Step by Step Stress Analysis of Ammonia Scrubber Vertical Pressure Vessel Using Finite Element Method

Arhami, Amir Zaki Mubarak, Andika Saputra, Masri Ali

Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Syiah Kuala, Banda Aceh, Indonesia

E-mail: arhami@unsyiah.ac.id

Abstract. In order to solve the stress problem of ammonia scrubber pressure vessel, this paper proposes the analysis steps using finite element method. The study is a type of vertical pressure vessel used to remove pollutants in ammonia gas. Firstly, the modeling of the vertical pressure vessel geometry was carried out using Autodesk Inventor Professional 2017. Secondly, the meshing was carried out using Autodesk Nastran In-CAD 2017. For the meshing, we used tetrahedral element and we chose the parabolic type element order. Thirdly, the analysis was carried out using Autodesk Nastran In-CAD and based on the design pressure of 0.5 MPa with a volume of 1.21 m³. The results showed that the maximum stress occurred in the connection area between the inspection opening and the shell of 333.4 MPa. The maximum displacement occurred in the shell adjacent to the inspection opening was 3.268 mm. Based on ASTM standards on SA 240/A240M Gr 304 material, the maximum strength of material stress was 515 MPa. Finally, theoretical validation was carried out for the entire model, and the results were within the permissible limit.

1. Introduction

Based on field conditions, popping gas from the low-pressure system could not be handled by the existing ammonia scrubber because the gas mixture entering the existing ammonia scrubber is already too much, so the loaded gas that must be absorbed has exceeded the capacity. To overcome this problem, it was planned to install a special ammonia scrubber for gas output from the low-pressure system so that the exhaust gas is not exposed to the environment [1].

Ammonia scrubber in its design requires high accuracy because the ammonia is a dangerous substance. The author will design and analyze the stress occurred in the ammonia scrubber pressure vessels. Stress in a pressure vessel cannot be ignored since it is the basis for determining the dimensions and the type of material used in a pressure vessel.

In the designed step of pressure vessels, the most important thing is to take into account the stresses that appear on the walls of the pressure vessels. Stresses that occur can be caused by external factors such as disaster effect of wind and earthquake loads as well as internal factors caused by work pressure instability and the weight of the vessel itself. The modeling was completed on a modeling software Autodesk Inventor Professional, and a finite element analysis assisted by Autodesk Nastran In-CAD software was carried out to highlight the stress concentration and displacement of various points.
The pressure vessels analyzed were the vertical type with the ammonia gas fluid and assumed as a static load. The analytical application and software approaches are expected to be able to provide the stress analysis step that occurs in pressure vessels so that the magnitude and position of the maximum stress can be known, in order to avoid undesirable things [2]. To be used safely, the design results and stresses refer to ASTM standards on SA 240/A240M Gr 304 material [3]. If the stress occurred is greater than the yield strength of the material used, it will cause the material failure. Theoretical validation was carried out for the entire model, and the results were within the permissible limit.

**Stress condition of pressure vessel**

The stresses in pressure vessels caused by operating loads are divided into two types namely if the internal pressure (P) is less than 1/6 of the permissible stress, it is called a thin-walled vessel, and if the internal pressure (P) is greater than 1/6 of the allowable stress, it is called a thick-walled vessel [4]. In this study, the pressure vessel as shown in Fig. 1 is assumed to be a thin cylinder. Therefore, the analysis follows the thin cylinder formulae.

a) Longitudinal stress

\[ \sigma_L = \frac{PD}{4t} \]  
(1)

b) Circumferential stress

\[ \sigma_\theta = \frac{PD}{2t} \]  
(2)

where \( t \) is the wall thickness in mm, \( P \) is the internal pressure in MPa, \( D \) or \( 2r \) is the inside diameter of vessel in mm, \( \sigma_L \) is the longitudinal stress in MPa, and \( \sigma_\theta \) is the circumferential stress in MPa.

![Figure 1. Stress conditions on the cylinder wall of the pressure vessel [4]](image)

**Stress condition of Head**

Stresses occurred on the head in several locations are acquired using the following formula, a) At the junction of crown and knuckle area

\[ \sigma_L = \frac{PL}{2t}, \text{ and } \sigma_\theta = \frac{PL}{4t} \left( 3 - \frac{L}{R} \right) \]  
(3)
b) Inside of crown area
\[
\sigma_L = \frac{PL}{2t}, \quad \text{and} \quad \sigma_\phi = \sigma_L
\]  
(4)

c) Inside of knuckle area
\[
\sigma_L = \frac{PL}{2t}, \quad \text{and} \quad \sigma_\phi = \frac{PL}{t}\left(1 - \frac{L}{2r}\right)
\]  
(5)

d) At the tangent line
\[
\sigma_L = \frac{PR}{2t}, \quad \text{and} \quad \sigma_\phi = \frac{PR}{t}
\]  
(6)

where \(L\) is the crown radius in mm, \(R\) is radius of head in mm, and \(r\) is radius of knuckle in mm.

**Equivalent Stress**

Equivalent stress (equivalent) is obtained from the relationship of three-dimensional main stresses namely the longitudinal, circular, and radial directions. There are denoted as \(\sigma_L\), \(\sigma_\phi\), \(\sigma_r\), and for equivalent stress \(\sigma'\), then the formula can be written as follows [5]

\[
\sigma' = \sqrt{\frac{(\sigma_L - \sigma_\phi)^2 + (\sigma_\phi - \sigma_r)^2 + (\sigma_L - \sigma_r)^2}{2}}
\]  
(7)

If the stresses occurred only in circumference and longitudinal directions, the equation (7) becomes

\[
\sigma' = \sqrt{\sigma_L^2 + \sigma_\phi^2 - \sigma_L\sigma_\phi}
\]  
(8)

2. **Finite Element Analysis**

Finite element analysis is one of the numerical methods that can be used to obtain solutions for various engineering problems. This method is widely used to calculate the structure due to the loading or other effects on the structure. The main objective is to obtain the value of the stress approach occurred in the structure. This approach is an analysis based on stress or strain assumptions [6].

**Modeling Step of Vertical Pressure Vessel**

In this step, the 3D geometry model of vertical pressure vessels created by the Autodesk Inventor Professional 2017. Fig. 2 shows the models made in 2D and 3D forms based on the designed results.
The proposed dimension and design specifications of the pressure vessels analyzed and material grades are listed in Table 1 and 2.

### Table 1. Pressure vessel specification and design data

| Fluid          | Ammonia gas |
|----------------|-------------|
| Density (ρ)    | 0.73 kg/m³  |
| Operating pressure (P₀) | 0.44 MPa |
| Design pressure (Pᵥ) | 0.5 MPa  |
| Operating temperature (T₀) | 35°C   |
| Design temperature (Tᵥ) | 100°C |
| Inside diameter (Dₒ) | 750 mm  |
| High of shell (H)    | 2500 mm   |
| High of head (h)     | 187.5 mm  |
| Thickness of shell (t) | 3.3 mm |
| Crown radius (L)     | 675 mm    |
| Knuckle radius (r)   | 128.9 mm  |
| Corrosion factor     | 0.5       |
| Capacity            | 1.21 m³   |

### Table 2. Material specification

| Component type | Material grade |
|----------------|----------------|
| Shell          | SA 240 Gr.304 |
| Head           | SA 240 Gr.304 |
| Nozzle         | SA 182 Gr.304 |

**Meshing Step of Vertical Pressure Vessel**

At the pre-processing stage, there are several things determining the division of the elements (meshing) namely the determination of the type of material, idealization and selection of the elements. The element used to conduct stress analysis on pressure vessels is the tetrahedral type.
Meshing on the vertical pressure vessels is conducted thoroughly. The first step that must be done is to set the size of the element to 15 mm with a tolerance of 0.00272846 mm, and then choose the element order as parabolic. Fig. 3 shows the results of the division of elements (meshing) with the type of tetrahedral elements. The total numbers of nodes are 532118 and elements 269371.

![Meshing model of equipment](image1)

**Figure 3.** Meshing model of equipment

**Stress Analysis Step of Vertical Pressure Vessel**

This step is a determination for loading on pressure vessels. The degree of freedom which becomes a constraint or boundary condition was also determined so that the reference surface is fixed. Loading in pressure vessel analysis was carried out under internal pressure uniform acting on the entire surface of the inner cylinder as shown in Fig. 4 and 5. The loading was applied to the inner pressure vessel with a pressure of 0.5 MPa.

![Application of pressure load in the pressure vessel](image2)

**Figure 4.** Application of pressure load in the pressure vessel
In principle, the stresses occurred in pressure vessels caused by operating loads are longitudinal stresses ($\sigma_L$) and circular stresses ($\sigma_\phi$). The formulation used and stress calculation results can be seen in Table 3.

| Stress location                  | Equation | $\sigma_\phi$, MPa | $\sigma_L$, MPa |
|---------------------------------|----------|--------------------|-----------------|
| Shell                           | (2) and (1) | 56                | 28              |
| Junction of crown and knuckle   | (3)      | 30                 | 50              |
| Inside crown                    | (4)      | 50                 | 50              |
| Inside knuckle                  | (5)      | -162              | 50              |
| Tangent line                    | (6)      | 56                 | 28              |

3. Result and discussion

The results of stress distribution simulations on ammonia scrubber pressure vessels can be seen in Fig. 6 (a). The figure shows the several colors produced due to the stress conditions occurred in pressure vessels. It appears that the maximum stress occurred in the connection area between the inspection opening and the shell. Fig 6 (b) shows that the value of the largest stress distribution occurred in the X-Y plane was at location 13 namely the curvature of the lower head where the stress value was 124.4 MPa, and the smallest one was seen at location 11 in the connection between the lower head and shell where the stress value was 25.22 MPa.

Fig. 7 (a) shows the location points determined to the Y-Z plane based on the color difference (stress value). Fig. 7 (b) shows that the value of the largest stress distribution occurred in the Y-Z plane was at location 11 namely the connection area between the inspection opening and the shell with a stress value of 333.4 MPa, the smallest stress occurred at location 2 precisely in the area of the peak of the upper head with stress rated of 35.5 MPa.
Figure 6. (a) The stress distribution location determined against the X-Y plane, (b) Graph of the stress distribution value in the X-Y plane.

Figure 7. (a) The stress distribution location determined against the Y-Z plane, (b) Graph of the stress distribution value in the Y-Z plane

The value of displacement occurred in pressure vessels can be seen in the following figures. Fig. 8 (a) shows some location points that are determined against the Y-X plane based on the color difference occurred. To find out the amount of displacement occurred in the area can be seen in Fig. 8 (b). The figure shows the greatest displacement value occurred at location 2
precisely in the top head region with a value of 1.505 mm, displacement occurred at the connection of the head with the shell is shown by location 8 with a value of 1.21 mm.

Figure 8. (a) Displacement location determined against the X-Y plane, (b) graphs of displacement values in the X-Y plane.

Figure 9. (a) Displacement location determined against the Z-Y plane, (b) graphs of displacement values in the Z-Y plane.

Fig. 9 (a) shows several locations that are determined against the Y-Z plane based on the color difference occurred as shown in Fig. 9. To find out the amount of displacement occurred in the area can be seen in Fig. 9 (b).
Fig. 9 (b) shows the maximum displacement value occurred at location 8 precisely on the tube body adjacent to the inspection opening with a value of 3.267 mm, displacement occurred at the curvature of the upper head is marked by location 3 with a value of 1.32 mm, while displacement occurred in the lower head region is marked by location 15 with a value of 1.45 mm.

**Theoretical Validation of Stress Analysis**

The calculation results of the failure analysis occurred in the pressure vessel showed the greatest stress occurred on the inside of the knuckle radius shown based on an equivalent stress value of 192 MPa. This shows that the pressure vessel theories is in safe condition, since the value of the equivalent stress obtained does not exceed the value of the yield strength which is equal to 205 MPa.

The simulation results using analysis software show the maximum stress value occurred in the pressure vessel is in the connection area between the inspection opening with the shell of 333.4 MPa. Maximum displacement occurred in the shell adjacent to the inspection opening of 3,268 mm. Based on ASTM standards theories on SA 240/A240M Gr 304 material, the maximum strength of material stress is 515 MPa. Thus, the stress occurred in the pressure vessel design simulation results is 333.4 MPa and still within the permitted limits. It can be concluded that the pressure vessel condition is safe to be used.

**Summary**

In this paper, the steps analysis using finite element method to solve the stress problem on ammonia scrubber pressure vessel was proposed. The study is a type of vertical pressure vessel used to remove pollutants in ammonia gas. The modelling of the vertical pressure vessel geometry was carried out using Autodesk Inventor Professional 2017 and meshing was carried out using Autodesk Nastran In-CAD 2017. For the meshing, we used tetrahedral element and element order we chose the parabolic type. The analysis was carried out using Autodesk Nastran In-CAD and based on the design pressure of 0.5 MPa with a volume of 1.21 m³. The results showed the maximum stress value occurred in the connection area between the inspection opening and the shell of 333.4 MPa. The stress values were less then the ASTM standards on SA 240/A240M Gr 304 material, the maximum strength of material stress is 515 MPa. The maximum displacement occurred in the shell adjacent to the inspection opening was 3,268 mm. Based on theoretical validation carried out for the entire model, the results are within the permissible limit.

**References**

[1] Moss Dennis R., 1987. *Pressure Vessel Design Manual, Illustrated Procedures for Solving Every Major Pressure Vessel Design Problem*. Gulf Publishing Company, Houston.

[2] Manullang E., Tangkuman S., Maluegha B.L, Stress Analysis of Vertical Pressure Vessel 13ZL100040291 at Aneka Gas Industry Ltd. Journal of Mechanical Engineering Dept. Sam Ratulangi University. (2016) 92-105

[3] ASTM, 2010, *Annual Book of ASTM Standard*, ASTM International, 100 Barr Harbor Drive, PO Box C700, West Conshohocken.

[4] Popov E. P., 1990. *Engineering Mechanics of Solid*, Prentice-Hall International, Englewood Cliffs, New Jersey.

[5] Budynas Richard G., Nisbett J.keith, 2015, *Shigley’s Mechanical Engineering Design*. Tenth Edition, McGraw-Hill Education, New York.

[6] Moaveni, Saeed. 1999. *Finite Element Analysis Theory and Application with Ansys*, Prentice Hall, Inc. New Jersey.