

# 基于 ANSYS 的高压容器疲劳特性分析

## 摘要

高压容器被广泛的应用在我们生活的方方面面，尤其是在工业中，更是起着非常重要的作用。但是随着石油化工和其他工业的快速发展，使得元件结构和载荷愈发复杂，疲劳破坏成为高压容器失效的重要原因。因此，高压容器疲劳问题成为了需要解决的重要问题。本课题根据有限元技术，对高压容器进行研究，在考虑到材料特点以及静态特性的条件下，进行三维建模、静态有限元分析和疲劳特性有限元分析。本文运用 SolidWorks 和 ANSYS 软件，建立高压容器模型并进行有限元分析，在建模过程中对模型进行必要的简化，以便快捷的进行优化设计计算。根据实际工况，对高压容器进行仿真，分析其应力最大处，是否能够安全运行，结构强度能否满足实际使用需求，疲劳寿命.....

**关键词:** 高压容器，疲劳寿命，SolidWorks，ANSYS

# **Fatigue analysis of high pressure vessel based on ANSYS**

## **Abstract**

High pressure vessel is widely used in all aspects of our life, especially in industry, and plays a very important role. However, with the rapid development of petrochemical industry and other industries, the structure and load of components become more and more complex. Fatigue failure has become an important reason for the failure of high-pressure vessels. Therefore, the fatigue problem of high pressure vessel has become an important problem to be solved. According to the finite element technology, this paper studies the high-pressure vessel, and makes three-dimensional modeling, static finite element analysis and fatigue finite element analysis considering the material characteristics and static characteristics. In this paper, SolidWorks and ANSYS software are used to establish the high pressure vessel model and carry out the finite element analysis. In the process of modeling, the model is simplified, so as to carry out the optimal design calculation quickly. According to the actual working condition, simulate the high-pressure vessel, analyze whether it can operate safely, whether the structural strength can meet the actual use demand, and whether the fatigue life can be

**Key words:** High pressure vessel, fatigue life, SolidWorks, ANSYS

---

# 目录

<b>第一章 绪论</b> .....	<b>1</b>
<a href="#">1.1 课题的目的及意义</a> .....	1
<a href="#">1.2 高压容器国内外研究现状</a> .....	1
<a href="#">1.3 疲劳基础理论</a> .....	2
<a href="#">1.3.1 疲劳累积损伤理论</a> .....	4
<a href="#">1.3.2 疲劳分析方法</a> .....	4
<a href="#">1.4 高压容器研究方向及问题</a> .....	5
<b>第二章 结构设计</b> .....	<b>7</b>
<a href="#">2.1 高压容器设计参数</a> .....	7
<a href="#">2.2 高压容器设计计算</a> .....	7
<a href="#">2.2.1 高压容器筒壁厚度计算</a> .....	7
<a href="#">2.2.2 高压容器封头厚度计算</a> .....	8
<b>第三章 SolidWorks 与 ANSYS 软件简介及高压容器选择</b> .....	<b>9</b>
<a href="#">3.1 SolidWorks 软件简介</a> .....	9
<a href="#">3.2 ANSYS 简介</a> .....	9
<a href="#">3.3 高压容器模型建立以及导入 ANSYS</a> .....	9
<b>第四章 模型导入以及有限元分析</b> .....	<b>11</b>
<a href="#">4.1 有限元分析步骤</a> .....	11
<a href="#">4.2 有限元分析结果</a> .....	12
<a href="#">4.2 改变载荷查看对高压容器带来的影响</a> .....	14
<a href="#">4.2.1 改变载荷查看高压容器应力云图</a> .....	14
<a href="#">4.2.2 改变载荷查看高压容器结构变形量</a> .....	15
<a href="#">4.4 改变高压容器材料查看变形量及应力云图</a> .....	17
<b>第五章 高压容器疲劳特性分析</b> .....	<b>20</b>
<b>第六章 总结与展望</b> .....	<b>21</b>

---

以上内容仅为本文档的试下载部分，为可阅读页数的一半内容。  
如要下载或阅读全文，请访问：

<https://d.book118.com/078141072063006116>