Ss. Cyril and Methodius University in Skopje

FACULTY OF MECHANICAL ENGINEERING

Case Study

Static structural analysis using FEA of pressure vessel with Ansys Mechanical

Aleksandra Krstevska and Marjan Gavriloski

Department of Welding and Welded Structures, Faculty of Mechanical Engineering – Skopje, N. Macedonia

Edited and curated by the Ansys Academic Development Team

education@ansys.com

Summary

The main purpose of this case study is to design a pressure vessel according to EN 13445-3 and do a static structural analysis on the pressure vessel. It starts with discussion of the main components of pressure vessels, applications, design parameters, and selection of material. When designing is crucial to define the loadings which are the main cause of stresses of the vessel and to select the appropriate analytical method for analysis. The stresses developed in wall pressure vessel are analyzed by using Ansys. Analysis of cylindrical pressure vessel is performed and maximum equivalent stresses are calculated by Ansys.

Table of Contents

1. Introduction

Pressure vessels are widely used in industry, especially in power and nuclear industry, chemical and petroleum industries. It presents a container with a pressure differential between inside and outside, nozzles are used to satisfy requirements as inlet and outlet connections. Welded nozzles are placed on the shell and the heads of the vessel, connecting a pressure vessel to a piping system.

Due to pressure acting there is a chance for the contained fluid to leak out which can cause serious accident and loss of life. For that reason, the design, fabrication and testing techniques are controlled by few legislation organizations like ASME, EN, BS standards.

A pressure vessel is composed of the following main types of components: shell, head and nozzles. The shell is the primary component that contains the pressure and forms a structure that has rotational axis. The shell is closed at the ends by heads that can be torispherical, ellipsoidal, conical and flat ends. Heads also are used inside the vessel to create separate sections of the pressure vessel and to enable different design conditions in each section. The nozzle is a component that penetrates the shell or heads and usually is flanged to allow for the connections. They are used for attaching the vessel to a piping system (flow into or out), attach instrument connections (thermowells, pressure gauges), attachment to heat exchangers or other equipment.

2. Design of pressure vessel

The design is made according to the European standard EN 13445-3, Unfired pressure vessels – Part 3 – Design. The following discussed design parameters must be taken into consideration during the design process. EN 13445-3 requires the pressure vessel to be designed based on the specific requirements during normal operation, possible deviation, test operation.
Operating pressure – the maximum internal or external pressure during service.
Design pressure – maximum internal pressure when designing the pressure vessel. Design pressure is always higher than the operating pressure.
Operating temperature - Operating temperature is set according to the maximum and minimum metal temperatures that the pressure vessel will encounter during its service life.
Design temperature - The maximum fluid temperature which occurs under normal operating conditions is the design temperature.
Allowable stress – For safe design, the stresses imposed by the loading are limited to the maximum allowable stresses specified in the European Standard EN 13445.
Corrosion allowance – Adding additional thickness of the design thickness of the vessel to prevent thinning of the inner wall of the vessel which can lead to shorter lifespan or hazards of the vessel. Due to operation of long period of time of the vessel the corrosion can happen from chemical reaction on the inner wall surface, rusting of the material, oxidation with the increase in temperature, erosion of the material.

2.1 Material

For the design of the shell is considered structural steel. The property of the material is used for the design calculation.

Table 1: Properties of the material

Density	7.85e-006 kg/mm3
Coefficient of thermal expansion	1.2e-005 C^-1
Specific heat	4.34e+005 mJ/ (kg*C)
Thermal conductivity	6.05e-002 W/(mm* C)
Resistivity	1.7e-004 ohm mm

2.2 Design calculation

The following parameters are given for the cylindrical shell that is subject of analysis of this study:

- Internal pressure, P = 0.5MPa
- External diameter, D = 1000 mm
- Length of the shell, 1500 mm
- Corrosion allowance, 0
- Design temperature, T = 180°C
- Thickness, 5mm
- Tensile Yield Strength, 250 MPa

3. Modeling and analysis

For modeling the geometry is used Ansys SpaceClaim[1] and is created 3D solid structure. The next step is meshing the 3D solid structure in Ansys Mechanical. Meshing quality should be checked after meshing without failing to capture the critical areas with good quality mesh, as shown in Figure 1.

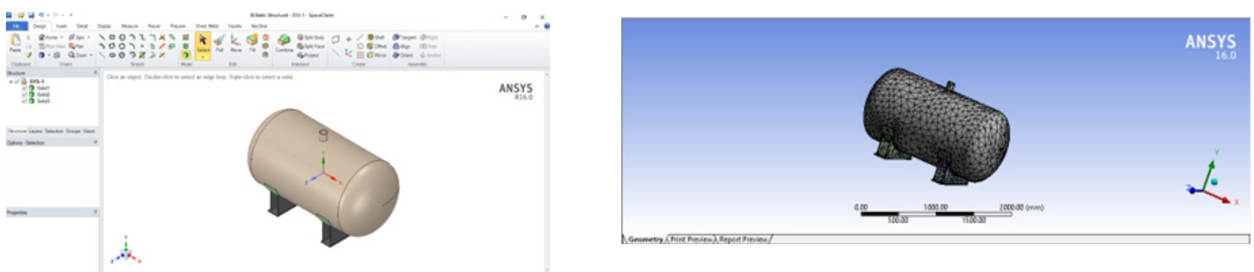

Figure 1: Shell Model and Mesh

The boundary conditions and loads in this step are applied. We apply the symmetry boundary conditions along the X-axis and the pressure is applied on the inner shell surface of the vessel.

4. Results and discussions

For the specified pressure vessel, Ansys will be used to analyze the stresses and deflections in the vessel walls due to the internal pressure. Analysis of cylindrical pressure vessel is performed and maximum equivalent stresses are calculated by Ansys.

1 After the 2023R1 release, Ansys SpaceClaim became a legacy product and Ansys Discovery (a simulation-driven design tool that combines instant physics simulation, high fidelity simulation and interactive geometry modeling in a single easy-to-use experience) became the primary built-in Geometry tool. If you want to learn more about specifically modeling in Ansys Discovery, check out this Ansys Innovation Course "Learn Solid Modeling with Ansys Discovery".

4.1 Directional deformation

The directional deformation is oriented on X-axis and after solving the example we can read the maximum and minimum deformation of the vessel which values are given also in table 2. With red color is presented the maximum value of directional deformation of the vessel and with blue color the minimum directional deformation of the vessel, as seen in Figure 2.

Table 2: Minimum and maximum directional deformation:

Directional deformation	
Geometry	All bodies
Orientation	X-axis
Minimum	-6.069e002 mm
Maximum	1.1662e005 mm

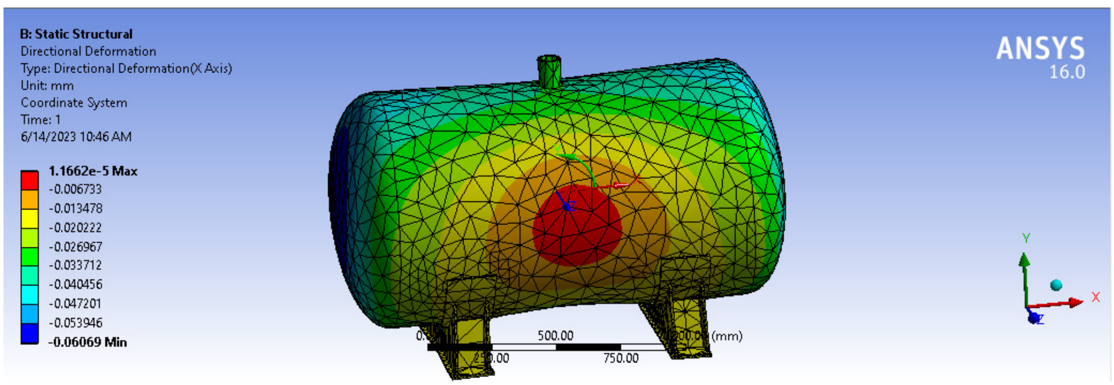

Figure 2: Directional Deformation Stress Results

4.2 Radial stresses

Design pressure is applied on the inner walls of the vessel with the boundary conditions. On table 3 are given the minimum and maximum value of the radial stresses of the pressure vessel according to the X-axis orientation of the vessel, as seen in Figure 3.

Table 3: Radial stresses:

Radial stresses	
Geometry	All bodies
Orientation	X-axis
Minimum	-38.192 MPa
Maximum	7.598 MPa

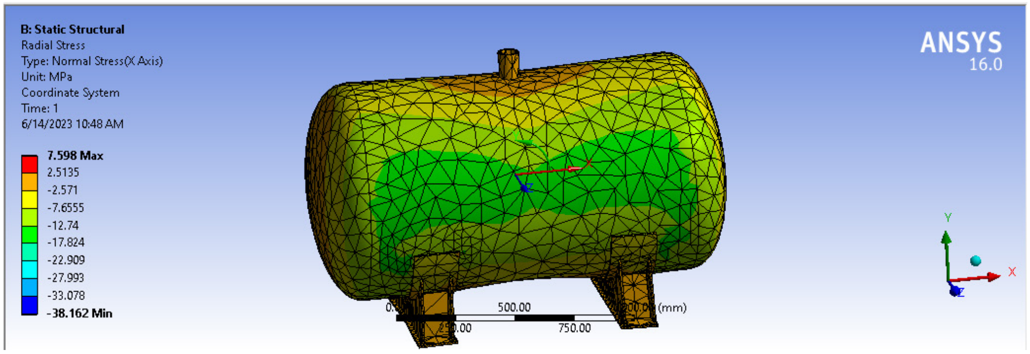

Figure 3: Radial Stress Simulation Results

4.3 Circumferential stresses

For calculation of circumferential stresses Y-axis is chosen as orientation axis of the pressure vessel. On table 4 are presented the minimum and maximum value of the circumferential stresses of the pressure vessel as chosen geometry, shown in the simulation results in Figure 4.

Table 4: Circumferential stresses:

Circumferential stresses	
Geometry	All bodies
Orientation	Y-axis
Minimum	-28.769 MPa
Maximum	-0.18838 MPa

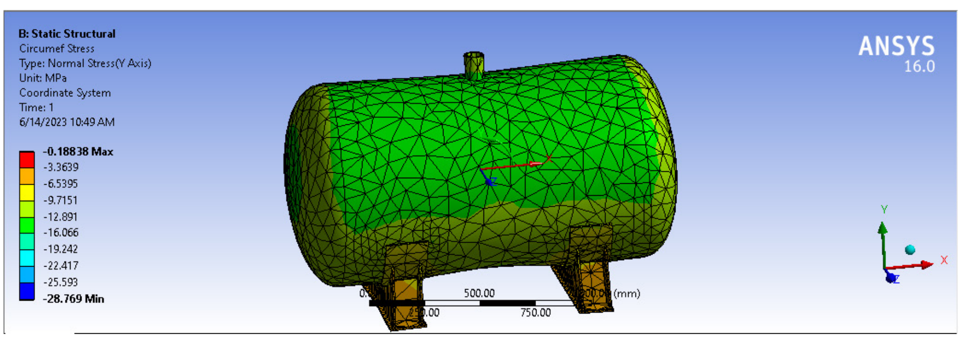

Figure 4: Circumferential Stress Simulation Results

4.4 Normal stresses (scoping method – path, X-axis)

In table 5 and 6 are presented the minimum and maximum values of normal stresses for a chosen geometry of the pressure vessel cylindrical body. The scoping method is a selected path of the cylindrical body with X-axis orientation for table 5 values (Figure 5) and with Y-axis orientation for table 6 values (Figure 6).

Table 5: Normal stresses (path – geometry, X-axis):

Normal stresses	
Geometry	Path
Orientation	X-axis
Minimum	-12.193 MPa
Maximum	-9.6375e^{002} MPa

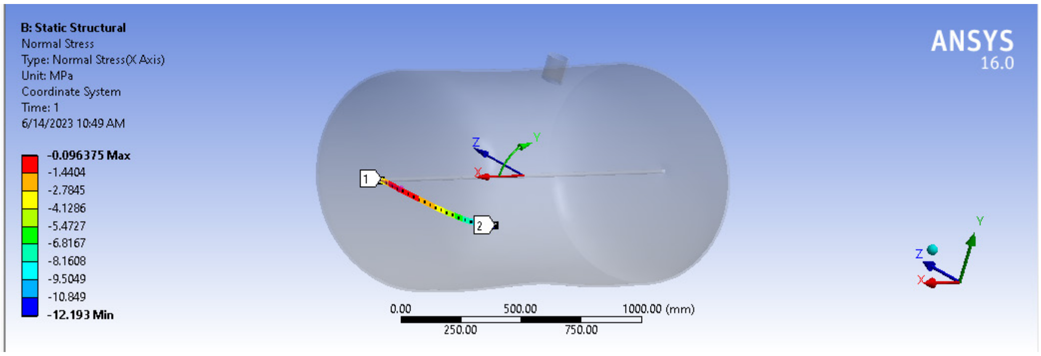

Figure 5: Normal stresses on x-axis

4.5 Normal stresses (scoping method – path, Y-axis)

Table 6: Normal stresses (path – geometry, Y-axis):

Normal stresses	
Geometry	Path
Orientation	Y-axis
Minimum	-13.685 MPa
Maximum	-11.725 MPa

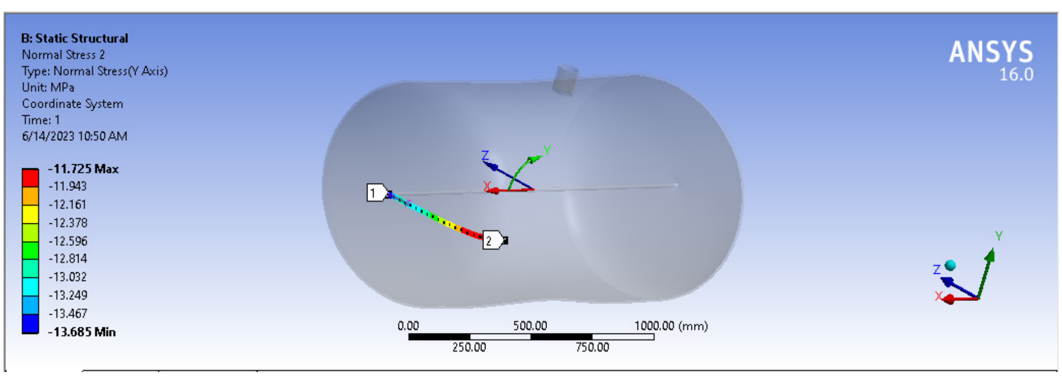

Figure 6: Normal stresses on y-axis

5. Conclusion

In this study, a pressure vessel is designed by recommendation of EN 13445 – design of unfired pressure vessels. The modeling and analysis is carried out in Ansys software. Deformation and stresses of the pressure vessel were subject of investigation. The pressure vessel is subjected to normal and circumferential stresses and the variation of stresses is analyzed by the Ansys results hence the design is safe under the operating conditions. The results are given in table where are indicated the maximum and minimum value of the stress and deformation of the vessel.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com