VOL. 12, NO. 3, FEBRUARY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

STRUCTURE DESIGN AND CHARACTERISTIC ANALYSIS OF BUCKLING STRENGTH ON SWEDGE FRAME PRESSURE HULL WITH FINITE ELEMENT ANALYSIS

Ahmad Fauzan Zakki[1], Dong Myung Bae[2], Sulistiyono Susilo[1], Eli Akim Sipayung[1] and Suharto[3]
[1]Department of Naval Architecture, Diponegoro University, Semarang, Central Java, Republic of Indonesia
[2]Department of Naval Architecture and Marine Systems Engineering, Pukyong National University, Nam-gu Daeyon Busan, Republic of Korea
[3]Janata Marina Indah Shipyard and Vocational Program of Naval Architecture, Diponegoro University, Semarang, Central Java, Republic of Indonesia
E-Mail: ahmadfzakki@undip.ac.id

ABSTRACT

This study discusses design and analysis of the strength of submarine's inner hull, which uses swedge frame, in Pasopati Submarine owned by the Indonesian Ministry of Defence. The study analyzes pressure hull in various depths (100 meters, 300 meters, and 500 meters), with the use of 35 mm plate thickness, and the T profile size. While modeling is done with the FEM software, buckling analysis will make use of a software. The making of the model design, followed by an analysis of the model, will generate an output or data, i.e. calculation in the form of model images, analysis finding, and parameters of the necessary data, such as the rate of stress voltage, location of the critical point due to pressure, and security level of the construction.

Keywords: buckling strength, swedge frame pressure hull, finite element analysis, structure design, characteristics.

INTRODUCTION

Along with the development of offshore industry, several types of submarine can be used for the benefit of supply and maintenance on offshore buildings, in either fixed or submersible platform. In general, construction of inner hull in a submarine has two types, namely ring frame pressure hull and swedge frame pressure hull. Swedge Frame Pressure Hull is intended to (1) maximize the existing space on the submarine so that materials and economic value for the shipbuilding are cheaper, and to (2) strengthen the submarine itself so that it becomes better.

The growing trend in submarine design shows the need for the increase of depth level for both industrial and military sectors. Moreover, the complexity of modern submarine and the demand for efficiency, safety, and greater reliability become a challenge for designers in making the submarine design, especially in the pressure hull components. Determination of the design must consider its specification, which includes type, main size, speed, and others.

In general, the structure of a submarine consists of two hulls, namely outer hull and inner hull. The outer hull serves to control the influence of hydrodynamic loads. This part is called hydrodynamic hull. In the other hand, the inner hull serves to withstand hydrostatic pressure when the submarine is in diving condition. Based on the functions used to withstand the pressure, this part is commonly referred to as pressure hull (American Bureau of Shipping, 1993). The design of pressure hull is a combination of cylinder, cone, and dome shape built to withstand the pressure as a result of diving at a high depth level. The increased pressure for additional depths of 100 feet is followed with the increased hydrostatic pressure of 44.5 psi for the standard sea water and 43.5 psi for the fresh water (Carlberg, 2011).

The depth level to which the pressure hull structure has failed is so-called collapse depth. One topic that attracts a naval architect is the power of the submarine. Consideration of the submarine's strength deals with the fact that a submarine built too strong will be very heavy, slow, and require a greater cost for it definitely uses construction with large dimensions and thickness. Moreover, the burden of heavy construction would hamper the submarine to reach the top speed during operation as planned when designed. The submarine will be slow and take time to operate. Conversely, when the submarine is too weak, it will be at high risk of structural failure. This is due to the structure. In this case, the construction uses lightweight construction and has a thin dimension. This is done for cost savings in the submarine construction, so that most likely it will not be able to withstand the varied loads during operation, from either inside or outside the submarine.

In this study, the design of pressure hull in a submarine with missile launchers will be developed. Based on the numerical results of non-linear static analysis on the hydrostatic pressure with operational depth condition, this study seeks to provide maximum operational depth level taken from elastic to plactic behavior variations. Calculation of the swedge frame strength will be done in this study. This research is expected to give significant contribution to the development of submarine technology.

This study aims to formulate the manufacture of structural design of swedge frame pressure hull of a submarine, and to know the strength value in it at a depth of 100, 300, and 500 meters. This study is restricted with a linear static analysis calculation. The plates used in the

design is an Alloy Steel HY 80 type, σ minimum yield strength of 80 ksi (552 MPa), thickness of 35 mm, and loading conditions in maximum stress and maximum displacement. Furthermore, the entire test conducted refers to ABS Rules for Buildind and Classing Underwater Vehicles, Systems, Hiperbaric Facilities 2002 Concept Design of a Commercial Submarine.

LITERATURE REVIEW

Pressure hull

Weight of a submarine depends on the maximum depth. Greater diving depth level requires a larger pressure hull. The depth level used as a design consideration includes operational or normal depth level; maximum permitted depth level, and collapse depth level.

The maximum permitted depth level is the maximum depth level where a submarine is still safe to operate. This depth is achieved only on certain conditions. Collapse depth is the depth where the pressure hull structure has failed.

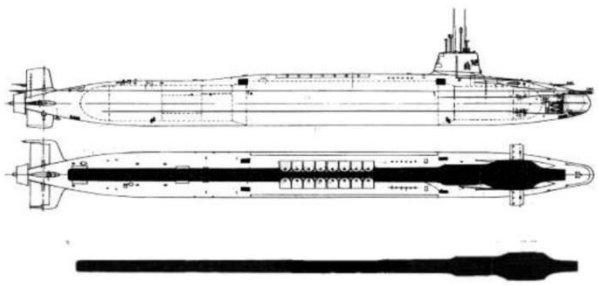

Figure-1. Submarine's Pressure Hull.

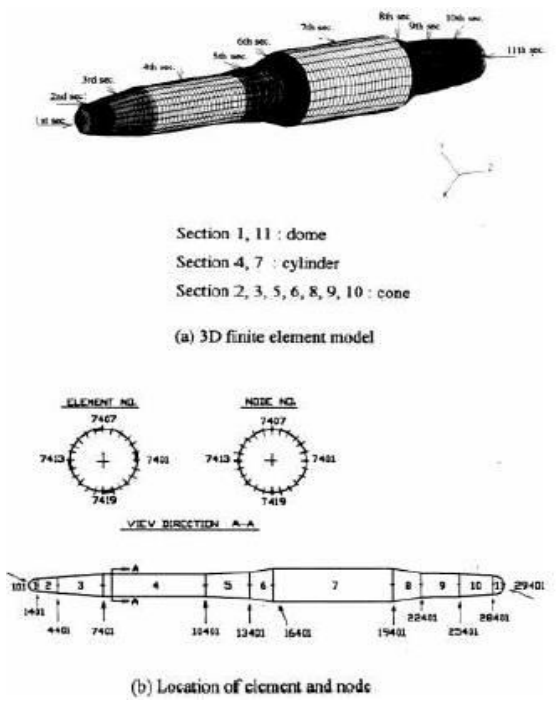

Figure-2. Submarine's pressure hull structure using swedge frame.

The submarine's pressure hull is built not only to withstand hydrostatic pressure but also to withstand a strong blast shock. In general, collapse depth is a multiplier between the safety factor and the operating depth. Carlberg (2011) showed that the value of safety factor for a submarine ranged between 1.5-2.0. This figure may be acceptable in engineering. In a military submarine design, consideration of hydrostatic load at the operating depth level and the load caused by a blast shock should be given. In condition of war, the effects of the blast shock can cause the structure to undergo large deformations, which can lead to material failure. The large size and the distance of the explosion provide shock effect of the underwater explosion. High intensity of waves can also cause damage to the submarine's pressure hull, and gradually some of its equipments does not function properly (Friedman, 1984).

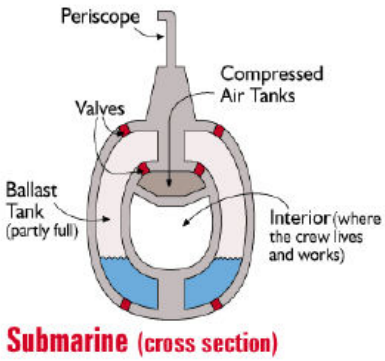

Figure-3. Front side of the submarine's pressure hull.

Studies on pressure hull were conducted by several researchers, particularly in regard to the analysis and design of submarine's pressure hull. Gorman and Louie (1991) developed an optimization method to examine the material, form, and architecture of the pressure hull by considering the hull yielding, buckling, general instability, and instability of failure mode. Jackson (1992) presented a concept design of submarine, which later becomes a basis for submarine planning process. By using Finite Element, Sibarani (2013) analised the structural design and buckling strength characteristics on ring frame submarine pressure hull.

Ross (1987a; 1987 b; Ross, 1992) and Ross and Palmer (1993) reviewed the conventional pressure hull and novel design. Based on the finite element method and experiment, Ross (1987a), Ross and Palmer (1993) proposed an efficiency improvement of dome structure by changing the dome of a submarine's pressure hull. Ross introduced a pressure hull axisymmetric design with swedge frame to withstand hydrostatic pressure. Comparison between swedge frame with ring frame has also been carried out, where, in term of structure, swedge frame is considered more efficient than conventional ring frame (Ross, 1987a; Ross, 1992). Ross (1987b) presented his experiment result showing a plastic failure of the thin walled ring frame on the conical shell, with pressure load of external uniform. Yuan, Liang and Ma (1991) presented

a theoretical analysis of elastic instability of the cylinder swedge frame against hydrostatic pressure load by considering the influence of various angles of conical section. Yuan et al. (1991) also did an elastoplastic analysis and non-linear response of Ross' swedge model (Ross and Palmer, 1993).

Buckling analysis

Buckling is an instability that leads to the failure mode (Assakkaf, 2003). Buckling stress may refer to a process in which a structure is not able to maintain its original form. This happens because the structure receives a stress excess resulting on deflection that reshapes the structure. In a stable structure, buckling can return its original shape. Meanwhile, if the structure is not able to return to its original shape, the structure is said to be failing. Factors influencing on the buckling are elasticity of materials, size dimensions, loading, and measurement factor.

Practical aspects of finite element simulation

Manual calculation of buckling occurring in the pressure hull structure of a submarine will be discussed using the rule of Hyper Works (2012) about the practical aspects of finite element simulation. There are several theories and formulas used to calculate the amount of pressure boundary in accordance with the components of swedge frame pressure hull used in the design of the submarine's pressure hull. Some of them are:

The overall buckling strength is formulated with:

$$Pn = \left(\frac{Et}{R}\right)A_1 + \frac{EIA_2}{LR^3}$$

Where,

Pn = overall pressure of cylinder
Et = modulus elasticity
R = mean of radius
I = moment of inertia of the section composite consisting of a stiffener
L = distance between stiffeners

The result of pressure with cylindrical stiffener (swedge frame), including a circular pressure and bending stress arising from outside the circle, is calculated in compliance with the following equation:

$$\boldsymbol{\sigma y} = \frac{Pt\sigma y}{Pyt} + \frac{3Ec\delta Pt}{(Pn-Pt)R^2}$$

Where, σy = yield point minimum
Pt = cylinder stiffner longitudinal yield stress pressure
Pyf = pressure in the middle of cylinder
E = modulus elasticity
R = mean of radius

Finite Element Method

The method used in the final work to be made is a modeling method and finite element analysis. Finite element method having recently developed has convincingly become a powerful device for analysis of various types of plate's problems and construction structure, because the result is more favorable than theoretical settlement. The element method will eventually replace the experimental stress analysis techniques in determining the strength of the element. The coefficient of element stiffness that can be used directly has been generally available and provided precise result. Once the coefficient is determined, structural system analysis will show the same result as the matrix methods used in engineering mechanics of which the computer program has been available (Prabowo *et al*, 2016; Zakki *et al*, 2016).

The accuracy of finite element method is influenced by the following parameters (arranged in accordance with its virtues), i.e. movement patterns defined for the element, the number of elements, loads presentation techniques, edge conditions of specific problems, and computer programs. Although the finite element method is still in the developmental phase, this method will be widely used later on in many areas of structural and continuum mechanics. By assuming the real cases.Bae et al. (2016) stated that finite element method can produce virtual experimental data. Because the method is developed by the engineers based on engineering logic of physics, its development in the future will be driven by a more exact assessment by the mathematicians.

Nowadays, most of assessments of the finite element method is aimed at developing the shape function or enhanced elements, which can provide a rapid convergence and better accuracy. In addition, the size and speed of next-generation computers will require a new computer program. Application of the basic concepts of the finite element method today has been expanded to the problem, such as plate thickness and membrane structure (shell), non-linear problem (including plasticity), stress due to temperature, analysis of aeroalastis and hidroelastis system structure, flow of liquid, post buckling behavior of the plate, membrane structure (shell), and others.

Finite element method can be viewed as an extension of the displacement method (for the frame structure) to continuous problem with two or three dimensions, such as plates, membrane structure (shell), and rigid bodies. In this method, the continuum is actually replaced by an equivalent ideal structure consisting of unique elements (discrete element). This element is called finite elements and linked together in a number of nodes.

With regard to the form of displacement or stress patterns in the element and to the use of energy theories, we can reduce the matrix of stiffness that connects nodal forces with nodal displacement on the element. Stiffness matrix of the elements is then formed in the same way as in the portal case. If balance requirement is applied at every node on the ideal structure, a set of simultaneous algebraic equations can be formed. The solution produces

the whole displacement of the nodes, which will be used to determine all inner stresses. The finite element method is first applied by R.H Clough (1960) to the problem of stress by using triangle and rectangle elements (see. Clough, 1960). This method is later expanded, and now we can use the elements of a triangle and a rectangle on a particular plate, elements of four and six areas in three-dimensional stress analysis, and curved elements on the problem of membrane structure with single or double curves. The scope of its application has also been extended to the issue of stability and vibration. By applying incremental approach that treats each additional burden in linear elastic, the problems of material or geometric nonlinearity have been solved well with this method.

One development of the finite element method is the finite strip method. In this method, the actual structure is idealized into lanes which are connected in nodal lines, while both ends of all lanes are interconnected to form two opposite edges domain. Displacement and force along the nodal lines are considered to vary in accordance with the basic functions specified. Later, the rate of the displacement and force at a point is obtained by multiplying the parameters of nodes and the value of basic functions at that point. Thus, the problem of two-dimensional plate is reduced to one-dimensional problem, so that we get quite a lot of savings in the calculation when compared with finite element approach. Finite nodal method is particularly useful for the analysis of the bridge plate that is straight or circular, the bridge with box girder, and the folding plate (Ghali and Neville, 1978).

Dealing with submarine type, each submarine contains different objects. The sample of loading system can be seen when the submarine is in condition of full of fuels and foods and is combined with various conditions of the surrounding environment or the waves causing the submarine in sagging and hogging conditions.

Finite element application in submarine

Finite Element Method (FEM) is a method used to analyze a construction. This method is now widely used in the construction of submarine, coastal structures, and offshore. Basically, the coverage of this method is very wide, not limited to steel construction but also to the fluid. Structural analysis using finite element method makes it possible to obtain stress deployment on the construction being analyzed. Failure of a construction can be known by using this analysis and at the point in which the failure is indicated.

Therefore, it will be easier for designers to make modification of the construction and to strengthen the construction that is identified to damage or fail. To analyse using the finite element method, there are many softwares created to facilitate the analysis, some of which are MSC Nastran, Ansys, Algor, Solidwork, etc. Broadly speaking, these softwares have a working system and same steps in conducting the analysis, i.e. beginning with the modeling and followed by meshing, determination of the boundary conditions, loading, and analysis.

Program of MSC patran

MSC Patran is a helping program for pre- and post- processing in the finite element method analysis, in which one software analysis process used is Msc Nastran as described in the next section. Analysis process of the finite element method or analysis begins (pre-processing) from this helping program, i.e. Msc Patran. In general, the steps employed in using the software of Msc Patran are described as follows: creating a database and unit, creating a geometry model, checking and correcting the geometry errors, determining the loads and boundary condition, `determining the material and element properties, meshing process, optimization process and equivalence, and analysis process.

MSC Nastran (MacNeal Schewendler Corporation NASA Structural Analysis) is a finite element program created and developed by NASA (National Aeronautics and Space Administration) for solving the structure and component analysis. The first is static analysis which consists of static analysis with changes in surface and stiffness, and buckling analysis. The second is dynamic analysis which consists of normal model analysis, response analysis of frequency (harmonics analysis), transient response analysis, and linear and non-linear transient response analysis. The third is transient or steady state heat transfer analysis. In MSC Nastran, the manufacture of object geometry can be made directly in the program. Besides, we can also do translating or geometry importing of objects from CAD (Computer Aided Design) program in DXF and IGES format, or from any othe finite element analysis programs, such as ALGOR, ANSYS, COSMOS, I-DEAS, Mtab*STRESS, MSCpal and CDA/ Sprint, NASTRAN, PATRAN, STAAD ISDS, and STARDINE.

METHODOLOGY

Determination of the main size of submarine

In general, there are several requirements in planning a submarine, namely radius and cruise route, official speed (Vs), deadweight of submarine (DWT), type of goods, quantity of goods, and passenger capacity, in addition to other requirements relating to safety, comfort, and beauty. These mission requirements become critical information that will be relied upon by planners of the vessel to determine its main dimensions that are compatible in technical and economical aspect.

Submarine design in this study uses a comparison method. The difference between the comparison submarine and the proposed submarine is located in height (h) only, while other measures in both of them are the same. Here are some methods included in the comparison method, such as linear regression method, cube root format, and the geosim procedure.

According to the above explanation, the comparison method has its advantages, that is, the arrangement of primary measure calculation is relatively short. This is due to the submarine's main dimensions generated by multiplying the main dimensions of comparison submarine with the scale factor. However, in

VOL. 12, NO. 3, FEBRUARY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

the process of determining the submarine's main dimensions, we must consider several things, including:

a) Length of submarine (L)

The length of the submarine has an influence on the speed and strength of elongated submarine. Extra length of the submarine on displacement condition and same volume will reduce the resistance of the submarine. However, it can also lead to an increase in longitudinal bending stress and a decrease in stability and maneuverability of the submarine. Addition of the submarine's length will reduce engine power at constant speed, reduce weight of the main engines, reduce fuel consumption, and enable more stable route of the submarine.

b) Breadth of submarine (B)

Addition and subtraction on the breadth of the submarine have an influence on the height metacentra. Extra breadth of the submarine on displacement condition and same speed will lead to a high increase in metacentra (MG), so that the stability of the submarine becomes good. Extra breadth of the submarine is also used to increase the breadth the submarine's rooms.

Data of the submarine's pressure hull to be designed in this study is the Pasopati Submarine with 76.6 meters in length, plate type of Alloy Steel HY 80, 6.3 meters in breadth, and 30 mm of plate thickness.

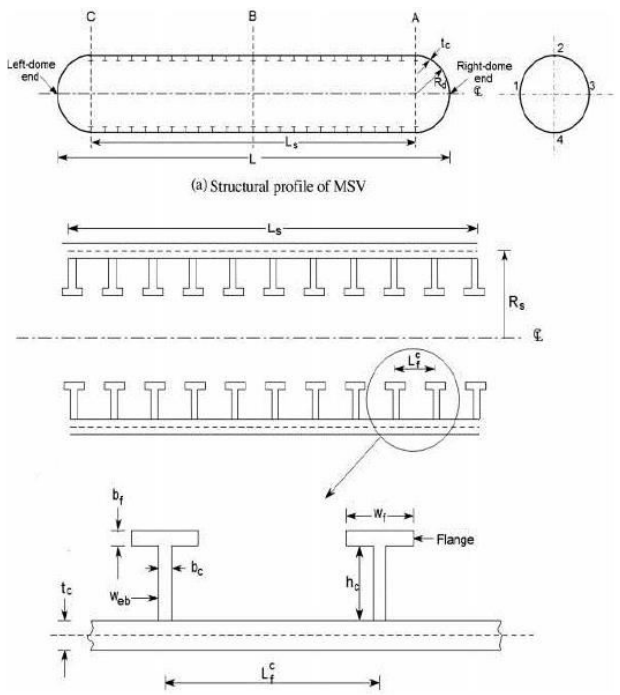

Figure-4. Pressure hull construction of pasopati submarines.

Assessment of structure strength

In CSR standards, an assessment of the strength of a structure using Finite Element Analysis is an obligation. Therefore, the proposed design of a submarine pressure hull is further analyzed in its structure strength using MSC Patran and MSC Nastran program. This analysis will be performed on the hull inner area to find out how much the Yield Strength happened.

The making of the FE model from the proposed design

The design of geometry and meshing operation is done using MSC Patran. This program provides a general need for modeling of geometric elements and FE models. FE modeling from system design of a submarine's inner hull construction is illustrated in accordance with the actual dimensions and includes all existing elements. Likewise, the material and the properties of the 1D and 2D elements can be defined and visualized by FEM Software program to be an 1D element in the form of beam and 2D element for a hull type of the same size with many previous studies.

The main structure of submarine construction consists of inner hull structure and beam. The modeling process begins by defining the geometry for each element to be created. The system or coordinate axes of X, Y, Z (co-ordinate system) in this modeling are:

X-axis: Longitudinal, positive direction points to FWD
Y-axis: Vertical, positive direction points Upwards
Z-axis: Transverse, positive direction points to Port

The submarine modelling is made according to the length and breadth of its diameter to analyze the strength of its structure based on the image that is designed and later imported into the MSC Patran.

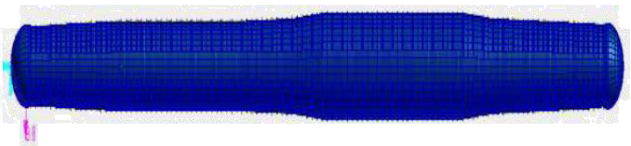

Figure-5. FE model of submarine using the swedge construction.

All major structural components either longitudinal or transverse must be modeled. Some other structures in this modeling include:

- A dome shape located in the bow and stern of the submarine
- A cylindrical shape located in both the rising and falling room in the submarine cylindrical
- Conical shape located in a room whose height is at par with the submarine
- The profile used is modeled in the form of beam
- Plates modeled in the form of shell
- Consisting of four transverse bulkheads

Geometry model is created into an interconnected system, so that it becomes a unity in which each geometry is defined in accordance with shape, size, and type of property in each geometry element, such as shell elements for plate shell geometry, ID beam element for frame geometry, and so on for all components of the model.

The type of elements used for the finite element model of the primary structure of submarine includes:

- Elements of breadth (shell) to model the hull, structure, and wrang
- Elements of the line (beam) to model the beam

In defining the material, steel with a maximum stress of 235 GPa = 2.35 x 1011N/ m2 is considered to have normal strength. The steel's grade that has normal strength includes grade A, B, D, and E steel. In the finite element modeling of submarine construction, material and Steel A 36 are used with the following properties:

Name of the material: Alloy Steel HY 80 Ksi (552 Mpa)
Modulus Young: 207 Gpa = 2.07 x 1011 N/m2
Density: 7.85 Mg/m³ = 7850 kg/m³
Poisson Ratio: 0.3

The steel used in this structure design is Alloy Steel HY 80. Steel from this type is commonly used for military fleets, such as tanks, warships, and submarines.

Determination of the boundary condition

One of the most serious tasks related to the proper modeling of submarine's structure with finite element is the definition of boundary conditions. Boundary Conditions as called in the finite element method is the final stage of a finite element modeling, i.e. the determination of the foundation before the model is analyzed. Therefore, it can be interpreted also as a clamping condition that functions to maintain the objects, so it does not move when analyzed. The move could be in the form of slide (translation) and rotation. While the boundary condition is applied to the fore end and the aft end on the finite element model as a fulcrum of the analysis, the nodal dots on the fore end and the aft end of each are connected rigidly against the independent point, which is also defined as center of gravity of the submarine. This is done, so that the combination of the loads and the results of the stress response that occurs can be carried out maximally for giving the boundary condition on this submarine model.

Rigid Link is made on the X axis translation and Y, Z axis rotation, both on aft end (all longitudinal element) and fore end (longitudinal element) location.

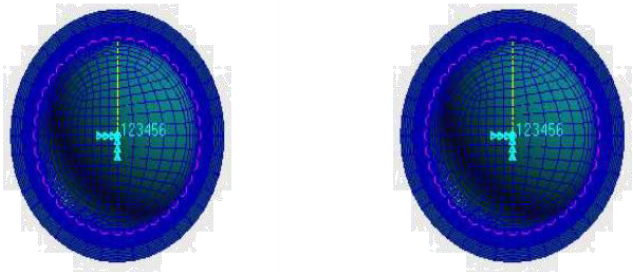

Figure-6. MPC FE model of submarine using swedge construction.

Figure-6 shows an MPC model where the Independent Point at the fore end and the aft end has been determined, i.e. for H 2.125 m, the fore end is at the node 171, and the aft end is at the node 31 072. In boundary condition of those nodes, the submarine will undergo a clamping treatment to avoid a shifting and rotation movement during the analysis. The code in the software is as follows:

Translational (x, y, z) = <0, 0, 0>
Rotational (x, y, z) = <0, 0, 0>
Rotational (x, y, z) = <0, 0, 0>

Spring elements are divided into Vertical Springs, i.e. Side, Inner hull, Longitudinal Bulkhead, and Horizontal Springs including Deck, Inner Bottom, and Bottom Shell. Calculation of the stiffness of the spring element uses the following formula (case III-13):

$$c = \left(\frac{E}{1+\nu}\right)\frac{A_{S\text{-net50}}}{l_{tk\,n}} = 0{,}77\,\frac{A_{S\text{-net50}}}{l_{tk\,n}}$$

From calculation the above formula, the stiffness of the spring element on the model is obtained, i.e. c = 82634144 N/m.

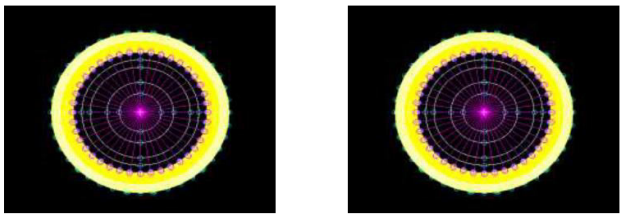

Figure-7. Spring element of the model.

Verification of the model

Verification is carried out after the modeling is complete. The purpose of this validation is to check the elements of the model that are made, such as:

Connectivity element is a checking of elements relationship or connectivity. All elements must connect each other. If they are not connected, the model cannot be analyzed.

Duplicate node is a measure that aims to determine the absence of nodes that are duplicated or

doubled. If the nodes are duplicated, the model cannot be analyzed.

Duplicate element is a checking intended to eliminate one element that overlaps and is not required. If some elements overlap, the model cannot be analyzed.

Consistent plate normal is a checking aimed at restoring the normal vector that is in wrong direction. The area must have a uniform direction, so that the direction of the applied load will not uniform. This condition will cause an error in analyzing the voltage structure.

Guilding element is an element whose normal vector is used as a reference to alter the normal vector that is wrong. Error in the normal vector is crucial to be corrected, especially when the load is in the form of pressure.

Determination of loading condition

This submarine construction, which has been modeled with the finite element method, is then made its modeling in computing program of FEM software as presented in the previous section. The following section will discuss Local Static Loads. This loading is done to see the structure strength of a model towards hydrostatic load and cargo load. Determination of the structure strength analysis criteria under stress is the main purpose, that is, to explain the elements or components of the working system and to maintain the dimensional accuracy and the strength of components that are capable of working within safe limit.

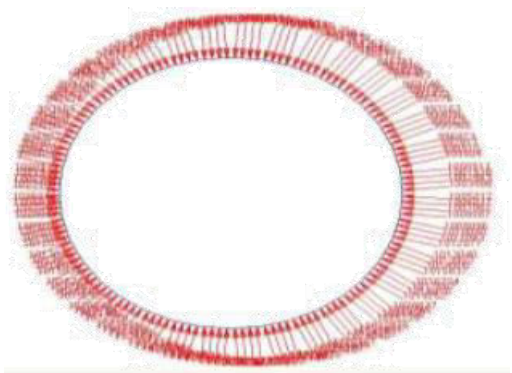

Figure-8. Hydrostatic load.

In determining the amount of pressure in seawater acceptable by the structure of the model, this study uses a physics approach, with a maximum load of:

Ph: P x g x h

Where
Ph = Hydrostatic Pressure (N/m2)
P = Density (kg/m3) (seawater) = 1025kg/m3
g = Acceleration of Gravity = 9.8m/s
h = Depth of the Surface (m) = 2.125m
Ph: 1025 kg/m3x 9.8 m/s x 2.125m: 21345N/m2 (Pascal)

Therefore, the maximum load accepted by the structure is 21345 N/ m^2 (Pascal). Later, the amount of pressure from inside (the cargo space of a submarine) is ignored because its value is small and not influential.

Procedure of direct strength analysis

Pressure hull structure modeling of submarine using swedge construction

The form of submarine's pressure hull that uses swedge construction is typically built using a combination of cylinder, cone, and dome shape, which is located on the left and right end, as shown in Figure-9.

This structure is what will become object of this study. Bulkhead that becomes a connecting door between the rooms inside the Pressure Hull is not modeled since it is not a weight-bearing structure too influential on the strength (merely a divider between the rooms), so that the doors are not experiencing a significant pressure. Pressure on the pressure hull in this study refers to the ABS Rules for submarines, concept design of a commercial submarine consisting of five main sections separated by bulkhead that is able to hold when leakage from the next-door room occurs. Conical section on submarine connects the main part of the pressure hull with a small airlock diameter. Transition of the conical section is useful for achieving a smaller diameter without requiring a large voltage. Stiffener with T circular profile is made of steel with a thickness that has been customized according to rules of steel density, while Young's modulus and Poisson's ratio are assumed to have a special value. The thickness of the plate should be the same where welded parts of the plates and all parts of the pressure hull plates have the same thickness, be in the cone, cylinder, and dome section.

Thickness of the plates that is planned should be the same. Strength of the plates with small diameter will result in great strength and can hold a larger load than that with greater or same diameter. It enables the use of thickness and sharpness displacement during the welding, where there are diverse demands for the plate thickness when high extra cost of the steel is found. The fineness of the weld is required to avoid high stress concentrations during the welding. The structure of properties swedge frame to be designed:

Length: 58 m
Maximum Diameter: 4.25 m
Minimum Diameter: 3.20 m
Frame Distance: 0.50 m
Plate thickness: 0.0035 m

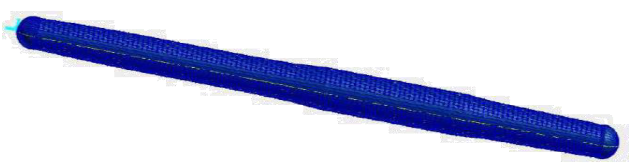

Figure-9. Swedge frame model on a pressure hull cylinder.

Boundary conditions

Boundary Condition in the analysis process of finite element method-based software is a phase to maintain the boundaries (parts to be clamped) of the model prior to loading and analysis. The clamped parts become the foundation and unable to move, shift, or rotate. In pressure hull modeling of a submarine, there are two types of boundary condition. The first condition is the pressure hull structure with only one clamping point, which is at one end of the model. The second condition is by clamping both ends of the pressure hull.

In this study, the author tends to use the second condition, because the first condition does not satisfy the expectation during analysis. This is because there is the end point that is left free to move, shift, or rotate, so the fixed end part moves to the pressure exerted.

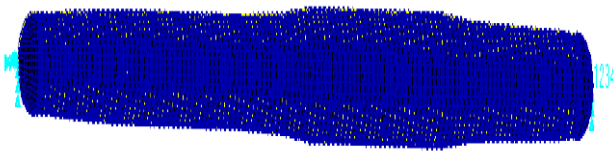

Figure-10. Boundary condition on the submarine's pressure hull using swedge frame.

For the case of boundary condition modeling on the submarine's pressure hull, MPC Method is not used to clamp the pedestal.

Loading conditions

The condition or depth desired by the researcher in this study in analyzing the characteristics of buckling in the pressure hull is based on depth variations (100 m, 300 m and 500 m), profile size, and plates used.

Verification of analysis result

The last stage of the analyzing process is verification of the data, that is, by considering the forms of buckling characteristics with variations of loading that has been given. The deflection result of the pressure that has been given does not become a reference to the writer, because there is no standardization of the rules applied to submarines (submarine is a type of special or military ships).

Variation of loadings

Loading *at Depth of 100 m*

The load placed on the structure of the submarine's pressure hull is the pressure of seawater in accordance with the conditions of its depth. The pressure given will suppress the plate element and beam element. The use of beam dimension on both the pressure hull's structures in this study is different.

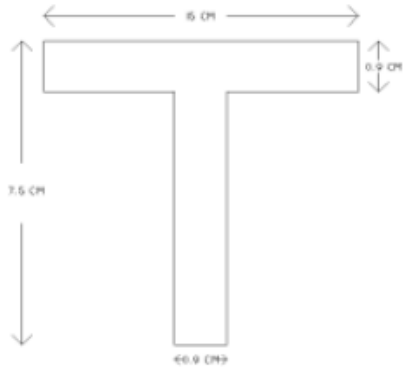

Figure-11. Beam dimension of pressure hull's frames swedge.

By using hydrostatic pressure calculation (h: depth surface = 102.125 m), the hydrostatic pressure on the structure of the submarine's pressure hull using swedge frame is:

Ph = 1025 kg/m^3 x 9.8m/s^2 x 102.125 m
= 1025845.62 N/m^2 (Pascal)
= 1.025 MPa

So, the load received by the structure of the submarine's pressure hull using swedge frame is 1.025 MPa.

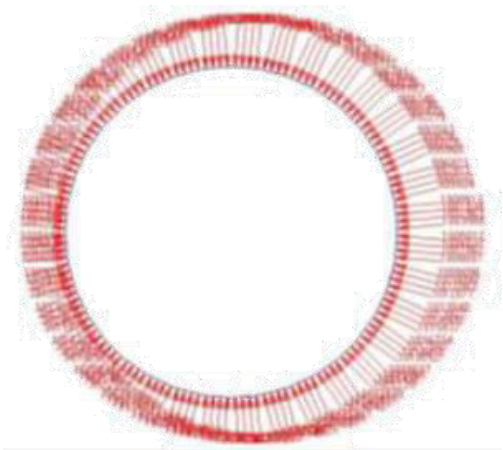

Figure-12. Seawater loading and amount of pressure for a submarine with swedge frame (100 m).

Loading at Depth of 300 m

By using hydrostatic pressure calculation (h: depth surface = 302.125 m), the hydrostatic pressure on the structure of the submarine's pressure hull using swedge frame is:

Ph = 1025 kg/m3 x 9.8m/s2 x 302.125 m
= 3034845.62 N/m2 (Pascal)
= 3.034MPa

So, the load received by the structure of the submarine's pressure hull using swedge frame is 3.034 MPa.

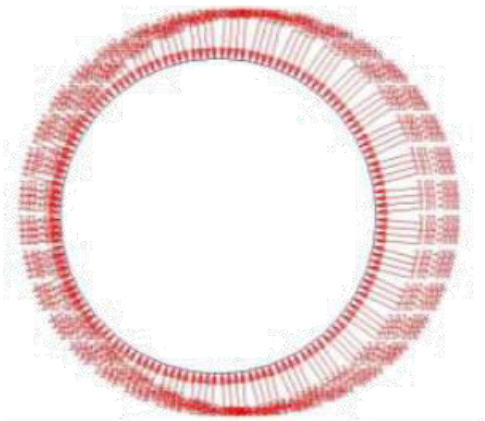

Figure-13. Sea water loading and the amount of pressure for a submarine with swedge frame (300 m).

Loading at depth of 500 m

By using hydrostatic pressure calculation (h: depth surface = 502.125 m), the hydrostatic pressure on the structure of the submarine's pressure hull using swedge frame is:

Ph = 1025 kg/m3 x 9.8m/s2 x 502.125 m
= 5043845.62 N/m2 (Pascal)
= 5.043 Mpa

So, the load received by the structure of the submarine's pressure hull using swedge frame is 5.043 MPa.

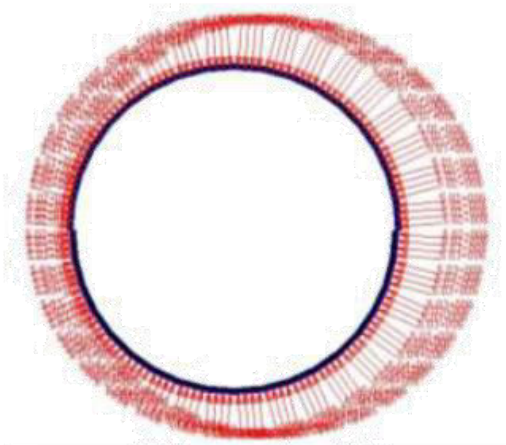

Figure-14. Sea water loading and the amount of pressure for a submarine with swedge frame (500 m).

Body force

Body force in the process of analysis is defined as the gravitational force. Therefore, in this sense, the value of the body force is equal to the force of gravity, i.e. 9.8m/ s2. In the finite element method, it is also known as Inertial Load.

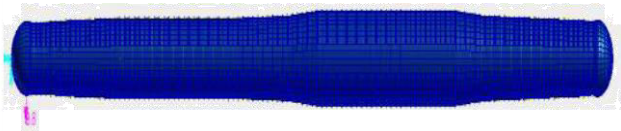

Figure-15. Body force on the swedge frame model.

RESULT

The amount of the maximum voltage accepted by the structure, both the plate and beam (swedge frame), in many variations is as follows:

Plate stress

A depth of 100 m with swedge frame

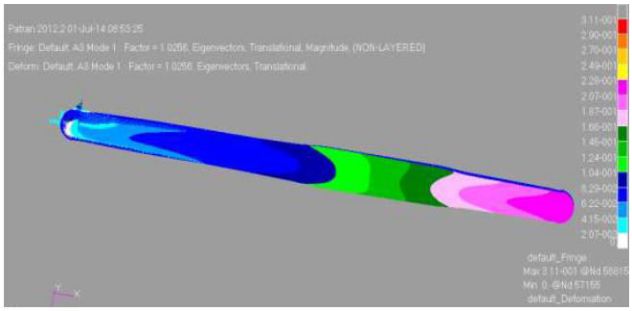

Figure-16. Buckling condition at depth of 100 m.

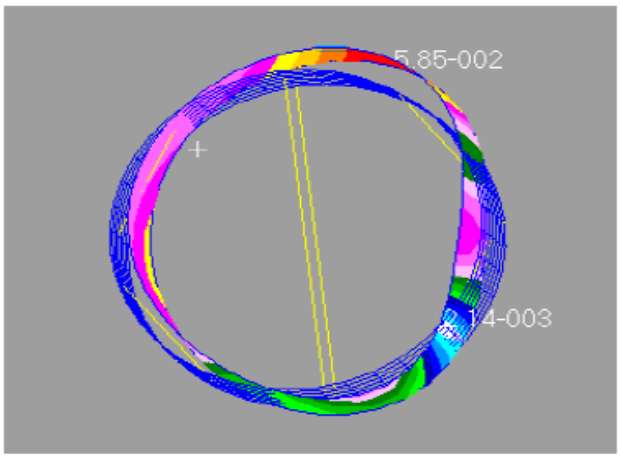

Figure-17. Buckling of the pressure hull structure.

In the case of analysis on the submarine's buckling pressure hull using swedge frame at a depth of 100 m, 5 cases showing each of their eigenvalues are produced.

VOL. 12, NO. 3, FEBRUARY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

Table-1. The value of Buckling Load Factor of pressure hull with swedge frame at a depth of 100 meters.

Mode/Description	BLF (λ)	CBL λ {P}
1 /Pressure Hull Buckling	1,0256	1052107,26
2 /Pressure Hull Buckling	1,0859	1113965,75
3 /Pressure Hull Buckling	1,1068	1135405,93
4 /Pressure Hull Buckling	1,1385	1167925,23
5 /Pressure Hull Buckling	1,1608	1190801,59

From the table above, we come to know that at the time of the Buckling Load Factor reaches 1.0256, the structure receives Critical Buckling Load at 1.05 MPa. This value is the minimum critical value for the pressure hull structure at a depth of 100 m. Meanwhile, when the Buckling Load Factor reaches 1.1608, the structure receives critical buckling load of 1.19 MPa. This value is the maximum critical value for the pressure hull swedge frame structure at a depth of 100 m.

A depth of 300 m with swedge frame

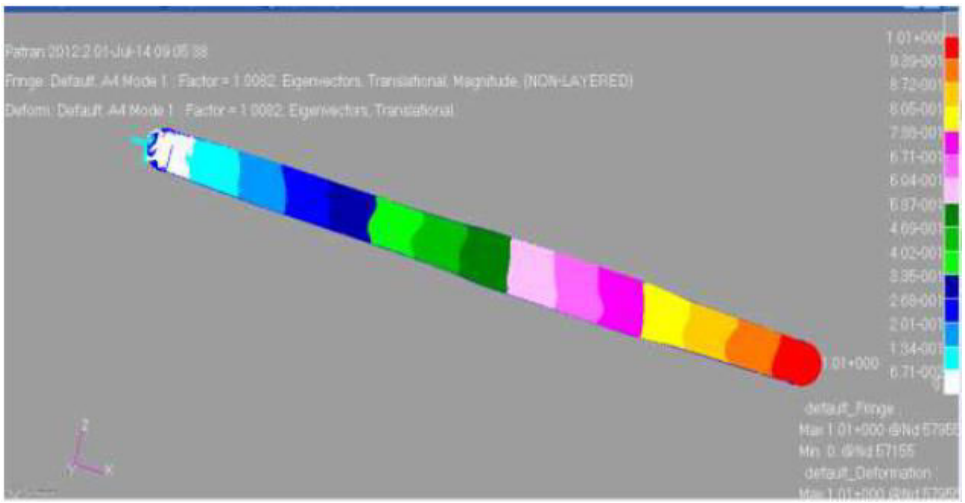

Figure-18. Buckling condition at a depth of 300 m.

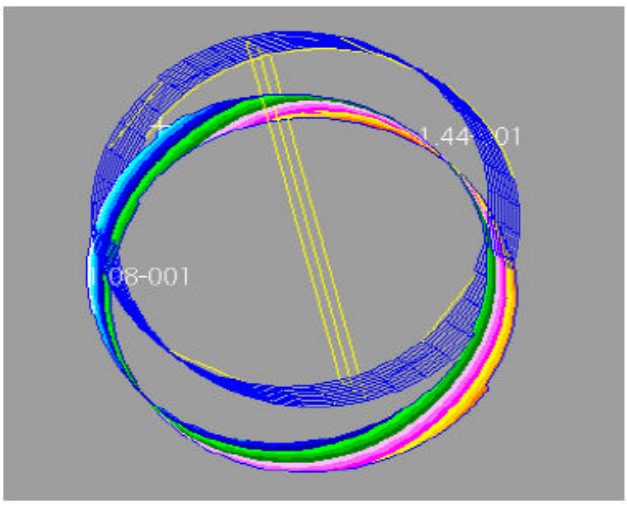

Figure-19. Buckling of the pressure hull structure.

In the case of analysis on the submarine's buckling pressure hull using swedge frame at a depth of 300 m, 5 cases showing each of their eigenvalues are produced.

Table-2. The value of buckling load factor of pressure hull with swedge frame at a depth of 300 meters.

Mode/Deskripsi	BLF (λ)	CBL λ {P}
1 /*Pressure Hull Buckling*	1,0083	3059731,359
2 /*Pressure Hull Buckling*	1,0387	3150776,727
3 /*Pressure Hull Buckling*	1,0529	3195388,958
4 /*Pressure Hull Buckling*	1,0644	3230289,683
5 /*Pressure Hull Buckling*	1,0738	3258817,232

From the table above, we come to know that at the time of the Buckling Load Factor reaches 1.0083, the structure receives Critical Buckling Load at 3.05 MPa. This value is the minimum critical value for the pressure hull structure at a depth of 300 m. Meanwhile, when the Buckling Load Factor reaches 1.0738, the structure receives critical buckling load of 3.25 MPa. This value is the maximum critical value for the pressure hull swedge frame structure at a depth of 300 m.

A depth of 500 m with swedge frame

In the case of analysis on the submarine's buckling pressure hull using swedge frame at a depth of 500 m, 5 cases showing each of their eigenvalues are produced.

Table-3. The value of buckling load factor of pressure hull with swedge frame at a depth of 500 meters.

Mode/Deskripsi	BLF (λ)	CBL λ {P}
1 /*Pressure Hull Buckling*	1,0017	5048568,076
2 /*Pressure Hull Buckling*	1,0120	5104371,393
3 /*Pressure Hull Buckling*	1,0199	5144218,152
4 /*Pressure Hull Buckling*	1,0247	5168428,611
5 /*Pressure Hull Buckling*	1,0304	5197178,532

From the Table above, we come to know that at the time of the Buckling Load Factor reaches 1.00017, the structure receives Critical Buckling Load at 5.04 MPa. This value is the minimum critical value for the pressure hull structure at a depth of 500 m. Meanwhile, when the Buckling Load Factor reaches 1.0304, the structure receives critical buckling load of 5.197 MPa. This value is the maximum critical value for the pressure hull swedge frame structure at a depth of 500 m.

According to the analysis result using MSC Nastran software, the value of buckling load factor (λ) is obtained. Later, the value will be validated with a correction from the Practical Aspects of Finite Element Simulation stating that for buckling analysis, the value of buckling load factor (λ) must comply a correction of > 1. For this correction, the structure we are analyzing is safe. In the other hand, if the value of buckling load factor (λ) is <1, it indicates that the structure is experiencing a condition where there is no longer a resistance from the structure to withstand the pressure it receives. In other words, the structure reaches a failed structure condition.

Table-4. The value of buckling load factor (λ) pressure Hull swedge frame.

Depth (m)	**Mode**	**Buckling load factor (λ) pressure hull swedge frame**	**Correction (λ)/Status**
100	1	1,0256	> 1 / Safe
	2	1,0859	> 1 / Safe
	3	1,1068	> 1 / Safe
	4	1,1385	> 1 / Safe
	5	1,1608	> 1 / Safe
300	1	1,0083	> 1 / Safe
	2	1,0387	> 1 / Safe
	3	1,0529	> 1 / Safe
	4	1,0644	> 1 / Safe
	5	1,0738	> 1 / Safe
500	1	1,0017	> 1 / Safe
	2	1,0120	> 1 / Safe
	3	1,0199	> 1 / Safe
	4	1,0247	> 1 / Safe
	5	1,0304	> 1 / Safe

Table-5. The value of critical buckling load (λ) swedge framepressure hull.

Depth (m)	**Mode**	**Critical buckling load pressure hull swedge frame** (Mpa)
100	1	1,05
	2	1,11
	3	1,13
	4	1,16
	5	1,19
300	1	3,05
	2	3,15
	3	3,19
	4	3,23
	5	3,25
300	1	5,04
	2	5,10
500	3	5,14
	4	5,16
	5	5,19

REFERENCES

American Bureau of Shipping. Rules for Building and Classing Underwater Vehicles, Systems and Hyperbaric Facilities. Second Printing October 1993. American Bureau of Shipping: New York, USA.

Assakkaf I. A. 2003. Columns: Buckling (Pinned Ends) (Doctoral dissertation, University of Maryland, College Park).

Bae D. M., Prabowo A. R., Cao B., Zakki A. F. and Haryadi G. D. 2016. Study on collision between two ships using selected parameters in collision simulation. Journal of Marine Science and Application. 15(1): 63-72.

Carlberg H. 2011. Concept Design of a Commercial Submarine. Trondheim: NTNU.

Clough R. W. 1960. The Finite Element Method in Plane Stress Analysis. Proceedings of 2nd ASCE Conference on Electronic Computation, Pittsburgh, PA, September 8-9.

Friedman N. 1984. Submarine design and development. London: Conway Maritime.

Ghali A. and Neville A. M. 1978. Structural Analysis, 2. Aufiage Chapman and Hall, London.

Gorman J. J. and Louie L. L. 1991. Submersible pressure hull design parametrics. SNAME Trans. 99: 119-146.

Hyper Works. 2012. Practical Aspects of Finite Element Simulation, Altair.

Jackson H. A., Fast C., Abels F., Burcher R., Couch R., Wood F. and Allmendinger E. 1992. Fundamentals of submarine concept design. Discussion. Transactions-Society of Naval Architects and Marine Engineers. 100, 419-448.

Prabowo A. R., Bae D. M., Sohn J. M. and Zakki A. F. 2016. Evaluating the parameter influence in the event of a ship collision based on the finite element method approach. International Journal of Technology. 7(4): 592-602.

Ross C. T. 1987a. A novel submarine pressure hull design. Journal of Ship Research. 31(3): 186-188.

Ross C. T. 1987b. Design of dome ends to withstand uniform external pressure. Journal of Ship Research. 31, 139-143.

Ross C. T. 1992. Collapse of inverted hemi-ellipsoidal shell domes underuniform pressure. Journal of Ship Research. 36: 378-386.

Ross C. T. and Palmer A. 1993. General instability of swedge-stiffened circular cylinders under uniform external pressure. Journal of Ship Research. p. 37.

Sibarani Hendry Maringan. 2013. Desain Struktur dan Analisa Karakteristik Kekuatan Buckling pada Ring Frame Pressure Hull Kapal Selam dengan menggunakan Finite Element Analysis. Semarang: Universitas Diponegoro.

Yuan K. Y., Liang C. C. and Ma Y. C. 1991. Investigations of the cone angle of a novel swedge-stiffened pressure hull. Journal of ship research. 35(1): 83-86.

Zakki A. F., Windyandari A. and Bae. 2016. The development of new type free-fall lifeboat using fluid structure interaction analysis. Journal of Marine Science and Technology. 24(3): 575-580.