Structural Modeling Using 2D Shell Element to Predict the Initial Plastic Condition of Tubular Frame Structures

Asykur*, Y S Hadiwidodo¹, and Handayanu¹

¹Department of Ocean Engineering, Faculty of Marine Technology, Institut Teknologi Sepuluh Nopember (ITS), Surabaya, Indonesia

*Email: mas.asykur@gmail.com

Abstract. A Fixed Jacket Platform is usually designed using tubular structures in which every steel pipe has a thickness and diameter. Generally, for the structural analysis of this structure, the tubular members are represented using 1D beam elements. But stress distribution, plastic strain and also ovalization around the tubular member cannot be determined by a beam element. In this research, all tubular structures were modeled using 2D shell elements to identify how the stress distribution around a tubular member is. And the non-linear analysis was carried out to identify the initial plastic condition. This study was focusing on the splash zone where this area was assumed to be exposed to the extreme wave in 8 directions according to the API RP2A-WSD code, in addition to the structure weight and deck load. Due to each direction of the wave loads, the stress distribution and initial plastic condition are studied. The modeling and non-linear static analysis in this study were done using Altair Hyperworks software.

1. Introduction

The tubular frame is one type of structure commonly used in many constructions such as vehicle frames, cranes, and towers including the offshore platform structure. In this research, the strength of a fixed jacket platform was studied. The fixed jacket platform is a huge steel-framed structure used for the exploration and extraction of oil and gas from the earth’s crust in shallow water depth that typically consists of tubular members of various diameters and wall thicknesses.

The offshore structures standards i.e. API (American Petroleum Institute) RP2A LRFD and RP2A WSD that normally used for designing Offshore Jacket Platforms provides detailed procedures to assess the performance of the jacket platform [1]. In API RP2A LRFD (Load and Resistance Factor Design) there are many parameters that mandatory to be used for planning, designing and constructing fixed offshore platforms including assessment of loads and specific events that could occur for the selected platform over its intended service live and service functions, while API RP2A WSD (Working Stress Design) provides detailed procedures for the assessment of existing platform structures.

Based on that standard, assessment of jacket platforms has rarely been studied. Abdel Raheem [2] evaluated nonlinear response of fixed jacket offshore platform under structural and wave loads and used SAP2000 software for the analysis. Alessi et al. [3] proposed a new simplified tool used to study the structural analysis of the jacket structure and the retrofitted wind tower configuration using MATLAB and Strand7. Fayazi and Aghakouchak [4] detailed reliability-based Assessment of existing fixed offshore platforms using The Software for Analysis of Ultimate Strength for Framed Offshore
Structures (USFOS). Zadeh et al. [5] analyzed linear response of a fixed offshore platform under the wave forces using SAP2000 software. Sandhya [6] described multiple loads on offshore jacket structure analysis using SACS software. Alhasan and Ibrahim [7] studied the specific parameters for the elements of an existing fixed platform using SACS software. Taheri and Jahangir [8] described the nonlinear static pushover analysis of an existing fixed platform using SACS software. Hadiwidodo and Prastianto [9] studied the response of a fixed offshore platform subjected to extreme wind and wave loading (lateral loads) under corrosion conditions using SACS software. Ali Sari et al. [10] illustrated structural corrosion modeling of fixed offshore platform for life Extension Study using Abacus software.

Except for the last mentioned, all of the above studies are using 1D beam element for simulating the tubular structure, which cross-section characteristics of the beam such as cross-section area, inertia, and torsional constant are the most important parameter. For tubular structures with small diameters, modeling as 1D beam elements may be sufficient, but for tubular structures with relatively large diameters such as fixed jacket platforms, it is not enough because stress concentrations can start from one side of the tubular pipe surface, and also corrosion usually occurs on a part of the tubular pipe surface that cannot be represented by 1D beam element accurately.

2. Literature review
According to Cook [11], there are the advantage of using 2D shell element.

2.1. 2D Plate and Shell Element
Plates may be considered similar to beams, however:
– Plates can bend in two directions
– Plates are flat with a thickness (can’t have an interesting cross-section)

Shell elements are different from plate elements in that:
– They carry membrane and bending forces.
– They can be curved.

The most simple shell element combines a bending element with a membrane element.
– E.g., combines a plate element and a plane stress element.
– These elements are flat, therefore it is important that elements are not all co-planar where they meet at a node.

To do a proper FE analysis, the analyst must understand how the structure is likely to behave and how elements are able to behave. In some cases it is more appropriate to use shell elements rather than beam elements. A curved I-beam reacts to moments as shown, therefore shell elements would more accurate than beam elements. Pipe bends react to moments as shown. Use shell elements or specialized beam elements with correction factors.

![Figure 1. Beam moment](image)

According to above theory, to get more accurate result, the tubular structure must be modeled using 2D Shell element than beam element.
In this study we were analyzed the initial plastic condition of a fixed jacket platform structure, and non-linear analysis was carried out to identify the initial plastic conditions. This study focused on the splash zone area (the area immediately above and below the mean water level) where this area is assumed to be exposed to the extreme wave in 8 directions according to the API RP2A Working Stress Design (WSD) code, in addition to the structure weight and deck load.

3. Analysis Approach

3.1. Analysis Software
The program used in this research is *Atair Hyperworks* which is based on finite element method (FEM) modeling, the phase of modeling and plastic strength analysis using non-linear static analysis. This software has the ability to offer a wide array of analysis and design to its user, it has many tool for modeling a structure using 1D, 2D, and 3D element, checking element quality, and its can be modified easily. This software is also able to solve non-linear problem and report the structural responses caused by wind, sea waves, and other related loads. One of the important features of this module is the file size and required system are relatively smaller than other module in same case, so this can analyze a complex structure and its loading condition such a giant vessel or other structure by the combination of beam element and shell element.

3.2. Analysis Units
English Engineering units was used in this study. Inch, pound force, Psi are used for length, force, and stress/pressure consecutively.

3.3. Platform Structural Modeling
The platform structure in this study consisted of four leg with 34-in diameter and had five horizontal framing at EL. (-) 180'-0", EL. (-) 125'-0", EL. (-) 75'-0", EL. (-) 30'-0", and EL. (+) 12'-0". The jacket work point elevation was located at EL. (+) 17'-0". All structures were modeled using shell element, except deck beam face was modeled using beam element. Depend on pipe diameter, mesh size were about 5 in - 10 in, and mesh size at splash zone (the part near the water surface) was about 1 in, with fixed boundary condition at the bottom of jacket. The structure section drawing was shown in Figure 5, and structural modeling using *Altair Hyperworks* was shown in Figure 2.

All steel plate, bars and tubular with \( \varnothing < 18 \) in was using ASTM A36 steel grade, and API 5L Grade B for tubular with \( \varnothing > 18 \).
Table 1. Properties of mechanical steel used in the platform

| Property Type      | ASTM A36       | API 5L Grade B |
|--------------------|----------------|----------------|
| Specific Weight    | 0.284 lb/in³   | 0.284 lb/in³   |
| Elastic Modulus    | 2.9 x 10⁷ psi  | 2.9 x 10⁷ psi  |
| Shear modulus      | 1.16 x 10⁷ psi | 1.16 x 10⁷ psi |
| Poisson ratio      | 0.26           | 0.26           |
| Yield Strength     | 3.63 x 10⁴ psi | 3.5 x 10⁴ psi  |
| Ultimate Strength  | 5.8 x 10⁵ psi  | 6.0 x 10⁵ psi  |

3.4. Loading Condition
Gravity load, on deck load, and extreme wave load in 8 direction were applied at this analysis. Gravity load was calculated automatically by inputting specific gravity into the model. On deck load was modeled as distributed force, and wave load was modeled as pressure and current flow directions of 20°, 65°, 110°, 155°, 200°, 245°, 290°, and 335°. Non-linear static analysis was applied for evaluating initial plastic condition.

Figure 3. Wave load direction

4. Analysis Result
These analysis results focused on the stress distribution and initial plastic condition at the splash zone due to extreme wave condition.

4.1. Stress Distribution
The equivalent stress was evaluated in this study. Figures 4-11 present the equivalent stress distribution on tubular structure related to wave direction. The stress value are in Psi and the part in red means the stress value in that parts are exceed yield stress of material. As indicated in the figure, the most of stress concentration are in the tubular joint between working deck members and diagonal branch members of the platform.

Figure 4. Stress distribution (wave load 20°)
Figure 5. Stress distribution (wave load 65°)
4.2. Plastic Condition

Based on above stress results, the plastic condition most likely occurred in the tubular joint between working deck members and diagonal branch members of the platform. Four locations as shown in Figure 16 were considered to be evaluated. The plastic strain result was summarized in Tables 2 - 4 and the maximum plastic strain in each location are shown in Figures 13 - 16.
4.2.1. Summary of Plastic Strain Result. According to the result of this analysis, the most initial plastic condition has occurred in the above-mentioned location in any wave direction. It is indicated by stress value in that location exceeded the ultimate strength of the material. And by using non-linear plastic analysis in software Hyperworks, the plastic condition is indicated by plastic strain value. If the value is between 0 to 0.001, it means the strain is elasto-plastic, but if it exceeds 0.001, it means that locations are in plastic condition. Plastic strain results at 4 locations in 8 loading directions are shown in Table 2.

### Table 2. Summary of plastic strain result

| Load direction | Location | Stress (psi) | Plastic strain | Deformation |
|----------------|----------|--------------|----------------|-------------|
| 20 deg         | A        | 62930        | 0.0012         | plastic     |
|                | B        | 56910        | 0.0010         | plastic     |
|                | C        | 58068        | 0.0010         | plastic     |
|                | D        | 54250        | 0.0008         | elasto-plastic |
| 65 deg         | A        | 74163        | 0.0017         | plastic     |
|                | B        | 55917        | 0.0009         | elasto-plastic |
|                | C        | 66912        | 0.0014         | plastic     |
|                | D        | 55657        | 0.0007         | elasto-plastic |
| 110 deg        | A        | 79009        | 0.0019         | plastic     |
|                | B        | 62028        | 0.0012         | plastic     |
|                | C        | 68008        | 0.0014         | plastic     |
|                | D        | 55206        | 0.0009         | elasto-plastic |
| 155 deg        | A        | 60054        | 0.0011         | plastic     |
|                | B        | 71819        | 0.0016         | plastic     |
|                | C        | 53087        | 0.0008         | elasto-plastic |
|                | D        | 68554        | 0.0015         | plastic     |
| 200 deg        | A        | 56848        | 0.0010         | plastic     |
|                | B        | 74430        | 0.0018         | plastic     |
|                | C        | 55636        | 0.0009         | elasto-plastic |
|                | D        | 70297        | 0.0016         | plastic     |
From the above data, the loading directions were sorted based on the plastic strain value in Table 3, and the maximum plastic strain value in each location are shown in Table 4.

Table 3. Maximum plastic strain by loading direction

| No. | Plastic strain | Load direction |
|-----|----------------|----------------|
| 1   | 0.00193        | 110 deg        |
| 2   | 0.00176        | 200 deg        |
| 3   | 0.00173        | 65 deg         |
| 4   | 0.00167        | 290 deg        |
| 5   | 0.00164        | 155 deg        |
| 6   | 0.00162        | 245 deg        |
| 7   | 0.00126        | 335 deg        |
| 8   | 0.00123        | 20 deg         |

Table 4. Maximum plastic strain by location

| No. | Plastic strain | Location | Load direction |
|-----|----------------|----------|----------------|
| 1   | 0.00193        | A        | 110 deg        |
| 2   | 0.00176        | B        | 200 deg        |
| 3   | 0.00167        | C        | 290 deg        |
| 4   | 0.00157        | D        | 200 deg        |
Figure 13. Plastic condition at location A

Figure 14. Plastic condition at location B

Figure 15. Plastic condition at location C

Figure 16. Plastic condition at location D

At the above figures, the plastic condition based on the above figures, the initial plastic condition has occurred in the intersection between working deck members and diagonal branch members of the platform.

4.2.2. Overall Plastic Analysis Result. In addition, the plastic analysis result of the whole platform structure were resumed in Table 5 and Figure 17. The plastic strain was constant at a certain value depending on the loading conditions.

| Iteration | Wave direction |
|-----------|----------------|
|           | 20 deg | 65 deg | 110 deg | 155 deg | 200 deg | 245 deg | 290 deg | 335 deg |
| 0         | 0      | 0      | 0       | 0       | 0       | 0       | 0       | 0       |
| 1         | 4.48E-03 | 2.01E-03 | 2.22E-03 | 4.97E-03 | 5.05E-03 | 3.28E-03 | 2.66E-03 | 4.11E-03 |
| 2         | 8.00E-03 | 2.85E-03 | 3.81E-03 | 9.19E-03 | 9.18E-03 | 3.66E-03 | 2.82E-03 | 7.39E-03 |
| 3         | 8.06E-03 | 2.89E-03 | 3.86E-03 | 9.27E-03 | 9.25E-03 | 3.77E-03 | 2.82E-03 | 7.45E-03 |
| 4         | 8.06E-03 | 2.89E-03 | 3.86E-03 | 9.27E-03 | 9.25E-03 | 3.77E-03 | 2.82E-03 | 7.45E-03 |
Figure 17. Plastic strain curve

4.3. Ovalization of Tubular Frame

The difference between the beam frame and tubular frame is which an extreme load can cause the ovalization of tubular frame. Ovalization is the deviation of the tubular shape from a circle to an elliptic cross-section or the process of changing to an oval shape that can be evaluated by 2D shell elements modeling as shown in Figure 18.

The maximum ovalization of each evaluation locations were summarized in Table 6.

| Location | Diameter of horizontal frame (in) | Ovalization (in) |
|----------|----------------------------------|-----------------|
| A        | 16                               | 0.02            |
| B        | 16                               | 0.65            |
| C        | 16                               | 0.06            |
| D        | 16                               | 0.06            |

Figure 18. Process of ovalization
5. Conclusion
A methodology for structural modeling in strength assessment of tubular structure using 2D shell elements was studied. The approach was applied in case studies in which extreme wave loads in 8 directions were assumed applied on a four legs fixed jacket platform. The analysis results indicate that the critical point of this jacket platform structure is the tubular joint between working deck members and diagonal branch members of the platform. In this analysis, the ovalization value of the tubular frame is quite small compared with its diameter, but when an extreme concentration load is working on a part of the tubular frame, the ovalization can be more significant.

On a big tubular structure modeling such as fixed jacket platform, using 2D shell element is more suitable than using 1D beam element to get more accurate and detail result. Also, due to some tubular member must be watertight in which hydro-static pressure work on it, it can be represented only by 2D element or 3d element. Furthermore, Strength assessment of existing structure under corrosion can be done more accurately by modeling corrosion’s shape and thickness using 2D shell element based on an inspection data. But, the effectiveness of this analysis depends on the software performance and hardware capacity related to the modeling and analysis time.

Acknowledgments
The authors highly appreciate to PT. Terafulk Ship Design, Surabaya, Indonesia who have allowed using software Altair Hyperworks for this study.

References
[1] American Petroleum Institute 2000 Recommended Practice for Planning Designing and Constructing Fixed Offshore Platforms - Working Stress Design 21st edition (Washington, District of Columbia USA API Publishing Services) chapter 17.
[2] Abdel Raheem S E 2013 Nonlinear Response of Fixed Jacket Offshore Platform Under Structural and Wave Loads Coupled Systems Mechanics Vol 2 No. 1 pp 111-126.
[3] Lorenzo Alessi José A F O Correia and Nicholas Fantuzzi 2019 Initial Design Phase and Tender Designs of a Jacket Structure Converted into a Retrofitted Offshore Wind Turbine Energies 2019 12, 659.
[4] Alireza Fayazi and Aliakbar Aghakouchak 2015 Reliability Based Assessment of Existing Fixed Offshore Platforms Located in the Persian Gulf International Journal of Maritime Technology vol 4 pp 37-50.
[5] Zadeh S M G Payambarnia P Baghdar R S Alikhani E T 2015 Finite Element Analysis of Offshore Jacket Affected by Marine Forces Bulletin of Environment, Pharmacology and Life Sciences Vol 4 [Spl issue 1] pp 152-159.
[6] Gande Sandhya 2018 Analysis of Offshore Jacket Structure International Research Journal of Engineering and Technology vol 5 pp 722-729.
[7] Mahar Jasem Alhasan and Adnan Mohamed Ibrahim 2018 American Journal of Innovative Research and Applied Sciences ISSN 2429-5396 pp 91-96
[8] Abdolrahim Taheri and Mohammad Jahangir 2017 International Journal of Coastal & Offshore Engineering vol 1 No 3 pp 57-64.
[9] Hadiwidodo Y H and Prastianto R W 2018 Structural Corrosion Modeling for Reserve Strength Assessment of Fixed Jacket Platform International Journal of Civil Engineering and Technology vol 9 pp 1006-1012.
[10] Ali Sari, Ekkirala R, Nguyen H, Arablobei A and Azimov U 2016 Structural Corrosion Modeling for Strength Assessment of Fixed Offshore Platforms for Life Extension Study Offshore Technology Conference OTC-26521-MS.
[11] Cook R D Malkus D and Plesha M E 1996 Concepts and Applications of Finite Element Analysis 4th Edition (New York: Wiley–Interscience).