-Link. It can therefore be used for further investigations in the context of the simulation of dynamic effects and the relationship between elastic-linear and plastic-non-linear calculations.

6.2. Dynamic investigations

The practical setup of the dynamic tensile test is available. Due to the Covid-19 pandemic, it has not been possible to record all data required for the numerical model so far. The tests will be continued at the Institute for Materials Systems Technology - Thurgau. The theoretical fundamentals will be developed based on the FKM guideline linear as well as the non-linear version and compared with the numerical results. One of the future aims of the studies are life cycle predictions for an entire fish farm with a design-Wöhler line, which connects non-linear plastic and linear-elastic calculation results as well as test results. To achieve this, data must be generated via dynamic tensile tests to feed the simulation-model with information from the real-live behavior.

7. Conclusion

Fish farming nets have to withstand many environmental stresses in use. This can lead to damage and the reason for this is usually hidden. In order to gain insights into the system behavior, a numerical model is under development, which will be refined and compared with data from experimental tests. The small wire contact area is resolved more and more finely within a four-step sub-model process. When comparing the numerical model with the results from tensile tests (straight wire and One-Link), similarities can be seen. With simple methods the deformation of the One-Link can be simulated and the areas of failure made visible. As in the tensile test, plastic deformation occurs in the wire contact area. Below the surface of the notch transition, high stresses are generated which cause the wire to break eccentrically. Thus, the numerical model is considered validated. It can be refined and compared with the data and observations of the subsequent dynamic tensile tests.

8. Acknowledgement

The project was funded by the Geobrugg AG, Aachstrasse 11, CH-8590 Romanshorn, Switzerland. We thank the Geobrugg AG for the financial and organizational support of this research project. We would also like to thank the Institute for Materials Systems Technology - Thurgau, the Laboratory for Materials Systems Technology at the University of Applied Sciences and all employees who contributed their expertise to the project.

References

[1] Urmersbach B, Statistisches Bundesamt, 2020 Weltbevölkerung, https://de.statista.com/themen/75/weltbevoelkerung/, Retrieved May 27, 2020

[2] Mavraganis T., Constantina C., Kolygas M., Vidalis K., Nathanailides C., 2020 Environmental issues of Aquaculture development, *Egyptian Journal of Aquatic Biology & Fisheries*, Zoology Department, Faculty of Science, Ain Shams University, Cairo, Egypt., Vol. **24**(2): 441 – 450 ISSN 1110 – 6131

[3] FAO. 2018. The State of World Fisheries and Aquaculture 2018 - Meeting the sustainable development goals. Rome Licence: CC BY-NC-SA 3.0 IGO

[4] Kitazawa D, Mizukami Y, Shimizu H 2012 Development of the Fish Cage Installed in Variable Depths“, *Proceedings of the ASME, 31st International Conference on Ocean, Offshore and Arctic Engineering*, July 1-6, 2012, Rio de Janeiro, Brazil, DOI: 10.1115/OMAE2012-84145

[5] Seemann, P 2013 Catch of the day; *International Innovation, EuroFocus*, 01/2013; Seemann, P., Dornbierer, U., Sorg, M., Gümpel, P.: Development of a New Material System from High Tensile Stainless Steel Wire with Environmentally Compatible Antifouling Properties for Off-Shore Fishfarming Cage; APA 2013 (Asian-Pacific Aquaculture), Ho Chi Minh City (VN), December 2013

[6] Zhao Y-P, Chen Q-P, Bi C-W, Cui Y 2019 Experimental Investigation on Hydrodynamic Coefficients of a Column-Stabilized Fish Cage in Waves, *Journal of Marine Science and Engineering*, **7**, 418; doi:10.3390/jmse7110418

[7] Zhao Y-P, Xin-xin W, Decew J, Tsukrov I, Xiao-dong B, Chun-wie B 2015 Comparative Study of Two Approaches to Model the Offshore Fish Cages, *Chinese Ocean Engineering Society and Springer-Verlag Berlin Heidelberg, China Ocean Eng.,* Vol. **29**, No. 3, pp. 459 – 472, DOI 10.1007/s13344-015-0032-0, ISSN 0890-5487

[8] Strand I M, Faltinsen O M, 2020 Linear wave-induced dynamic structural stress analysis of a 2D semi-flexible closed fish cage, *Journal of Fluids and Structures* **94**, Published by Elsevier Ltd., https://doi.org/10.1016/j.jfluidstructs.2020.102909

[9] Gümpel P, Hörtnagl A, Sorg M 2019 High Tensile Stainless Steel as a Sustainable Material for Aquaculture“,*Procedia Manufacturing* **30** (2019) 315–322, Published by Elsevier Ltd., 14th Global Congress on Manufacturing and Management (GCMM-2018)

[10] Chemische Werke Kluthe GmbH, Heidelberg, Kluthe Magazin, https://kluthe.com/magazin/kaltumformung-von-metall/; Retrieved May 27, 2020

[11] Fiedler M, Wächter M, Varfolomeev I, Vormwald M, Esderts A, 2019 Richtlinie Nichtlinear: Rechnerischer Festigkeitsnachweis unter expliziter Erfassung nichtlinearen Werkstoff-verformungsverhaltens: für Bauteile aus Stahl, Stahlguss und Aluminiumknetlegierungen, *Frankfurt am Main: VDMA* Verlag GmbH, , pp. 13-16

[12] Gebharrdt C 2014 Praxisbuch FEM mit ANSYS Workbench, Einführung in die lineare und nichtlineare Mechanik, *Carl Hanser Verlag München*, Auflage **2**, pp. 213-214