Finite Element Method studies on the stability behavior of external radial loaded cylindrical shells with varying dimensions

made in
Department of Mechanical and Process Engineering Hochschule Niederrhein

presented by
Javier Equiza Lacabe

Matr.-Nr.: 868686
Supervising professor: Prof. Dr. Ing. Conrad Eller

Krefeld, April 2012
Summary of the final project work

Project Title: Finite Element Method studies on the stability behavior of external radial loaded cylindrical shells with varying dimensions.

Author of the project: Javier Equiza Lacabe
Matr.-Nr.: 868686

Supervising professor: Prof. Dr. Ing. Conrad Eller

Supervising organization: Hochschule Niederrhein

Summary of the project: With the help of FEM and ANSYS, the influence of varying dimensions and sections, was tested on the structural behavior of cylindrical shells under external radial load. First, the theoretical concepts concerning cylindrical shells are described. The results of stability analysis were shown in the form of graphs, charts and diagrams.

Keywords on the topic: FEM, stability analysis, buckling analysis, ANSYS

Submission of project: April 2012
Affidavit:

I certify by my signature that this final project work was written solely by me.
There were no uses other than those specified by my sources and resources.

The project consists of 76 Pages (without attachments).

Krefeld, in April 2012

(student signature)
Preface

At this point I would like to thank all those who were helpful to me during the preparation of project.

Specially I am very grateful to Prof. Dr. Ing. Conrad Eller. His consistent support and willingness to help me during the last 9 months have been very important and helpful in preparing my project.
# INDEX OF CONTENTS

1. INTRODUCTION ......................................................................................................................... 1
   1.1 Motivation ........................................................................................................................... 1
   1.2 Task and procedure ............................................................................................................. 1
   1.3 Literature review ................................................................................................................. 2

2. DEFORMATIONS AND STRESSES OF CYLINDERS ................................................................. 5
   2.1 Introduction ......................................................................................................................... 5
   2.2 Thin-walled Cylinders .......................................................................................................... 6
      2.2.1 Stresses ......................................................................................................................... 6
      2.2.2 Thin-walled Cylinder Displacement .............................................................................. 9

3. BUCKLING ................................................................................................................................ 12
   3.1 Introduction ....................................................................................................................... 12
   3.2 Types of Buckling ............................................................................................................... 14
      3.2.1 Non linear collapse ...................................................................................................... 14
      3.2.2 Bifurcation Buckling ................................................................................................... 14
   3.3 Postbuckling of perfect and imperfect systems ................................................................. 15
   3.4 Buckling of thin cylindrical shells ...................................................................................... 16
   3.5 Buckling of thin-walled cylinders under external pressure ............................................... 18
      3.5.1 Introduction ................................................................................................................ 18
      3.5.2 Ways of solving the buckling problem ....................................................................... 18
         3.5.2.1 Analytic Solution ................................................................................................ 18
         3.5.2.2 Weighted Method Solution ................................................................................. 22

4. FINITE ELEMENT METHOD ...................................................................................................... 24
   4.1 Introduction ....................................................................................................................... 24
   4.2 How the finite element method Works ............................................................................... 25
   4.3 Mechanical variables and basic equations ........................................................................ 28
   4.4 Equilibrium conditions of geometrically nonlinear structures ........................................... 29
   4.5 Stability Analysis ................................................................................................................ 35
5. FINITE ELEMENT MODELLING BY ANSYS ................................................................................. 37
  5.1 Introduction....................................................................................................................... 37
  5.2 Buckling with ANSYS ........................................................................................................ 38
    5.2.1 Eigenvalue buckling analysis ...................................................................................... 38
    5.2.2 Nonlinear buckling analysis ........................................................................................ 40
  5.3 ANSYS Elements ................................................................................................................ 41
    5.3.1 SOLID 45 ..................................................................................................................... 41
    5.3.2 SHELL 63 ..................................................................................................................... 42

6. SIMULATIONS WITH ANSYS ..................................................................................................... 44
  6.1 Previous verifications ........................................................................................................ 44
  6.2 Analytic Formula and ANSYS .......................................................................................... 50
  6.3 Comparison between the results from Solid45 and Shell63 ............................................. 54
  6.4 Variation of the dimensions and effects in the critical load ............................................. 58
    6.4.1 Variation of the Thickness .......................................................................................... 58
    6.4.2 Variation of the Radius ............................................................................................... 59
    6.4.3 Variation of the length ............................................................................................... 60
  6.5 Buckling of cylindrical shells with gradual variation of thickness .................................... 61
  6.6 Buckling of cylindrical shells with curve section .............................................................. 69

7. CONCLUSIONS ....................................................................................................................... 75

8. BIBLIOGRAPHY ..................................................................................................................... 76
1. INTRODUCTION

The object of this project is to analyze the buckling strength of thin cylindrical shells subjected to radial compressive loads.

1.1. Motivation

It is well known that thin cylindrical shell structures have wide applications as one of the important structural elements in many engineering fields. Moreover, its load carrying capacity is decided by its buckling strength which depends strongly on geometrical imperfections presented in it.

Thin cylindrical shell structures are in general highly efficient structures and they have wide applications in the field of mechanical, civil, aerospace, marine, power plants, petrochemical industries etc. The thin cylindrical shell structures are prone to a large number of imperfections, due to their manufacturing difficulties. These imperfections affect the load carrying capacity of these shells. Reliable prediction of buckling strength of these structures is important, because the buckling failure is catastrophic in nature. That is the reason why the stability tests are becoming increasingly and more and more engineers are being asked for the necessary skills to perform calculations in a more accurate way.

1.2. Task and procedure

The theme of this thesis is:

Finite Element Method studies on the stability behavior of external radial loaded cylindrical shells with varying dimensions.

The present work deals with the stability analysis of thin-walled cylindrical shells, with different combinations of thicknesses, lengths, radius, pressures and sections. It examines how
1. INTRODUCTION

the shell is deformed by the application of external load with different geometric conditions.

The application of thin-walled cylindrical shells, as the essential structural members, has been widely studied by engineers, due to their importance in modern industries. These structures are prone to fail by buckling under external pressure, which could happen during discharging or wind load. Buckling phenomena occur when most of the strain energy, which is stored as membrane energy, can be converted to bending energy requiring large deformation resulting in catastrophic failure. Although the buckling capacity of shells depends principally on two geometric ratios of "length to radius" (L/R) and "radius to thickness" (R/t), but the effect of thickness variation on the behavior of the shells is complicated to be studied. On the other hand, the buckling strength of thin cylindrical shells is sensitive to the magnitude and shape of geometric imperfections.

For the analysis of the different models the finite element program ANSYS will be used.

1.3. Literature review

Because of the tremendous and continuous interest in shell buckling and the multitude of reported theoretical and experimental investigations, reviews and surveys have appeared in the open literature since the 1950s.

The first theoretical investigations on the subject dealt with axially loaded configurations, and they were performed by Lorenz, Timoshenko and Southwell. The first experimental studies are those of Lilly, Robertson, Flugge, Lundquist and Donnell. The initial theoretical investigations were based on many simplifying assumptions, and they reduced the mathematical model to a linear Eigen-boundary-value problem (classical bifurcation approach).

Comparison between theoretical predictions (critical loads) and experimental results (buckling loads) revealed discrepancy of unacceptable magnitude. A tremendous effort was made in order to explain the discrepancy both analytically and experimentally.

From the analytical point of view, the initial simplifying assumptions were later reevaluated and removed. This led to studies which attempted to attribute the discrepancy to different factors such as the effects of pre-buckling deformations, in-plane boundary conditions, or
initial geometric imperfections.

Initially, the imperfection sensitivity of the system was established through strict post-buckling analyses of the perfect configuration. In addition, some of these investigators explained that the minimum post-buckling equilibrium load is a measure of the load carrying capacity of the system. This thinking came to an end when Hoff, Madsen, and Mayers concluded from their calculations that the minimum post-buckling load tends towards zero with increasing number of terms in the series expansion and with diminishing thickness.

Another approach for imperfection sensitivity studies is to deal directly with the imperfect configuration and employ nonlinear kinematic relations. The first attempt is Donnell's. Koiter was the first to question the use of the minimal post-buckling load as a measure of the load carrying capacity. He also dealt directly with the imperfect configuration. His theory is limited to the neighborhood of the classical bifurcation load (immediate post-buckling), and therefore to small initial imperfections. Many researchers adopted this approach. The single and most important conclusion of all the theoretical investigations of cylindrical shells is that the primary reason for the discrepancy between (linear) theoretical critical loads and buckling loads is that the system is extremely sensitive to initial geometric imperfections.

In parallel to the above analytical investigations, many experimental studies were performed with the same objective in mind (explain the discrepancy). While the old buckling loads fell in the range of 15-50% of the classical critical load, the new ones, with use of carefully manufactured specimens, fall in the range of 40-90% of the classical critical load. Moreover, Thielemann and Esslinger extended their research to include theoretical post-buckled state calculations on the basis of observed experimental results.

A similar development was followed for the case of buckling under lateral loading (pressure). The first analysis is attributed to von Mises. Several investigations based on linear analyses appear in the literature. Batdorf used simplified Donnell-type of shell equations to predict critical loads. Soong employed Sanders' shell theory. Simitses and Aswani compared critical loads for the entire range of radius to thickness and length to radius ratios and for various load behaviors during the buckling process (true pressure, constant directional pressure, and centrally directed pressure) for a thin cylindrical shell, employing several linear shell theories: Koiter-Budiansky, Sanders, Flugge, and Donnell. Sobe studied the effect of boundary conditions on the critical pressure.
Post-buckling and imperfection sensitivity analyses appear in the literature, as in the case of axial compression. These studies follow the same pattern and include strictly post-buckling analyses, and Koiter-type of analyses. Orthotropic, stiffened, and other constructions were considered by several researchers, including Becker and Gerard, Meek, Hutchinson and Amazigo, Sheinman, and Giri.

Buckling analyses for torsion started with the work of Donnell. Several studies followed with the emphasis on different considerations. Hayashi dealt with orthotropic cylinders, and Lundquist and Nash are two of several that reported experimental results. Hayashi Hirano and Budiansky are among those who reported on post-buckling analyses.
2. DEFORMATIONS AND STRESSES OF CYLINDERS

2.1 Introduction

Depending on the type of material, size, geometry of an object and forces applied, various types of deformation may result:

1. **Elastic Deformation**

   This type of deformation is reversible. Once the forces are no longer applied, the object returns to its original shape. Linear elastic deformation is governed by Hooke’s law, which states:

   \[ \sigma = E \varepsilon \]

   where \( \sigma \) is the stress, \( \varepsilon \) indicates the resulting strain and \( E \) is a material constant called Young’s modulus. This relationship only applies in the elastic range and indicates that the slope of the stress vs. strain curve can be used to find Young’s modulus. The elastic range ends when the material reaches its yield strength. At this point plastic deformation begins.

2. **Plastic Deformation**

   This type of deformation is irreversible. Two different phases can occurred during the plastic deformation:

   - Strain hardening: The material becomes stronger through the movement of atomic dislocations.
   - Necking: It is indicated by a reduction in cross-sectional area of the structure. Necking begins after the ultimate strength is reached. During necking, the material can no longer withstand the maximum stress and the strain rapidly increases. Plastic deformation ends with the fracture of the material.
That can be checked in the strain-stress graphic, which provide a huge amount of information.

Figure 1: Stress vs. Strain

2.2 Thin-walled Cylinders

2.2.1 Stresses

The stresses in thin-walled cylinders have been widely studied, due to the huge amount of applications that they have, such as boiler shells, pressure tanks, pipes...

Generally three types of stresses are developed in pressure shells:

- Circumferential or hoop stresses ($\sigma_\theta$)
- Longitudinal or axial stresses ($\sigma_L$)
- Radial stresses ($\sigma_r$)
2. DEFORMATIONS AND STRESSES OF CYLINDERS

1. **Hoop stresses**

   It is a type of mechanical stress of a cylindrically shaped part as a result of internal or external pressure. It can be defined as the average force exerted circumferentially (perpendicular both to the axis and to the radius) on every particle in the cylinder wall.

   \[
   F = 2 \int_0^{\pi/2} prL \cos \theta \, d\theta = 2prL = pD_1L
   \]

So long as the wall thickness is small compared to the diameter then the force trying to split the cylinder due to the pressure is
The cross-section area which sustains this force is given by

\[ A = 2tL \]

Therefore the hoop stress is defined by

\[ \sigma_\theta = \frac{F}{A} = \frac{pD_i L}{2tL} = \frac{pD_i}{2t} \]

An alternative to hoop stress in describing circumferential stresses is wall stress or wall tension \((T)\), which is usually defined as the total circumferential force exerted along the entire radial thickness.

\[ T = \frac{F}{L} \]

The classic example of this stress is the tension applied to the iron bands of a wooden barrel.

2. **Axial stresses**

They are the stresses that occur in the longitudinal direction.

![Figure 4: Axial Stress](image)

In this case we consider the forces trying to split the cylinder along the length. The force due to the pressure is
2. DEFORMATIONS AND STRESSES OF CYLINDERS

\[ F = \int_0^r 2\pi r \, dr = 2\pi \frac{r^2}{2} = \pi \frac{D_i^2}{4} \]

The cross-section area which sustains the force in this case is given by:

\[ A = \pi D_i t \]

This area has been approximated to a rectangle whose dimensions are the length of the circumference \((\pi D_i)\) and the thickness.

Consequently the axial stress is defined by:

\[ \sigma_L = \frac{F}{A} = \frac{p \frac{\pi D_i^2}{4}}{\pi D_i t} = \frac{p D_i}{4t} \]

3. Radial stresses

The radial stresses are normal to the curved plane of the isolated element.

In thin-walled cylinder theory, they are normally not considered, because they are negligibly small compared to the other two stresses.

2.2.2 Thin-walled Cylinder Displacement

Consider a small rectangular area which is part of the wall in a thin walled cylinder.

![Figure 5: Thin-walled cylinder wall](image)
There are two stresses perpendicular to each other, the axial ($\sigma_L$) and circumferential ($\sigma_c$) stresses, as explained before.

From basic stress and strain theory the corresponding axial strain is:

$$\varepsilon_L = \frac{1}{E} (\sigma_L - \mu \sigma_c)$$

Substituting both stresses with the formulas calculated in the last chapter, the following formula for the longitudinal strain is obtained:

$$\varepsilon_L = \frac{\Delta L}{L} = \frac{1}{E} \left( \frac{pD}{4t} - \frac{\mu pD}{2t} \right) = \frac{pD}{4tE} (1 - 2\mu)$$

The circumferential strain may be defined as follows.

$$\varepsilon_c = \frac{\text{change in circumference}}{\text{original circumference}}$$

$$\varepsilon_c = \frac{\pi(D + \Delta D) - \pi D}{\pi D} = \frac{\Delta D}{D}$$

The conclusion obtained is that the circumferential stress is the same as the strain based on diameter, in other words, the diametric strain.

From basic stress and strain theory, the corresponding circumferential strain is:

$$\varepsilon_c = \frac{1}{E} (\sigma_c - \mu \sigma_L)$$

Substituting both stresses with the formulas calculated in the last chapter, the following formula for the circumferential strain is obtained:

$$\varepsilon_c = \frac{\Delta D}{D} = \frac{1}{E} \left( \frac{pD}{2t} - \frac{\mu pD}{4t} \right) = \frac{pD}{4tE} (2 - \mu)$$

For defining the volumetric strain, it is necessary to take into account the following parameters:
2. DEFORMATIONS AND STRESSES OF CYLINDERS

- Original Length = \( L_1 \)
- Original cross sectional area = \( \frac{\pi D^2}{4} \)
- Original Volume = \( \frac{\pi D^2}{4} L_1 \)
- New Length = \( L_2 = L_1 + \Delta L \)
- New cross sectional area = \( \frac{\pi(D+\Delta D)^2}{4} \)
- New Volume = \( \left( \frac{\pi(D+\Delta D)^2}{4} \right) (L_1 + \Delta L) \)

Then, the volumetric strain can be written as

\[
\varepsilon_v = \frac{\Delta V}{V} = \frac{\left( \frac{\pi(D+\Delta D)^2}{4} \right) (L_1 + \Delta L) - \left( \frac{\pi D^2}{4} \right) L_1}{\left( \frac{\pi D^2}{4} \right) L_1}
\]

Dividing out, clearing brackets and ignoring the product of two small terms, this reduces to

\[
\varepsilon_v = \frac{\Delta L}{L_1} + 2 \frac{\Delta D}{D} = \varepsilon_L + 2\varepsilon_c
\]

Substituting the equations for the axial and circumferential stresses into this, leads to

\[
\varepsilon_v = \frac{pD}{4\ell E} (5 - 4\mu)
\]
3. BUCKLING

3.1 Introduction

Buckling is a process by which a structure cannot withstand loads with its original shape, so that it changes its shape in order to find a new equilibrium configuration. It is generally resulting from structural instability due to a compressive action on the element involved. The effects are basically geometric, like large displacements or even plasticity in the walls of the structure.

Different stability theories have been formulated in order to determine the conditions under which a structural system, which is in equilibrium, ceases to be stable. The load for which a structure ceases to be stable and starts to buckle is known as the “Critical Buckling Load” ($P_{cr}$).

![Load deflection diagram](image)

*Figure 6: Load deflection diagram*

It is a so relevant value in order to analyze the behavior of a structure. Three different types of equilibriums can be reached:

1. **Stable equilibrium**: The pressure applied doesn’t reach the critical load, so that the structure returns to its original equilibrium state when the force is removed.
3. **BUCKLING**

![Stable equilibrium diagram](image1)

**Figure 7: Stable equilibrium**

2. **Neutral or indifferent equilibrium:** The load reached exactly the critical point and the elastic restoring force is not sufficient to return the structure to its initial position.

![Neutral equilibrium diagram](image2)

**Figure 8: Neutral equilibrium**

3. **Unstable equilibrium:** If the system is subjected to a compressive load that exceeds the critical buckling pressure, the column will bend considerably. Depending on the magnitude of the load, the structure either will remain in the bent position or will completely collapse and fracture.

![Unstable equilibrium diagram](image3)

**Figure 9: Unstable equilibrium**
3.2 Types of Buckling

The way in which buckling occurs depends on how the structure is loaded and on its geometrical and material properties. The pre-buckling process is often non-linear if there is a reasonably large percentage of bending energy being stored in the structure throughout the loading history.

According to the percentage of bending energy, the two basic ways in which a conservative system may lose its stability are: nonlinear or limit point buckling and bifurcation buckling.

3.2.1 Nonlinear collapse

In this way of buckling, the systems initially deform slowly and the stiffness of the structure or the slope of the load-deflection curve, decreases with increasing load. At the critical buckling point, the load-deflection curve has zero slope, and if the load is maintained, failure of the structure is usually dramatic and instantaneous.

![Figure 10: Load deflection diagram: Nonlinear buckling](image)

3.2.2 Bifurcation Buckling

It refers to a different kind of failure. At the bifurcation point (critical load is reached), where the primary and secondary paths intersect, more than one equilibrium position is possible. The primary path is not usually followed after loading exceeds this point and the structure is in the
post-buckling state. The slope of the secondary path at the bifurcation point determines the nature of the post-buckling. Positive slope indicates that the structure will have post buckling strength, whilst a negative slope means that the structure will simply collapse.

Figure 11: Load deflection diagram: Bifurcation buckling

3.3 Post-buckling of perfect and imperfect systems

After the bifurcation point, three main situations can happen depending on the type of system under study.

1. **Stable-symmetric point of bifurcation**: The buckling is characterized by a quick growth of the deflections after the critical point of the perfect system is reached.

After the bifurcation, the load can be increased, that means the structure has a stable behavior after buckling. It is a common case of post-buckling in columns, beams and plates.

Figure 11: Stable-symmetric point of bifurcation
2. **Unstable-symmetric point of bifurcation:** After reaching the bifurcation load the structure collapses immediately. Imperfections play a more important role than in the case before. Small imperfections can have a huge influence reducing the critical load.

![Figure 12: Unstable-symmetric point of bifurcation](image)

3. **Asymmetric point of bifurcation:** Depending on the direction of the imperfection, a symmetric stable or an asymmetric unstable behavior results. It is the extreme case, because the system loses its stability at a limit point (quite reduced comparing to the critical point) for small positive values of the imperfection.

![Figure 13: Asymmetric point of bifurcation](image)

### 3.4 Buckling of thin cylindrical shells

As commented before, thin-walled shells have many applications in engineering such as underground pipes and outer shells of submarines. When designing those structures one should consider not only the strength problem in service, but also the buckling problem. They are prone to a large number of imperfections, due to their manufacturing difficulties.
These imperfections affect the load carrying capacity of these shells. The imperfections that affect the strength of thin cylindrical shells are classified into three main groups:

2. Structural: Small holes, residual stresses and material inhomogenities.
3. Loading imperfections: Imperfect boundary conditions, non-uniform edge load distribution, load eccentricities.

The differential equation of the classical buckling theory of a thin-walled shell is written as

\[
\frac{D}{h} \nabla^4 w + E k_x \frac{\partial^4 w}{\partial y^4} + 2 E k_x k_y \frac{\partial^4 w}{\partial x^2 \partial y^2} + E k_y \frac{\partial^4 w}{\partial x^4} - \sigma_x^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x^2} \right) - 2 \sigma_{xy}^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x \partial y} \right) - \sigma_y^{(0)} = 0
\]

where \( \sigma_x^{(0)} , \sigma_{xy}^{(0)} , \sigma_y^{(0)} \) are the initial membrane stresses; \( w \) is buckling deflection in the \( z \) direction; \( E \) is Young's modulus; \( \mu \) is the Poisson's ratio; \( h \) and \( L \) are, respectively thickness and length of the shell; \( D = \frac{E h^3}{12(1-\mu^2)} \) is the bending stiffness; \( k_x , k_y \) curvature in \( x , y \) direction.

Figure 14: Structure of a thin-walled shell and its coordinate system
3.5 Buckling of thin-walled cylinders under external pressure

3.5.1 Introduction

The stability of circular cylindrical shells under uniform lateral pressure has been widely investigated. The behavior of cylindrical shells under external pressure is quite sensitive to geometric imperfections. There have been many theoretical studies investigating the strength of cylinders with specific imperfection forms, and it is well established that axisymmetric imperfections cause the greatest reductions in the buckling strength.

When thin shells are subjected to external pressure, the collapse is initiated by yielding, which is often the dominant factor, while the interaction with the instability is meaningful. In fact, the presence of imperfections reduces the load bearing capacity, so the classical elastic solution appears to be not adequate.

The major factors that affect the collapse pressure of thin-walled cylinders are the diameter-to-thickness ratio $D/t$, the Young’s modulus and yield stress of the material in the circumferential direction, and initial imperfections in the form of ovality and wall thickness variations.

3.5.2 Ways of solving the buckling problem

3.5.2.1 Analytic Solution

There are a lot of different analytic solutions for thin-walled cylinders under external pressure, depending on the characteristics of the cylinder and the constraints applied.

Considering a single-wave buckling mode, Glock calculated a solution of the buckling problem of constrained elastic cylinders. Glock used energy formulation in order to obtain the formula of the critical buckling load.
According to the Glock’s approach, the deflection is described by the following equation:

\[ w = w_1 \sin^2 \left( \frac{\pi \theta}{2\theta} \right) \]

where \( w = \) deflection of the buckled area

\( w_1 = \) deflection amplitude in the buckling area

\( \theta_v = \) variable of angle

\( \theta = \) angle of the buckling area

As the cylinder buckles, the potential energy includes three parts:

1. Flexural moment \( M \) within the buckling region.
2. Hoop compressive force \( N \)
3. External uniform pressure \( P_{cr} \) accumulated during the process.

Combining these parts, the following expression can be obtained:

\[ \Pi = \frac{EI}{2r^3} \int_0^{\theta_v} (w + \ddot{w})^2 \, d\theta + \int_0^{\pi} \frac{N^2 r}{2EI} \, d\theta - \int_0^{\theta_v} (w + \ddot{w}) \int_0^{\theta_v} P_{cr} \, wr \, d\theta \]

where \( \Pi = \) potential energy
$EI =$ flexural stiffness of the thin cylinder

$EF =$ tensile stiffness of the thin ring

$N =$ hoop compressive force

After substituting and integrating the equation above, the potential energy may be written as

$$\Pi = \frac{1}{16} \frac{EI}{r^3} \left( \frac{\pi}{\theta} \right)^4 w_1^4 \theta + \frac{N^2 r \pi}{2 EF} - \frac{P_{cr} r}{2} w_1 \theta$$

The minimum potential energy criterion should satisfy the following requirement:

$$d\Pi = \frac{\delta \Pi}{\delta w_1} dw_1 + \frac{\delta \Pi}{\delta \theta} d\theta = 0$$

which leads to the following differential equations:

$$\frac{\delta \Pi}{\delta w_1} = \frac{1}{8} \frac{EI}{r^3} \left( \frac{\pi}{\theta} \right)^4 w_1 + N \frac{\delta N}{\delta w_1} \frac{r \pi}{EF} - \frac{P_{cr} r}{2} \theta = 0$$

$$\frac{\delta \Pi}{\delta \theta} = - \frac{3}{16} \frac{EI}{r^3} \left( \frac{\pi}{\theta} \right)^4 w_1^2 + N \frac{\delta N}{\delta \theta} \frac{r \pi}{EF} - \frac{P_{cr} r}{2} r w_1 = 0$$

Solving both equations simultaneously, the following equation for the critical external pressure is reached:

$$\left( \frac{P_{cr} r^3}{EI} \right)_{cr} = 0.969 \left( \frac{r^2 EF}{EI} \right)^{\frac{5}{2}}$$

That equation is the result of Glock’s approach, but if the assumption is made that the flexural modulus of elasticity is approximately equal to the tensile modulus and considering the condition of plane-strain, the equation can be simplified as:

$$P_{cr} = \frac{E}{1-\mu^2} \left( \frac{t}{D} \right)^{2.2}$$

where $E$ is the Young’s modulus, $\nu$ the Poisson’s ratio, $D$ the cylinder diameter, and $t$ the wall thickness.
Moreover, considering a single-lobe buckling mode for long (free of boundary conditions) perfectly round elastic cylinders, the value of the critical pressure is given by the following formula:

\[ P_{cr} = \frac{E}{1-\mu^2} \left( \frac{t}{D} \right)^3 \]

This applies for cylinder lengths that fit:

\[ L > \frac{4\pi \sqrt{6}}{27} (1 - \mu^2)^{0.25} \frac{d}{t} \]

If a cylinder does not fall into that “long” category above the last equation is not suitable. There are two equations for short cylinders, of which the Von Mises is considered the better.

The first one is the Southwell equation. It only takes radial pressure into account (not axial) and the critical buckling pressure is

\[ p_c = \frac{1}{3} (n^2 - 1) \frac{2E}{1-\mu^2} \left( \frac{t}{d} \right)^3 + \frac{2E \frac{t}{d}}{(n^2 - 1)n^4 \left( \frac{2L}{\pi d} \right)^4} \]

where \( n \) in the number of waves in circumferential direction at collapse.

The second equation is called Von Mises equation. It may be used for cylinders subject to combined radial and axial pressure, or just radial. The critical pressure is

\[ p_c = \frac{1}{n^2 - 1 + \left( \frac{\pi d}{8L} \right)^2} \left[ \left( n^2 + \left( \frac{\pi d}{2L} \right)^2 \right)^2 - 2k_1 n^2 + k_2 \right] \times \frac{1}{3} \frac{2E}{1-\mu^2} \left( \frac{t}{d} \right)^3 + \frac{2Et \frac{t}{d}}{\left( \frac{2nL}{\pi d} \right)^2 + 1} \]

where

\[ k_1 = (1 + (1 + \mu)\rho)(1 + (1 - \mu)\rho) \]
3. BUCKLING

\[ k_z = [1 - \rho \mu] \left[ 1 + \rho (1 + 2\mu) - \mu^2 (1 - \mu^2) \left( 1 + \frac{1 + \mu}{1 - \mu} \rho \right) \right] \]

and

\[ \rho = \frac{1}{\left( \frac{2nL}{\pi a} \right)^2 + 1} \]

For both Von Mises and Southwell equations, \( n \) is the number of buckling waves, \( t \) the wall thickness, \( d \) the diameter of the cylinder, \( \mu \) the Poisson’s ratio and \( L \) the length of the cylinder.

### 3.5.3.2 Weighted Method Solution

Theoretical method can only solve some relatively simple problems. However, for more complicated problems it leads to a very complicated form such as an exponential series solution or a Fourier series solution. The weighted method is a really useful tool for solving complicated buckling problems by making use of the solutions of special simple problems. To determine the weights, some special known results are applied.

As mentioned before, the differential equation of the classical buckling theory of a thin-walled shell is

\[
\frac{D}{h} \nabla^2 w + E k_x \frac{\partial^4 w}{\partial y^4} + 2 E k_x k_y \frac{\partial^4 w}{\partial x^2 \partial y^2} + 2 E k_y \frac{\partial^4 w}{\partial x^4} - \sigma_x^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x^2} \right) - 2 \sigma_{xy}^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x \partial y} \right) -
\]

\[-\sigma_y^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial y^2} \right) = 0 \quad (1)\]

where \( \sigma_x^{(0)}, \sigma_{xy}^{(0)}, \sigma_y^{(0)} \) are the initial membrane stresses; \( w \) is buckling deflection in the \( z \) direction; \( E \) is Young’s modulus; \( \mu \) is the Poisson’s ratio; \( h \) and \( L \) are, respectively thickness and length of the shell; \( D = \frac{E h^3}{12(1-\mu^2)} \) is the bending stiffness; \( k_x, k_y \), curvature in \( x, y \) direction.

In axisymmetric problems \( \sigma_{xy}^{(0)} = 0 \). Then, applying that to the equation 1, we have

\[
\frac{D}{h} \nabla^2 w + E k_x \frac{\partial^4 w}{\partial y^4} + 2 E k_x k_y \frac{\partial^4 w}{\partial x^2 \partial y^2} + 2 E k_x \frac{\partial^4 w}{\partial x^4} - \sigma_x^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x^2} \right) - \sigma_y^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial y^2} \right) = 0 \quad (2)\]
Because there are no odd partial derivatives, variables can be separated so that we can suppose a buckling deflection function that satisfies whose general form is

\[ w = f(x) \sin \frac{ny}{R} \]

where

\[ f(x) = \sum_{i=1}^{m} f_i(x) \]

Substituting the last two equations into the equation 2, a series of equations in x is obtained. Assuming that coefficients before the same powers of x are equal zero, there are \((m+1)\) equations for \(p_0, p_1, p_2, \ldots, p_m\).

Introducing weights \(\lambda_1, \lambda_2, \ldots, \lambda_m\), where, the corresponding solution can be written as

\[ p = \sum_{i=1}^{m} \lambda_i p_i \]

What should we do to determine the weights \(\lambda_0, \lambda_1, \lambda_2, \ldots, \lambda_m\)?

We can find \(\lambda_i\) substituting \(m\) groups of known results for \(p'_{cr}, n' (i=1, 2, \ldots, m)\) from the finite elements calculations and solving the group of equations. Finally, we can substitute \(n=1, 2, 3\ldots\) for different buckling modes into equation 1. The minimum is the desired critical load \(p_{cr}\).
4. FINITE ELEMENT METHOD

4.1 Introduction

The Finite Element Method (FEM) has become very important to solve engineering problems, allowing solving cases that until recent time were virtually impossible by traditional mathematical methods. This required to prototype, test them and make improvements iteratively, what brought high costs both financially and in time of development.

The FEM provides a mathematical model for calculating the real system, easier and cheaper to change than a prototype. However, it remains as an approximate calculating method due to the basic assumptions of the method. Therefore, prototypes are still necessary, but in lower quantity, since the first one can be well approximated to the optimum design.

The finite element method as mathematical formulation is relatively new; although its basic structure has been known for quite a long time, in recent years a great development has been achieved due to the advances in computer technology. These computer advances have allowed the user applying many programs that can perform finite element calculations. Furthermore, FEM allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements.

The general idea of the finite element method is to divide a continuum in a set of small elements interconnected by a range of points called nodes.

Figure 16: Finite Element Method
The equations governing the behaviour of the continuum also govern the element’s behaviour.

In any system subject to analysis, next parts can be distinguished:

- **Domain**: Geometric space where the system is analysed.
- **Boundary conditions**: Known variables that determine changes in the system: load, displacement, temperature, voltage, heat sources...
- **Unknown factor**: System variables to know after the boundary conditions have been applied on the system: displacements, stresses, temperatures...

### 4.2 How the finite element method works

In a continuum problem of any dimension the field variable (whether it is pressure, temperature, displacement, stress, or some other quantity) possesses infinitely many values because it is a function of each generic point in the body or solution region. Consequently, the problem is one with an infinite number of unknowns.

The finite element discretization procedures reduce the problem to one with a finite number of unknowns by dividing the solution region into elements and by expressing the unknown field variable in terms of assumed approximating functions within each element. The approximating functions (sometimes called interpolation functions) are defined in terms of the values of the field variables at specified points called nodes. In addition to boundary nodes, an element may also have a few interior nodes. The nodal values of the field variable and the interpolation functions for the elements define completely the behaviour of the field variable within the elements.

For the finite element representation of a problem, the nodal values of a field variable become the unknowns. Once these unknowns are found, the interpolation functions define the field variable throughout the assemblage of elements.

Clearly, the nature of the solution and the degree of approximation depend not only on the size and number of the elements used but also on the interpolation functions selected. As one would expect, we cannot choose functions arbitrarily, because certain compatibility conditions should be satisfied. Often functions are chosen so that the field variable or its derivatives are
continuous across adjoining element boundaries. These are applied to the formulation of different kinds of elements.

However, an important feature of the finite element method that sets it apart from other numerical methods has not been mentioned. This feature is the ability to formulate solutions for individual elements before putting them together to represent the entire problem. This means, for example, that if a problem in stress analysis is being treated, the force–displacement or stiffness characteristics of each individual element is found and then the elements are assembled to find the stiffness of the whole structure. In essence, a complex problem reduces to considering a series of greatly simplified problems.

Regardless of the approach used to find the element properties, the solution of a continuum problem by the finite element method always follows an orderly step-by-step process.

To summarize in general terms how the finite element method works, the steps of proceeding are briefly described.

A body of matter (solid, liquid, or gas) or simply a region of space in which a particular phenomenon is occurring is considered.

The steps to follow in order to perform a finite element method analysis are:

1. **Discretize the Continuum.** The first step is to divide the continuum or solution region into elements. In the example of Figure 17 the turbine blade has been divided into...
triangular elements that might be used to find the temperature distribution or stress distribution in the blade. A variety of element shapes may be used, and different element shapes may be employed in the same solution region. Indeed, when analysing an elastic structure that has different types of components such as plates and beams, it is not only desirable but also necessary to use different elements in the same solution. Although the number and the type of elements in a given problem are matters of engineering judgment, the analyst can rely on the experience of others for guidelines.

2. Select Interpolation Functions. The next step is to assign nodes to each element and then choose the interpolation functions to represent the variation of the field variable over the element. The field variable may be a scalar, a vector, or a higher-order tensor. Often, polynomials are selected as interpolation functions for the field variable because they are easy to integrate and differentiate. The degree of the polynomial chosen depends on the number of nodes assigned to the element, the nature and number of unknowns at each node, and certain continuity requirements imposed at the nodes and along the element boundaries. The magnitude of the field variable as well as the magnitude of its derivatives may be the unknowns at the nodes.

3. Find the Element Properties. Once the finite element model has been established, the matrix equations expressing the properties of the individual elements can be determined.

4. Assemble the Element Properties to Obtain the System Equations. To find the properties of the overall system modelled by the network of elements we must “assemble” all the element properties. In other words, we combine the matrix equations expressing the behaviour of the elements and form the matrix equations expressing the behaviour of the entire system. The matrix equations for the system have the same form as the equations for an individual element but they contain many more terms because they include all nodes.

The basis for the assembly procedure stems from the fact that at a node, where elements are interconnected, the value of the field variable is the same for each element sharing that node. A unique feature of the finite element method is that the system equations are generated by assembly of the individual element equations. In contrast, in the finite difference method the system equations are generated by
writing nodal equations.

5. **Impose the Boundary Conditions.** Before the system equations are ready for being solved, they must be modified to account for the boundary conditions of the problem. At this stage we impose known nodal values of the dependent variables or nodal loads.

6. **Solve the System Equations.** The assembly process gives a set of simultaneous equations that we solve to obtain the unknown nodal values of the problem. If the problem describes steady or equilibrium behaviour, then we must solve a set of linear or nonlinear algebraic equations. There are different standard solution techniques for solving these equations. If the problem is unsteady, the nodal unknowns are a function of time, and we must solve a set of linear or nonlinear ordinary differential equations.

7. **Make Additional Computations If Desired.** Many times we use the solution of the system equations to calculate other important parameters. For example, in a structural problem the nodal unknowns are displacement components. From these displacements we calculate element strains and stresses. Similarly, in a heat-conduction problem the nodal unknowns are temperatures, and from these we calculate element heat fluxes.

### 4.3 Mechanical variables and basic equations

An important advantage of the FEM is that its use is not restricted to certain structures. To achieve a uniform and clear presentation, an operator formulation is applied using matrix variables. The variables are divided into field and boundary variables.

Basic mechanical equations:

\[
\varepsilon = D_k \cdot u \quad \text{ (Kinematic law)}
\]

\[
\sigma = E \cdot \varepsilon + D \cdot \dot{\varepsilon} \quad \text{ (Material law)}
\]

\[
-(p + f) = D_p \cdot \sigma \quad \text{ (Dynamic equilibrium conditions)}
\]

Features in these three basic equations:
4. FINITE ELEMENT METHOD

- Vector of external force variables

- Vector of external displacement variables

- Vector of d'Alembert inertial forces

- Vector of internal force variables

- Vector of internal displacement variables (Distortion, stretching, curvature)

Along the boundaries the following variables are defined:

- Vector of boundary force variables

- Vector of boundary displacements variables.

In addition, the static and geometric constraints must be satisfied at the boundaries of the mechanical system.

\[ \mathbf{r} = R_f \cdot \mathbf{u} \quad \text{(Displacement boundary conditions)} \]  
\[ \mathbf{t} = R_f \cdot \mathbf{\sigma} \quad \text{(Force variables of boundary conditions)} \]  

4.4 Equilibrium conditions of geometrically nonlinear structures

For the formulation of the equilibrium of complex structures energy statements instead of differential equations are used. Such an energy principle is the principle of virtual displacements.

The principle of virtual displacements is a global equilibrium formulation. It is based on the
evidence that the sum of virtual work is zero.

\[ \delta W = \delta W_i + \delta W_a = 0 \]  
(The basic equation) \hspace{1cm} (7)

Thereafter, a deformed body is in balance if the internal and external virtual work due to a virtual displacement are identical:

\[ \delta W_i = \delta W_a \]  
(Equilibrium condition) \hspace{1cm} (8)

For the present thesis dealt with thin-walled shell structures, this principle is:

\[ \int_A \sigma^T \cdot \delta \varepsilon \cdot dA = \int_A p^T \cdot \delta u \cdot dA + \int_{S_r} t^T \cdot \delta r \cdot dS_r \]  
(9)

As part of geometric non-linear theories, large structural deformations are taken into account. The basic kinematic equation in this case reads as

\[ \varepsilon = D_K \cdot u = \left[ D_{KL} + \frac{1}{2} \cdot D_{KN} (u) \right] \cdot u \]  
(10)

The kinematic operator \( D_K \) can be split into a deformation-independent linear partial operator \( D_{KL} \) and a deformation-dependent, nonlinear partial operator \( D_{KN} (u) \).

The virtual strain \( \delta \varepsilon \) is obtained from Eq. (10) by variation.

\[ \delta \varepsilon = \frac{d \varepsilon}{du} \cdot \delta u = \left[ D_{KL} + D_{KN} (u) \right] \cdot \delta u \]  
(11)

The material law for physically linear material behavior is:

\[ \sigma = E \cdot \varepsilon = E \cdot D_K \cdot u \]  
(12)

The virtual boundary displacements \( \delta r \) are expressed with the boundary operator \( R_f \) by the field distribution displacements \( \delta u \):

\[ \delta r = R_f \cdot \delta u \]  
(13)

Setting equations (10) to (13) in the equilibrium requirement (9), yields:

\[ \int_A u^T \cdot \left[ D_{KL} + \frac{1}{2} \cdot D_{KN} (u) \right]^T \cdot E \cdot \left[ D_{KL} + D_{KN} (u) \right] \cdot \delta u \cdot dA = \]
4. FINITE ELEMENT METHOD

\[ \int_A p^T \cdot \delta u \cdot dA + \int_{S_t} t^T \cdot \delta u \cdot dS_t \]  \hspace{1cm} (14)

As apart of the finite element method, the deformations of the elements are approximated:

\[ u = \Omega \cdot \nu \]  \hspace{1cm} (15)

and

\[ \delta u = \Omega \cdot \delta \nu \]  \hspace{1cm} (16)

where \( \Omega \) denotes the matrix of shape function and \( \nu \) or \( \delta \nu \) represent the actual or virtual nodal displacements.

Inserting (15) and (16) into the equilibrium requirement (14), yields:

\[ \delta U^T \int_A \Omega^T \cdot \left[ D_{KL} + \frac{1}{2} D_{KN} (\Omega \cdot \nu) \right]^T \cdot E \cdot \left[ D_{KL} + D_{KN} (\Omega \cdot \nu) \right] \cdot \Omega \cdot \nu \cdot dA = \]

\[ = \delta U^T \int_A \Omega^T \cdot p \cdot dA + \delta U^T \int_{S_t} \Omega^T \cdot R_f^T \cdot t \cdot dS_t \]  \hspace{1cm} (17)

After the introduction of the linear and nonlinear strain displacement matrix:

\[ H_L = D_{KL} \cdot \Omega \]  \hspace{1cm} (18)

\[ H_N(\nu) = D_{KN} \cdot (\Omega \cdot \nu) \cdot \Omega \]  \hspace{1cm} (19)

and simplified division by \( \delta U^T \), the equation (17) can be written as:

\[ \int_A \left[ H_L + H_N(\nu) \right]^T \cdot E \cdot \left[ H_L + \frac{1}{2} H_N(\nu) \right]^T \cdot dA \cdot \nu = \]

\[ = \int_A \Omega^T \cdot p \cdot dA + \int_{S_t} \Omega^T \cdot R_f^T \cdot t \cdot dS_t \]  \hspace{1cm} (20)

This non-linear relationship for the element equilibrium cannot be solved and may require numerical solution methods. For this, the force and deformation variables are incremented, i.e. they are split into the fundamental state assumed to be known (-) and the increments (+).
4. FINITE ELEMENT METHOD

Figure 18: Fundamental state and increase parameters

IS – Initial state

FS - Fundamental increment

NS - Neighboring increment

In the figure below the mentioned states are demonstrated visually:

Figure 19: Visual demonstration of the states.
It is therefore:

\[ \nu = \nu^- + \nu^+; \quad p = p^- + p^+; \quad \tau = \tau^- + \tau^+; \quad (21) \]

Introducing the equation (21) into the equilibrium equation (20), yields:

\[
\int_A \left[ H_L + H_N(\nu^- + \nu^+) \right]^T \cdot E \cdot \left[ H_L + \frac{1}{2} H_N(\nu^- + \nu^+) \right] \cdot dA \cdot (\nu^- + \nu^+) = \\
= \int_A \Omega^T \cdot (p^- + p^+) \cdot dA + \int_{S_e} \Omega^T \cdot R \cdot (t^- + t^+) \cdot dS_t
\]

(22)

After multiplying out and sorting results, the following element matrices and vectors arise:

- Elastic stiffness matrix:

\[ k_e \cdot \nu^+ = \int_A H_L^T \cdot E \cdot H_L \cdot dA \cdot \nu^+ \]

- Linear initial stress matrix:

\[ k_{\sigma L} \cdot \nu^+ = \int_A H_N^T(\nu^+) \cdot E \cdot H_L \cdot \nu^- \cdot dA \]

- Nonlinear initial stress matrix:

\[ k_{\sigma N} \cdot \nu^+ = \int_A \frac{1}{2} H_N^T(\nu^+) \cdot E \cdot H_N \cdot (\nu^-) \cdot (\nu^-) dA \]

(23)

- Linear initial deformation matrix:

\[ k_{\mu L} \cdot \nu^+ = \int_A \left( H_L^T \cdot E \cdot H_N \cdot (\nu^-) + H_N^T(\nu^-) \cdot E \cdot H_L \right) \cdot dA \cdot \nu^+ \]

- Nonlinear initial deformation matrix:

\[ k_{\mu N} \cdot \nu^+ = \int_A \left( H_N^T \cdot (\nu^-) \cdot E \cdot H_N \cdot (\nu^-) \right) \cdot dA \cdot \nu^+ \]

- Vector of internal forces in the fundamental state:
\[ g \cdot v^+ = \int_A \left[ H_L + H_N \cdot (v^-) \right]^T \cdot E^\cdot \left[ H_L + \frac{1}{2} \cdot H_N (v^-) \right] \cdot v^- \cdot dA \]

- Element load vector in the fundamental state:
\[ p^- = \int_A \Omega^T \cdot p^- \cdot dA + \int_{S_t} \Omega^T \cdot R_f^T \cdot \xi^- \cdot dS_t \]

- Incremental load vector
\[ p^+ = \int_A \Omega^T \cdot p^+ \cdot dA + \int_{S_t} \Omega^T \cdot R_f^T \cdot \xi^+ \cdot dS_t \]

Adding up all the element-related matrices, we obtain the tangential element stiffness matrix
\[ k_T = k_e + k_{oL} + k_{\sigma N} + k_{uL} + k_{\mu N} \tag{24} \]

With the introduced matrices and vectors, the element equilibrium relationship can be converted into the following matrix representation:
\[ k_T \cdot v^+ = (k_e + k_{oL} + k_{\sigma N} + k_{uL} + k_{\mu N}) \cdot v^+ = p^- - g \cdot (v^-) \tag{25} \]

After the assembly of the finite elements, the system tangential stiffness relationship reads as:
\[ K_T \cdot V^+ = (K_e + K_{oL} + K_{\sigma N} + K_{uL} + K_{\mu N}) \cdot V^+ = P - G \cdot (V^-) \tag{26} \]

For the application of line-search algorithms, the load is incremented, i.e. divided into small load steps. Within this last step, equation (26) is solved iteratively until the right side is zero.

An algorithm that has proved is the Newton - Raphson – scheme, rebuilding the tangential stiffness matrix with each iteration step. (See figure 20)
In geometrically linear investigations the arising deformations $\vec{V}$ are by definition very small. The fundamental state can be connected to the output state ($\vec{V} = 0$). The deformation vector $\vec{V}^+ = \vec{V}$ then describes the sought displacement field. The only system matrix remaining independent of the fundamental state deformations is the elastic stiffness matrix, so that from equation (26) the following linear system of equations results:

$$ K_p \cdot \vec{V} = P $$  \hspace{1cm} (27)

The total stiffness matrix $K_p$ is symmetric and positive definite, so that efficient solution algorithms such as the Cholesky - procedures can be used.

### 4.5 Stability Analysis

In the implementation of stability testing is most sought after, whether next to a given equilibrium state (fundamental state $\vec{V}$) an adjacent equilibrium position (adjacent state $\vec{V}^+$) without changing the external load is possible. If the deformation vector $\vec{V}$ describes a balanced state of equilibrium, then the right side of the equation (26) is zero, because the external loads $P$ and the internal stresses $G(\vec{V})$ are in equilibrium.

The desired equilibrium position in the adjacent state (neighboring state $\vec{V}^+$) shall occur at
unmodified load. The equilibrium condition according to equation (26) reads as:

\[ K_T \cdot V^+ = 0 \]  

(28)

with the vector \( V^+ \neq 0 \).

Equation (28) describes a homogeneous system of equations. For non-trivial solution \( V^+ \) the determinant of the coefficient matrix \( K_T \) has to be zero. Adjacent equilibrium positions are therefore only possible if

\[ \det K_T = 0 \]  

(29)

This criterion can be examined by the incremental-iterative path-following after each step load balanced by an accompanying eigenvalue calculation. If one eigenvalue of \( K_T \) is equal to zero, so will be \( \det K_T = 0 \), there is an instability point.

As part of this thesis circular cylindrical shells under radial load. Therefore the fundamental state occurs in only radial deformation, leading to a so called linear prebuckling state. The initial deformation matrices \( K_{\mu L} \) and \( K_{\mu N} \) are zero in this case. Assuming further that the fundamental state deformations arising are small, so quadratic terms \( V \) are ignored. This means that the nonlinear initial stress matrix \( K_{\mu N} \) is negligibly small. Equation (28) is simplified thus:

\[ (K_T + K_{\mu L}) \cdot V^+ = 0 \]  

(30)

In the computer-based implementation of stability analysis, the structure with a reference load \( 0 \cdot P \) (e.g. \( q_0 = 1 \text{ N/mm}^2 \)) is loaded and determines the increase in load factor \( \lambda_{KR} \) in which an instability occurs. Considering this approach in the above equation (30), one gets the initial value problem of the so-called “classical stability”:

\[ (K_T + \lambda_{KR} \cdot K_{\mu L} (0 \cdot P)) \cdot V^+ = 0 \]  

(31)

From the solution of this eigenvalue problem, the critical load factors \( \lambda \) and the corresponding buckling modes \( V^+ \) are obtained. For the solution of the eigenvalue problem (see equation 31) the Lanczos algorithm was used in the present study.
5. FINITE ELEMENT MODELLING BY ANSYS

5.1 Introduction

ANSYS is a general purpose finite element modeling package for solving different mechanical problems numerically, such as static/dynamic structural analysis, heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS is divided into three steps when analyzing and solving a problem with the finite element method:

1. Preprocessing: It is basically defining the problem; the major steps in preprocessing are given below:
   - Define key-points, nodes, lines, areas, volumes.
   - Define element type and geometric properties.
   - Meshing.

2. Solution: the following steps should be specify before solving the problem:
   - Type of analysis: static, modal, buckling, ...
   - Loads: point load or pressure, direction of the load, ...
   - Constraints.

3. Post-processing: Viewing of the results, such as:
   - Lists of nodal displacements and displacement plots.
   - Element forces and moments.
   - Stress contour diagrams.
5.2 Buckling with ANSYS

Two techniques are available in ANSYS for predicting the buckling load and buckling mode of a structure:

- Nonlinear buckling analysis
- Eigenvalue buckling analysis.

5.2.1 Eigenvalue buckling analysis

Eigenvalue buckling analysis predicts the theoretical buckling strength of an ideal elastic structure. This analysis is used to predict the bifurcation point using a linearized model of elastic structure. It computes the structural eigenvalues for the given system loading and constraints. This is known as classical Euler buckling analysis.

However, in real-life, structural imperfections and nonlinearities prevent most real-world structures from reaching their eigenvalue predicted buckling strength. That means that this method over-predicts the expected buckling loads. That is the reason why it is not recommended for accurate, real-world buckling prediction analysis.

![Figure 21: End Shortening vs Axial load](image-url)
The basic form of the eigenvalue buckling analysis is given by:

\[
[K_r]{\{\phi_i\}} = \lambda_i[K_{st}]{\{\phi_i\}}
\]

where

- \([K_r]\) = Elastic stiffness matrix
- \({\{\phi_i\}} = Eigen\ vector
- \(\lambda_i\) = Eigen\ value\ for\ buckling\ mode
- \([K_{st}]\) = Initial stress matrix. This matrix includes the effects of membrane loads on the stiffness of the structure. It is also assembled based on the results of a previous linear static analysis.

The eigenvalue solution uses an iterative algorithm that obtains at first the eigenvalues and secondly the displacements that define the corresponding mode shape.

The eigenvalue represents the ratio between the applied load and the buckling load. This can be expressed as follows:

\[
\lambda_i = \frac{Buckling\ Load}{Applied\ Load}
\]

That’s why it is often said that the eigenvalue is like a safety factor for the structure against buckling. On one hand, an eigenvalue less than 1.0 indicates that a structure has buckled under the applied loads. On the other hand, an eigenvalue greater than 1.0 indicates that the structure will not buckle.

Another important point to note about this formulation is that only the membrane component of the loads in the structure is used to determine the buckling load. This means that the effect of prebuckling rotations due to moments is ignored.
5.2.2 Nonlinear buckling analysis

It is more accurate than eigenvalue analysis because it employs non-linear, large-deflection, static analysis to predict buckling loads. Its mode of operation is very simple: the applied load is gradually increased until a load level is found whereby the structure becomes unstable. Normally each of these load increments will converge in a small number of iterations.

It is also common that suddenly a very small increase in the load cause very large deflections.

The nonlinear solver is ideally suited for modeling structures that do not collapse while buckling. In the nonlinear analysis the stiffness matrix is updated periodically (for every iteration of every load increment) based on the current deformed shape of the structure. This is important from a buckling point of view since the effect of the pre- and postbuckling deformations are included in the analysis. When we talk about 'pre-buckling deformations' we are generally referring to those deflections caused by the moments in the structure prior to buckling. 'Post buckling deformations' refers to those deflections that result from some initial buckling failure of the structure.

It also permits the modeling of geometric imperfections, load perturbations and gaps. Another important point that should be made about the nonlinear buckling analysis is that material nonlinearities (yielding) can be considered in addition to the geometric effects.

Figure 22: Buckling
5.3 ANSYS Elements

For realizing the simulations of our project, two different elements from ANSYS library will be used:

- **SOLID 45**
- **SHELL 63**

### 5.3.1 SOLID 45

This element is used for three-dimensional modeling of solid structures. The element is defined by eight nodes, having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

Pressures may be input as surface loads on the element faces and positive pressures act into the element. Temperatures and fluencies may be input as element body loads at the nodes.

The geometry, node locations, and the coordinate system for this element are shown in the following figure:
5.3.2 SHELL 63

It has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes.

Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection analyses.

The geometry, node locations, and the coordinate system for this element are shown in the following figure:

![Figure 24: SHELL63](image)

The element is defined by four nodes, four thicknesses, an elastic foundation stiffness, and the orthotropic material properties and the thickness is assumed to vary smoothly over the area of the element.

Pressures may be input as surface loads on the element faces and positive pressures act into the element. Edge pressures are input as force per unit length. The lateral pressure loading may be an equivalent element load applied at the nodes or distributed over the face of the element.
The equivalent element load produces more accurate stress results with flat elements representing a curved surface or elements supported on an elastic foundation since certain fictitious bending stresses are eliminated.
6. SIMULATIONS WITH ANSYS

6.1 Previous verifications

The aim of this part of the project is to find out if the basic model and the program created with ANSYS fit with some already calculated results. In order to make these first verifications, we will use an article of the online magazine “Science Direct”, and we will simulate with ANSYS the cylinder model of that article. Finally the results provided by ANSYS will be compared with the ones obtained with the previously explained “weighted method” of that article.

The differential equation of the classical buckling theory of a thin-walled shell is written as

\[
\frac{D}{h} \nabla^8 W + E k_2 \frac{\partial^4 w}{\partial x^4} + 2E k_y k_2 \frac{\partial^4 w}{\partial x^2 \partial y^2} + E k_2 \frac{\partial^4 w}{\partial y^4} - \sigma_x^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x^2} \right) - 2\sigma_{xy}^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial x \partial y} \right) - \\
-\sigma_y^{(0)} \nabla^4 \left( \frac{\partial^2 w}{\partial y^2} \right) = 0
\]

We will consider the following problems. A cylindrical shell, clamped at two ends, is subjected to homogeneous radial external pressure \( p \). If the shell is in a momentless state before buckling and there is only a hoop stress \( \sigma_y^{(0)} = -pR/h \). The buckling equation can be simplified to

\[
\frac{D}{h} \nabla^8 W + \frac{E}{R^2} \frac{\partial^4 w}{\partial x^4} + \frac{pR}{h} \nabla^4 \left( \frac{\partial^2 w}{\partial y^2} \right) = 0
\]

The clamped end conditions are:

\[
x=0, \quad L: \quad w=0, \quad \frac{\partial w}{\partial x}=0
\]

The buckling deflection function which satisfies the given end condition can be

\[
w = A \left( \cos \frac{2\pi x}{L} - 1 \right) \sin \frac{n y}{R}
\]
where \( L \) expresses the length of the shell and \( n \) the buckling wave number of the eigen mode.

Substituting the last equation into the equation 2 and assuming the constant term and the coefficients of the term before \( \cos \left( \frac{2\pi x}{L} \right) \) equal zero, the following values are obtained:

\[
p_0 = \frac{Dn^2}{R^3}, \quad p_1 = \left[ \frac{(1 + \theta^2)^2\eta}{12(1 - \mu^2)} + \frac{\theta^4}{(1 + \theta^2)^2\eta} \right] \frac{Eh^2}{R^2}
\]

Introducing a weight \( \lambda \), the solution of the problem can be estimated by

\[
p = p_0 + (1 - \lambda)p_1
\]

Finally we will use a group of results previously calculated with the finite element method, in order to obtain the value of \( \lambda \).

The problem that is going to be solved has the following dimensions:

- \( R=68\text{mm} \)
- \( h=2.7\text{mm} \)
- \( E=70\text{GPa} \)
- \( \mu=0.33 \)

On one hand, using the method explained above (the weighted method), the solutions showed
in the article are

\[
\begin{array}{|c|c|c|}
\hline
\frac{L}{R} & \text{Weighted solution } p_c \text{ (MPa)} & n \\
\hline
20 & 1.895 & 2.0 \\
10 & 4.038 & 3.0 \\
5 & 6.724 & 3.0 \\
2 & 17.00 & 5.0 \\
1 & 41.58 & 6.0 \\
\hline
\end{array}
\]

Figure 26: Article Solutions

On the other hand, after simulating the model in ANSYS we obtain the following values for the critical buckling pressure:

\[
\begin{itemize}
\item \frac{L}{R} = 20
\end{itemize}
\]

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.23098413</td>
</tr>
<tr>
<td>2</td>
<td>1.23098449</td>
</tr>
<tr>
<td>3</td>
<td>3.1686264</td>
</tr>
<tr>
<td>4</td>
<td>3.1686265</td>
</tr>
<tr>
<td>5</td>
<td>3.5279170</td>
</tr>
</tbody>
</table>

In the second buckling mode (n=2): \( p_c = 1.7309843 \) MPa
• \( \frac{L}{R} = 10 \)

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.42106772</td>
</tr>
<tr>
<td>2</td>
<td>3.42186788</td>
</tr>
<tr>
<td>3</td>
<td>4.0573255</td>
</tr>
<tr>
<td>4</td>
<td>4.0573256</td>
</tr>
<tr>
<td>5</td>
<td>5.0242090</td>
</tr>
</tbody>
</table>

In the third buckling mode (n=3): \( p_{cr} = 4.0573255 \) MPa

• \( \frac{L}{R} = 5 \)

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.7640991</td>
</tr>
<tr>
<td>2</td>
<td>5.7640993</td>
</tr>
<tr>
<td>3</td>
<td>6.5845219</td>
</tr>
<tr>
<td>4</td>
<td>6.8045221</td>
</tr>
<tr>
<td>5</td>
<td>9.7064074</td>
</tr>
</tbody>
</table>

In the third buckling mode (n=3): \( p_{cr} = 6.5845219 \) MPa

• \( \frac{L}{R} = 2 \)

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>14.293305</td>
</tr>
<tr>
<td>2</td>
<td>14.293305</td>
</tr>
<tr>
<td>3</td>
<td>16.141562</td>
</tr>
<tr>
<td>4</td>
<td>16.141562</td>
</tr>
<tr>
<td>5</td>
<td>16.732688</td>
</tr>
</tbody>
</table>

In the fifth buckling mode (n=5): \( p_{cr} = 16.732688 \) MPa

• \( \frac{L}{R} = 1 \)

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>33.399888</td>
</tr>
<tr>
<td>2</td>
<td>33.399888</td>
</tr>
<tr>
<td>3</td>
<td>33.399888</td>
</tr>
<tr>
<td>4</td>
<td>33.048001</td>
</tr>
<tr>
<td>5</td>
<td>37.130363</td>
</tr>
</tbody>
</table>

In the third buckling mode (n=3): \( p_{cr} = 4.0573255 \) MPa
### 6. Simulations with ANSYS

<table>
<thead>
<tr>
<th>Length/Radius</th>
<th>Buckling Mode(n)</th>
<th>Weighted Solution (Mpa)</th>
<th>ANSYS solution (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>2</td>
<td>1,895</td>
<td>1,7309847</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>4,038</td>
<td>4,0573255</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>6,724</td>
<td>6,5845219</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>17,000</td>
<td>16,732688</td>
</tr>
<tr>
<td>1</td>
<td>6</td>
<td>41,580</td>
<td>37,138363</td>
</tr>
</tbody>
</table>

Diagram 1: Critical Load as a function of the ratio Length/Radius

It can be concluded that our ANSYS model is totally correct, because it reflects with a high accuracy what is really happening and because the results obtained with both methods are almost identical (comparable). The chart and the diagram confirm this previous assertion.

At last, it is also necessary to check if the simulation of our model provides the right figures in comparison with the ones that appeared on the paper used. In the followings figures we can check that the same graphical results are also obtained.
6. SIMULATIONS WITH ANSYS

Figure 28: Article Simulation

Figure 29: ANSYS top view

Fig 30: ANSYS 3D view
6.2 Analytic Formula and ANSYS

In this section, we will compare the values of the critical buckling pressure provided by ANSYS with the analytical formula.

\[
p_{cr} = \frac{E}{1 - \mu^2} \left( \frac{t}{D} \right)^3
\]

It can be used when:

\[
L > \frac{4\sqrt{E}}{27} (1 - \mu^2)^{0.25} \frac{d}{\sqrt{t}} > 0 \quad \text{(Long cylinder)}
\]

In the first step it should be checked if the combinations of length, radius and thickness values for which we will calculate the critical buckling pressure satisfy the length condition above. This is reflected in the following chart:

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>Radius (mm)</th>
<th>Length (mm)</th>
<th>Length Condition (&gt;0)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>5000</td>
<td>10000</td>
<td>-2,39E+05</td>
</tr>
<tr>
<td>20</td>
<td>1000</td>
<td>10000</td>
<td>-1,23E+04</td>
</tr>
<tr>
<td>20</td>
<td>850</td>
<td>10000</td>
<td>-7,45E+03</td>
</tr>
<tr>
<td>20</td>
<td>700</td>
<td>10000</td>
<td>-3,04E+03</td>
</tr>
<tr>
<td>20</td>
<td>600</td>
<td>10000</td>
<td>-3,50E+02</td>
</tr>
<tr>
<td>20</td>
<td>520</td>
<td>10000</td>
<td>1,65E+03</td>
</tr>
<tr>
<td>20</td>
<td>500</td>
<td>10000</td>
<td>2,13E+03</td>
</tr>
<tr>
<td>20</td>
<td>480</td>
<td>10000</td>
<td>2,59E+03</td>
</tr>
<tr>
<td>20</td>
<td>400</td>
<td>10000</td>
<td>4,37E+03</td>
</tr>
<tr>
<td>20</td>
<td>250</td>
<td>10000</td>
<td>7,22E+03</td>
</tr>
<tr>
<td>15</td>
<td>5000</td>
<td>10000</td>
<td>-2,77E+05</td>
</tr>
<tr>
<td>1</td>
<td>5000</td>
<td>10000</td>
<td>-1,10E+06</td>
</tr>
<tr>
<td>100</td>
<td>1000</td>
<td>30000</td>
<td>2,00E+04</td>
</tr>
<tr>
<td>100</td>
<td>1250</td>
<td>30000</td>
<td>1,61E+04</td>
</tr>
<tr>
<td>100</td>
<td>1500</td>
<td>30000</td>
<td>1,17E+04</td>
</tr>
<tr>
<td>100</td>
<td>2000</td>
<td>30000</td>
<td>1,83E+03</td>
</tr>
</tbody>
</table>
The red results are the ones which satisfy the condition. Consequently the critical buckling pressure for these values can be calculated with the formula above.

Both analytical and ANSYS solutions for the values of the radius, length and thickness that satisfy the condition are shown in the following chart:

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>Radius (mm)</th>
<th>Length (mm)</th>
<th>Length Condition (&gt;0)</th>
<th>Analytical Critical Load (Mpa)</th>
<th>ANSYS Critical Load (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>520</td>
<td>10000</td>
<td>1,65E+03</td>
<td>3,282448094</td>
<td>3,8119736</td>
</tr>
<tr>
<td>20</td>
<td>500</td>
<td>10000</td>
<td>2,13E+03</td>
<td>3,692307692</td>
<td>3,4623625</td>
</tr>
<tr>
<td>20</td>
<td>480</td>
<td>10000</td>
<td>2,59E+03</td>
<td>4,173344017</td>
<td>4,2283282</td>
</tr>
<tr>
<td>20</td>
<td>400</td>
<td>10000</td>
<td>4,37E+03</td>
<td>7,211538462</td>
<td>6,9130227</td>
</tr>
<tr>
<td>20</td>
<td>250</td>
<td>10000</td>
<td>7,22E+03</td>
<td>29,53846154</td>
<td>26,2946260</td>
</tr>
<tr>
<td>20</td>
<td>480</td>
<td>70000</td>
<td>6,26E+04</td>
<td>4,173344017</td>
<td>3,9273160</td>
</tr>
<tr>
<td>100</td>
<td>1000</td>
<td>30000</td>
<td>2,00E+04</td>
<td>57,69230769</td>
<td>49,9740610</td>
</tr>
<tr>
<td>100</td>
<td>1250</td>
<td>30000</td>
<td>1,61E+04</td>
<td>29,53846154</td>
<td>26,7505630</td>
</tr>
<tr>
<td>100</td>
<td>1500</td>
<td>30000</td>
<td>1,17E+04</td>
<td>17,09401709</td>
<td>16,2011030</td>
</tr>
<tr>
<td>100</td>
<td>1700</td>
<td>30000</td>
<td>7,93E+03</td>
<td>11,74278602</td>
<td>11,6616720</td>
</tr>
<tr>
<td>100</td>
<td>2000</td>
<td>30000</td>
<td>1,83E+03</td>
<td>7,211538462</td>
<td>7,9510104</td>
</tr>
<tr>
<td>100</td>
<td>4000</td>
<td>400000</td>
<td>3,20E+05</td>
<td>0,901442308</td>
<td>0,8714321</td>
</tr>
<tr>
<td>100</td>
<td>4000</td>
<td>160000</td>
<td>8,03E+04</td>
<td>0,901442308</td>
<td>0,8848817</td>
</tr>
</tbody>
</table>
L=10000mm, t=20mm:

![Diagram 2: Critical load as a function of the radius (L=10000, t=20)](image1.png)

L=30000mm, t=100mm

![Diagram 3: Critical load as a function of the radius (L=30000, t=100)](image2.png)

Finally, one last point is going to be proved. The analytical formula that has been used in this
chapter of the project, should satisfy two requirements in order to be applied correctly:

- Single-wave buckling mode.
- Long cylinder.

We will check graphically with ANSYS Post-Processor if some of the combinations of length, radius and thickness, which satisfy the Length Condition, fulfill also these two requirements.

\[ L=10000\text{mm}, R=500\text{mm}, t=20\text{mm} \]

**Figure 31: ANSYS top and 3D views (L=10000, R=500, t=20)**

\[ L=30000\text{mm}, R=1700\text{mm}, t=100\text{mm} \]

**Figure 32: ANSYS top and 3D views (L=30000, R=1700, t=100)**
6.3 Comparison between the results from Solid45 and Shell63

As previously explained in another part of the project, two different kinds of elements will be used: Shell63 and Solid45. For a normal cylinder, with a rectangular section the ANSYS results should be totally comparable. But there is something that should be previously commented.

Shell elements are only appropriate when the thickness of the structure is small compared to other dimensions. In ANSYS, only the mid-surface of the structure is modeled, while the thickness and other cross-sectional properties are incorporated into the element stiffness matrix and input as “real constants”.

For this reason, different cylinders will be analyzed, changing the ratio Radius/Thickness, so that we can find out from what ratio, the results obtained with Shell63 and Solid45 are no more comparable.

The radius and the length of the cylinder will be constant and equal 5000mm and 10000mm respectively.

**R/t= 500 → t=10mm**

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.24389990E-01</td>
<td>1</td>
<td>0.21630183E-01</td>
</tr>
<tr>
<td>2</td>
<td>0.24389984E-01</td>
<td>2</td>
<td>0.21630183E-01</td>
</tr>
<tr>
<td>3</td>
<td>0.25774956E-01</td>
<td>3</td>
<td>0.25674579E-01</td>
</tr>
<tr>
<td>4</td>
<td>0.25774956E-01</td>
<td>4</td>
<td>0.25674579E-01</td>
</tr>
<tr>
<td>5</td>
<td>0.26104903E-01</td>
<td>5</td>
<td>0.23109449E-01</td>
</tr>
</tbody>
</table>

**R/t= 250 → t=20mm**

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.12784779</td>
<td>1</td>
<td>0.12378428</td>
</tr>
<tr>
<td>2</td>
<td>0.12784779</td>
<td>2</td>
<td>0.12378428</td>
</tr>
<tr>
<td>3</td>
<td>0.13198897</td>
<td>3</td>
<td>0.12611943</td>
</tr>
<tr>
<td>4</td>
<td>0.13198897</td>
<td>4</td>
<td>0.12611943</td>
</tr>
<tr>
<td>5</td>
<td>0.14904431</td>
<td>5</td>
<td>0.14047330</td>
</tr>
</tbody>
</table>
6. SIMULATIONS WITH ANSYS

Figure 33: ANSYS 3D Views for Solid45 and Shell63 (t=20)

R/t= 100 → t=50mm

Solid45

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.2489383</td>
</tr>
<tr>
<td>2</td>
<td>1.2489383</td>
</tr>
<tr>
<td>3</td>
<td>1.2599862</td>
</tr>
<tr>
<td>4</td>
<td>1.2599862</td>
</tr>
<tr>
<td>5</td>
<td>1.4394226</td>
</tr>
</tbody>
</table>

Shell63

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.2477085</td>
</tr>
<tr>
<td>2</td>
<td>1.2477085</td>
</tr>
<tr>
<td>3</td>
<td>1.2619546</td>
</tr>
<tr>
<td>4</td>
<td>1.2619546</td>
</tr>
<tr>
<td>5</td>
<td>1.4286060</td>
</tr>
</tbody>
</table>

R/t= 50 → t=100mm

Solid45

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7.0152821</td>
</tr>
<tr>
<td>2</td>
<td>7.0152821</td>
</tr>
<tr>
<td>3</td>
<td>7.1317764</td>
</tr>
<tr>
<td>4</td>
<td>8.5572255</td>
</tr>
</tbody>
</table>

Shell63

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7.1325955</td>
</tr>
<tr>
<td>2</td>
<td>7.1325955</td>
</tr>
<tr>
<td>3</td>
<td>7.2881945</td>
</tr>
<tr>
<td>4</td>
<td>7.2881945</td>
</tr>
<tr>
<td>5</td>
<td>8.7320299</td>
</tr>
</tbody>
</table>

R/t= 25 → t=200mm

Solid45

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>39.925521</td>
</tr>
<tr>
<td>2</td>
<td>39.925521</td>
</tr>
<tr>
<td>3</td>
<td>40.6542803</td>
</tr>
<tr>
<td>4</td>
<td>40.6542803</td>
</tr>
<tr>
<td>5</td>
<td>49.049274</td>
</tr>
</tbody>
</table>

Shell63

<table>
<thead>
<tr>
<th>SHAPE NUMBER</th>
<th>LOAD MULTIPLIER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>41.831152</td>
</tr>
<tr>
<td>2</td>
<td>41.831152</td>
</tr>
<tr>
<td>3</td>
<td>42.068348</td>
</tr>
<tr>
<td>4</td>
<td>42.068348</td>
</tr>
<tr>
<td>5</td>
<td>52.235730</td>
</tr>
</tbody>
</table>
It can be observed that the results obtained by Solid45 and Shell63 are quite comparable. However they start to differ approximately for a ratio Radius/Thickness equal 10. The reason is really logic. According to the definition of shell elements, they are only efficient for thin structures. When the wall thickness of a cylinder is approximately less than $1/20^{th}$ of the internal diameter, the variation of the tangential stresses through the wall thickness is small and the radial stresses may be neglected. In this case, we can use either Shell Elements or Solid Elements for studying the stability of our cylinder. However, when the thickness is more than this value, the radial stresses are quite significant and Shell Elements cannot take them into account.

In the following table, the results obtained with Solid45 and Shell63 are compared in terms of the “coefficient of variation”.

$$CV = \frac{S}{\bar{X}}$$

where $S = \sqrt{\frac{\sum x_i^2 f_i}{N} - \bar{X}^2}$

The coefficient of variation is always used to compare the dispersion of two or more groups of values.
In the next two diagrams, it can be checked that the results that ANSYS provide with the mesh elements Solid45 and Shell63 are quite similar. However, as previously explained, they start to differ when the ratio Radius/Thickness gets closer to the limit (10).
6.4 Variation of the dimensions and effects in the critical load

Observing the previous chapters of the project, we can come to clear conclusions about the influence of the variations in the critical buckling pressure.

6.4.1 Variation of the Thickness

Using the results of the chapter 6.3, the following chart and diagram can be made:

\[ L=10000\text{mm}, \ R=5000\text{mm} \]
6. SIMULATIONS WITH ANSYS

Diagram 6: Critical Load as a function of the thickness (L=10000mm, R=5000mm)

If the length and the radius are kept constant, an increase in the thickness of the cylinder leads to a bigger critical load. The reason is that if the thickness is increased, the cylinder becomes more stable, so that more pressure is needed to make it buckling.

6.4.2 Variation of the Radius

In order to analyze the effects of radius variations, some results of the chapter 6.2 will be used.

\( L=10000, t=20 \)

<table>
<thead>
<tr>
<th>Radius (mm)</th>
<th>Analytical Critical Load (Mpa)</th>
<th>ANSYS Critical Load (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>520</td>
<td>3,282448094</td>
<td>3,8119736</td>
</tr>
<tr>
<td>500</td>
<td>3,692307692</td>
<td>3,4623625</td>
</tr>
<tr>
<td>480</td>
<td>4,173344017</td>
<td>4,2283282</td>
</tr>
<tr>
<td>400</td>
<td>7,211538462</td>
<td>6,9130227</td>
</tr>
<tr>
<td>250</td>
<td>29,53846154</td>
<td>26,2946260</td>
</tr>
</tbody>
</table>
As expected, the critical buckling pressure decreases when the radius is increased. It is related with the ratio Radius/Thickness. When the radius is increased keeping the thickness constant, the thickness becomes smaller in comparison to the radius and consequently the cylinder less stable. This instability leads to a smaller critical buckling pressure.

### 6.4.3 Variation of the length

The analytical formula is not suitable for checking the effects of the length variations, because the variable length does not appear on the analytical equation. That is the reason why only the ANSYS results will be analyze in this part.

### R=5000, t=20

<table>
<thead>
<tr>
<th>Length (mm)</th>
<th>ANSYS Critical Load (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>0,46459515</td>
</tr>
<tr>
<td>5000</td>
<td>0,26445013</td>
</tr>
<tr>
<td>10000</td>
<td>0,12784779</td>
</tr>
<tr>
<td>30000</td>
<td>0,04098195</td>
</tr>
<tr>
<td>50000</td>
<td>0,02430388</td>
</tr>
<tr>
<td>80000</td>
<td>0,01481652</td>
</tr>
</tbody>
</table>
6. SIMULATIONS WITH ANSYS

Diagram 8: Critical Load as a function of the length (R=5000mm, t=20mm)

We can conclude that maintaining the thickness and the radius constant, the bigger the length, the less external pressure is needed to reach the critical buckling point, from which the buckling starts.

6.5 Buckling of cylindrical shells with gradual variation of thickness

In this chapter, the buckling of cylinders with truncated cone section under external pressure will be analyzed. Obviously, the ANSYS element Shell 63 is not suitable for this problem, because the thickness is not constant.

The following model will be studied:
The influence of the radius variations can be observed not only in the ANSYS results for the critical buckling pressure, but also in the graphical simulations.

Maintaining constant the length (10000mm), the thickness in the top part (10mm) and the thickness in the bottom part (30mm), the results for different values of the radius are expressed in the following chart:

<table>
<thead>
<tr>
<th>Radius (mm)</th>
<th>Critical Buckling Load (n=1) (MPa)</th>
<th>Critical Buckling Load (n=3) (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>2.563713</td>
<td>3.3195989</td>
</tr>
<tr>
<td>1000</td>
<td>0.99782318</td>
<td>1.0895968</td>
</tr>
<tr>
<td>2000</td>
<td>0.36983587</td>
<td>0.38685274</td>
</tr>
<tr>
<td>3000</td>
<td>0.20758785</td>
<td>0.20864187</td>
</tr>
<tr>
<td>5000</td>
<td>0.10035773</td>
<td>0.1021332</td>
</tr>
</tbody>
</table>

It can be easily appreciated that the critical buckling values for both buckling modes are quite comparable and similar. However, it can be assured that the difference between the values of the two buckling modes increases when the Radius is reduced.
This can be showed in the following graphic:

![Critical Load as a function of the Radius](image)

*Diagram 9: Critical load as a function of the Radius (L=10000mm, t_{top}=10mm, t_{bottom}=30mm)*

Nevertheless, it is also important to know, what effects the radius variations have in the buckling shape. The next figures answer this question and show the cylinder’s behavior with different radius configurations.

*Figure 35: ANSYS 3D view (R=5000mm)  Figure 36: ANSYS 3D view (R=3000mm)*
After these results it can be concluded, that when the radius is diminished, maintaining the other dimensions constant, the number of buckling waves decreases, while the amplitude of the buckling wave increases.

On the other hand, it is also interesting to see what is happening with the critical load in comparison to the normal rectangular section of the previous chapters.

Attending to the logic, although the thickness in the bottom part of the cylinder is bigger than in the previous cylinders, it can be expected that the values for the truncated cone section are smaller due to the smaller thickness of the top part of the cylinder. The following chart and diagram show this comparison:
### 6. SIMULATIONS WITH ANSYS

<table>
<thead>
<tr>
<th>Radius (mm)</th>
<th>Critical Buckling Load (Truncated cone section) (MPa)</th>
<th>Critical Buckling Load (Rectangular Section) (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>2,563713</td>
<td>3,8119736</td>
</tr>
<tr>
<td>1000</td>
<td>0,99782318</td>
<td>1,3826085</td>
</tr>
<tr>
<td>2000</td>
<td>0,36983587</td>
<td>0,48620833</td>
</tr>
<tr>
<td>3000</td>
<td>0,20758785</td>
<td>0,26628536</td>
</tr>
<tr>
<td>5000</td>
<td>0,10035773</td>
<td>0,12784779</td>
</tr>
</tbody>
</table>

Diagram 10: Critical load as a function of the radius for both section configurations

As expected, the obtained values for the truncated cone section are smaller and the difference between the values for both sections become bigger when the radius is decreased. That is related with the ratio Radius/Thickness. As explained before, in the cylinder with truncated cone section, the thickness in the top part is smaller than in the cylinder rectangular section.

When the radius is decreased, the thickness gains more importance, cause it is bigger in comparison with the radius and the gaining of importance leads to a smaller critical buckling pressure.

The difference between both types of cylinder sections is shown in the following figures for a radius equal to 500mm.
In the figures above it can be noticed that for the truncated cone section, the maximal displacement is closer to the top part of the cylindrical shell than in the shell with rectangular section, due to the effect of the smaller thickness in the top part of the section.

In the next part of this chapter, the variation of the top and bottom thicknesses will be studied. The variables will be both thicknesses, while the length and the radius will be constant during the analysis.
Figure 42

<table>
<thead>
<tr>
<th>Thickness\text{top} (mm)</th>
<th>Thickness\text{bottom} (mm)</th>
<th>Critical Buckling Load (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>10</td>
<td>0,02435385</td>
</tr>
<tr>
<td>10</td>
<td>30</td>
<td>0,10035773</td>
</tr>
<tr>
<td>10</td>
<td>50</td>
<td>0,20884732</td>
</tr>
<tr>
<td>10</td>
<td>70</td>
<td>0,32976949</td>
</tr>
<tr>
<td>10</td>
<td>90</td>
<td>0,46112851</td>
</tr>
<tr>
<td>30</td>
<td>10</td>
<td>0,12633996</td>
</tr>
<tr>
<td>30</td>
<td>30</td>
<td>0,35124401</td>
</tr>
<tr>
<td>30</td>
<td>50</td>
<td>0,64317008</td>
</tr>
<tr>
<td>30</td>
<td>70</td>
<td>1,0115178</td>
</tr>
<tr>
<td>30</td>
<td>90</td>
<td>1,4318338</td>
</tr>
<tr>
<td>50</td>
<td>10</td>
<td>0,26946392</td>
</tr>
<tr>
<td>50</td>
<td>30</td>
<td>0,71891277</td>
</tr>
<tr>
<td>50</td>
<td>50</td>
<td>1,2561049</td>
</tr>
<tr>
<td>50</td>
<td>70</td>
<td>1,8449868</td>
</tr>
<tr>
<td>50</td>
<td>90</td>
<td>2,5302595</td>
</tr>
<tr>
<td>70</td>
<td>10</td>
<td>0,42895588</td>
</tr>
<tr>
<td>70</td>
<td>30</td>
<td>1,2195365</td>
</tr>
<tr>
<td>70</td>
<td>50</td>
<td>1,969839</td>
</tr>
<tr>
<td>70</td>
<td>70</td>
<td>2,8312625</td>
</tr>
<tr>
<td>70</td>
<td>90</td>
<td>3,8378579</td>
</tr>
<tr>
<td>90</td>
<td>10</td>
<td>0,6055315</td>
</tr>
</tbody>
</table>
6. SIMULATIONS WITH ANSYS

<table>
<thead>
<tr>
<th></th>
<th>30</th>
<th>50</th>
<th>70</th>
<th>90</th>
</tr>
</thead>
<tbody>
<tr>
<td>90</td>
<td>1,8073843</td>
<td>2,8569736</td>
<td>4,0569011</td>
<td>5,4520183</td>
</tr>
</tbody>
</table>

Diagram 11: Critical Load as a function of the top thickness (L=10000mm, R=5000mm)

Diagram 12: Critical Load as a function of the bottom thickness (L=10000mm, R=5000mm)
The figures above show the effect that variations of the top and bottom thicknesses have in the final value of the critical buckling pressure.

On one hand, it can be easily recognized that the values of the critical load follow an almost linear relation when the thickness of the bottom part of the section is changed. On the other hand, when the top thickness is changed, the relation is not linear.

This difference between both figures is related with the different initial constraints applied to both parts of the truncated cone section. While in the bottom part of the section, the displacements in x,y and z directions are restricted, in the top part, only the displacement in x is restricted.

6.6 Buckling of cylindrical shells with curve section

In this chapter, typical imperfections of cylindrical shells will be simulated. Sometimes when manufacturing processes are not completely exact, the thickness of shells is not perfectly regular.

The thickness of the middle of the wall will take different sizes than in the edges. Two different situations will be studied:

1. When the thickness in the middle point of the height is bigger than in the edges (Cylinder1).
2. When the thickness in the middle point of the height is smaller than in the edges (Cylinder 2).

![Figure 44](image)

Varying the radius of the cylinder and maintaining the sections constant, the following values for both shells with imperfections are obtained:

<table>
<thead>
<tr>
<th>Radius (mm)</th>
<th>Critical Load Cylinder 1 (Mpa)</th>
<th>Critical Load Cylinder 2 (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>8,3526335</td>
<td>1,1149611</td>
</tr>
<tr>
<td>1000</td>
<td>3,0931381</td>
<td>0,44823423</td>
</tr>
<tr>
<td>2000</td>
<td>0,94642763</td>
<td>0,16688149</td>
</tr>
<tr>
<td>3000</td>
<td>0,51352842</td>
<td>0,092657049</td>
</tr>
<tr>
<td>5000</td>
<td>0,24254941</td>
<td>0,046025382</td>
</tr>
</tbody>
</table>

As expected, the cylinder with bigger thickness in the middle of the wall needs a bigger load to buckle, which means that it is a stronger shell against buckling. Although the thickness in the edges of the Cylinder1 is smaller in comparison to the standard cylinder with rectangular section, the increase of the thickness in the middle point of the wall behave like reinforcement against buckling under external pressure.

However, with the second cylinder the opposite result is obtained. The decrease of the thickness in the middle of the wall has a bigger effect than the increase of the thickness in the
edges. That leads to a lower strength against buckling, so that the structure will buckle easily.

That can be checked in the following diagram:

![Critical Load as a function of the radius](image)

**Diagram 13: Critical load as a function of the radius for different sections**

As far as the displacements are concerned, totally different results for both cylindrical shells are reached. These results will be analyzed using ANSYS graphical post-processing tools.

Using a value of the radius equal to 1000mm, the deformed shapes in these two shells can be observed in the following figures from the top of the cylinders.

![ANSYS top view of Cylinder 1](image)

**Figure 45: ANSYS top view of Cylinder 1**
It can be quickly noticed, that there is a significant difference in the deformed shapes of both shells. In the top of the Cylinder 1 a light blue band around the normal cylindrical shape turns the attention. The blue band reflects the deformation occurred in the area close to the top of the cylinder. In Cylinder 1, the thickness of the edges is smaller, so that the deformation of the areas close to the edges is consequently bigger in Cylinder 1 than in Cylinder 2, and that is the main reason why the band can be found only in the first figure.

The following figures, obtained with ANSYS, shows the displacements occurred in the shells in a more accurate way:
6. SIMULATIONS WITH ANSYS

Figure 47: ANSYS Cylinder 1

Figure 48: ANSYS Cylinder 2
There are a few details to stand out in both displacement figures.

As previously observed in ANSYS deformed shapes, the cylinder 1 has higher displacements in the area close to the top of cylinder, due to the smaller thickness in that part of the section.

On the one hand, the maximal displacement in the Cylinder2 takes place in the middle of the shell, where the thickness is the smallest. On the other hand, the maximal displacement in Cylinder 1 occurs in the middle of the top half of the shell, due to the increase of the thickness in that part of the shell and the difference between the constraints in the top and bottom areas of the cylinder.

At least, it is also important to mention, the difference in terms of values of displacements for both cylinders. In the second cylinder the maximum displacement is more than 2.5 times bigger than the maximum displacement in Cylinder1.
7. Conclusions

The basic idea of this thesis was to discuss the influence that different dimensions (such as length, radius and thickness) have in the value of the critical buckling load of cylindrical shells under external pressure.

In order to achieve that aim, also different sections configurations have been analyzed. That has provided information about the buckling behavior of cylinders with different imperfections.

After applying different configurations, an important conclusion have been reached. As far as buckling is concerned, the critical and most important and representative dimension of cylinders under external pressure is the ratio Radius/Thickness. When this ratio is increased, the system becomes more unstable and consequently, the critical buckling load decreases.

Because of my interest in the field of finite element analysis, I have decided to deepen my knowledge in this area through this thesis. In this thesis, I have succeeded in this area further training and to integrate the measurement program ANSYS with success.

The skills learned are a good foundation for my career to become a design engineer.
8. Bibliography


