University of
Stavanger

Faculty of Science and Technology

MASTER'S THESIS

Study program/ Specialization: Offshore technology – Marine and Subsea Technology.	Spring semester, 2012 Open / Restricted access
Writer: Frode Tjelta	………………………………………… (Writer's signature)

Faculty supervisors: Ove Tobias Gudmestad, Conrad Carstensen.

External supervisor: Helge Nesse, IKM Ocean Design.

Title of thesis: A comparison study of pressure vessel design using different standards.

Credits (ECTS): 30

Key words: Pressure vessel, Elastic stress analysis, Plastic stress analysis, Direct route, Finite element method, ASME VIII div. 2, 2010, NS-EN 13445; 2009, Calculations, Comparison, Recommendations. Guide to pressure vessel design.	Pages: 139 + Enclosure: 180 Stavanger, 15th June, 2012

A comparison study of pressure vessel design using different standards

Frode Tjelta | IKM Ocean Design
2012

University of Stavanger

© 2012
Frode Tjelta
ALL RIGHTS RESERVED

Dedicated to future students and

to my fiancée Ellen-Marita

Abstract

Due to a recent pressure vessel design error (see chapter 1.2.7) the design methods used for pressure vessel design is investigated.

Several codes are currently available for design and analysis of pressure vessels. Two of the main contributors are the American Society of Mechanical Engineers providing the ASME VIII code, and the Technical Committee in Brussels providing the European Standard.

Methods written in **bold** letters will be considered in this thesis.

The ASME VIII code contains three divisions covering different pressure ranges:

- Division 1: up to 200 bar (3000 psi)
- **Division 2: in general**
- Division 3: for pressure above 690 bar (10000 psi)

In this thesis the ASME division 2 Part 5 will be considered. This part is also referred to in the DNV-OS-F101 for offshore pressure containing components. Here different analysis methods are described, such as:

- **Elastic Stress Analysis**
- Limit Load Analysis
- **Elastic Plastic Analysis**

The Elastic Stress Analysis method with stress categorization has been introduced to the industry for many years and has been widely used in design of pressure vessels. However, in the latest issue (2007/2010) of ASME VIII div. 2, this method is not recommended for heavy wall constructions as it might generate non-conservative analysis results.

Heavy wall constructions are defined by: ($\frac{R}{t} \leq 4$) as illustrated in Figure 1.

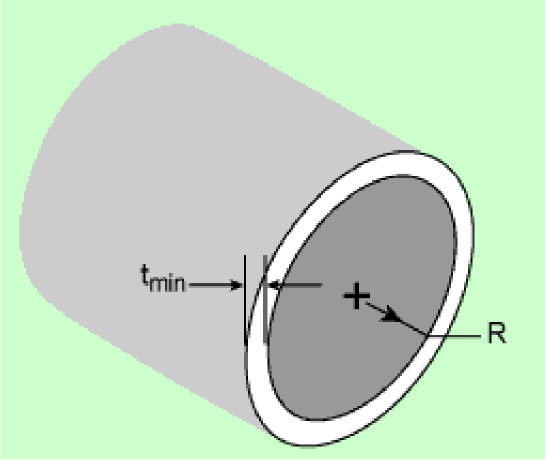


Figure 1: Simplified pressure vessel geometry.

In the case of heavy wall constructions the Limit Load Analysis or the Elastic-plastic method shall be used. In this thesis focus will be on the Elastic-plastic method and the Limit Load Analysis will not be considered.

Experience from recent projects at IKM Ocean Design indicates that the industry has not been fully aware of the new analysis philosophy mentioned in the 2007 issue of ASME VIII div.2. The Elastic Stress Analysis method is still (2012) being used for heavy wall constructions.

The NS-EN 13445-3; 2009 provides two different methodologies for design by analysis:

- **Direct Route**
- Method based on stress categories.

The method based on stress categories is similar to the Elastic Stress Analysis method from ASME VIII div. 2 and it will therefore not be considered in this thesis.

Heavy wall construction is not mentioned in NS-EN 13445. Therefore this thesis shall compare the results obtained using the Direct Route approach with the ASME VIII div. 2 for heavy wall pressure containing components.

The thesis will present some theory and examples to gain a general understanding about the content to be presented. The methods will be described in detail with references to the standard they are adopted from. Advantages and disadvantages for the different methods shall be included where applicable.

A complete design basis for a heavy wall pressure vessel and a thin wall pressure vessel will be established. Complete construction drawing sets, part lists, 3D models and material properties shall be included. Future construction and production of the pressure vessels for testing purposes shall be possible using the information provided in this thesis.

The analysis tools used will be described in detail and model simplifications shall be explained. The calculation shall be carried out with respect to the relevant standard and the approach will be presented in a way that is easy to follow for the reader. The results will be presented in table format for easy comparison.

The use of the different methods shall be commented upon. The comments will be based upon experience gained during the work with this thesis

Recommendation on application of the different methods will be given along with a recommended scope for possible further studies.

Acknowledgements

The author would like to thank:

- Professor Ove Tobias Gudmestad, my faculty supervisor, for his support and guidance which have been a remarkable help during the work process.
- Helge Nesse, my supervisor at IKM Ocean Design, for introducing me to his company in the form of summer work and providing me with an interesting theme for my thesis. His help during my thesis have been tremendous.
- Loyd Kjetil Andersen at IKM Ocean Design for his valuable input to the calculations in the part containing ASME VIII in this thesis.
- Asle Seim Johansen at IKM Ocean Design for his help with the part containing FE analysis using ANSYS workbench.
- Eric Risa for helping me with Inventor 3D modeling.
- All other colleagues at IKM Ocean Design for the fantastic working environment in the company.
- IKM Ocean Design for providing me with equipment and office space.
- The University of Stavanger for providing me with the knowledge needed to solve complex engineering problems.
- And finally my fantastic fiancée Ellen-Marita Askestrand for her support and understanding during the time I`ve been working with this thesis.

Table of contents

List of figures

Nomenclature

Symbols

Latin characters

A	General action (EN-13445-3)
A_1	Curve fitting constant for the elastic region of the stress-strain curve (ASME VIII div. 2)
A_2	Curve fitting constant for the plastic region of the stress-strain curve (ASME VIII div. 2)
a	Radius of hot spot within a plate (ASME VIII div. 2)
D	Fatigue damage (EN-13445-3)
D_f	Cumulative fatigue damage (ASME VIII div. 2)
$D_{f,k}$	Fatigue damage for the k^{th} cycle
D_ε	Cumulative strain limit damage
$D_{\varepsilon\,form}$	Strain limit damage from forming
$D_{\varepsilon,k}$	Strain limit damage for the k^{th} loading condition
d	Design value (EN-13445-3)
E	Exceptional action (EN-13445-3)
E	Young`s modulus
E_d	Combined design effects of various actions.
E_y	Modulus of elasticity evaluated at the temperature of interest (ASME VIII div. 2)
$E_{ya,k}$	Value for modulus of elasticity of the point under consideration, at the k^{th} cycle
F	Additional stress produced by a stress concentration (ASME VIII div. 2)
G	Permanent action (EN-13445-3)
H_1	Stress-strain curve fitting parameter (ASME VIII div. 2)
i	i^{th} value (EN-13445-3)
inf	Lower bound (EN-13445-3)
j	j^{th} value (EN-13445-3)
K_1	Material parameter for stress-strain curve model (ASME VIII div. 2)
$K_{e,k}$	Fatigue penalty factor for the k^{th} cycle
$K_{v,k}$	Plastic Poisson’s ratio adjustment for local thermal and thermal bending stresses for the k^{th} cycle.
K_f	Fatigue strength reduction factor used to compute the cyclic stress amplitude or range
k	k^{th} value (EN-13445-3)
M	Total number of stress ranges at a point derived from the cycle counting procedure
M_1	Curve fitting exponent for the stress-strain curve equal to the true strain at the proportional limit and the strain hardening coefficient in the large strain region (ASME VIII div. 2)
M_2	Curve fitting exponent for the stress-strain curve equal to the true strain at the true ultimate stress (ASME VIII div. 2)
N_k	Permissible number of cycles for the k^{th} cycle
n_k	Actual number of repetitions for the k^{th} cycle
$N_{\Delta FP}$	Design number of full-range cycles (ASME VIII div. 2)
$N_{\Delta P0}$	Expected number of operating cycles in which the range exceeds a given value
$N_{\Delta TE}$	Number of cycles associated with ΔT_E (ASME VIII div. 2)
$N_{\Delta T\alpha}$	Number of temperature cycles for components with different coefficient of expansion.
P	Pressure action (EN-13445-3)

P	Specified design pressure (ASME VIII div. 2)
P_b	Primary bending equivalent stress (ASME VIII div. 2)
P_M	Primary membrane equivalent stress (ASME VIII div. 2)
P_L	Local primary membrane equivalent stress (ASME VIII div. 2)
P_d	Design pressure (EN-13445-3)
P_s	Maximum allowable pressure (EN-13445-3)
Q	Variable action (EN-13445-3)
Q	Secondary equivalent stress from operating loadings (ASME VIII div. 2)
R	Inside radius (ASME VIII div. 2)
R_1	Engineering yield to engineering tensile ratio (ASME VIII div. 2)
R_d	Design resistance.
RM	Material strength parameter (EN-13445-3)
R_{eH}	Minimum upper yield strength (EN-13445-3)
R_m	Minimum tensile strength (EN-13445-3)
$R_{m/t}$	Minimum tensile strength at temperature t in °C (EN-13445-3)
$R_{p\,0.2}$	Minimum 0.2% proof strength (EN-13445-3)
$R_{p\,0.2/t}$	Minimum 0.2% proof strength at temperature t in °C (EN-13445-3)
$R_{p\,1.0}$	Minimum 1.0% proof strength (EN-13445-3)
$R_{p\,1.0/t}$	Minimum 1.0% proof strength at temperature t in °C (EN-13445-3)
S	Allowable stress based on material and temperature (ASME VIII div. 2)
S_e	Computed equivalent stress (ASME VIII div. 2)
S_{PS}	*A*llowable limit on the primary plus secondary stress range
S_u	Minimum ultimate strength
S_y	Minimum specified yield strength at design temperature
$S_{a,\,k}$	Value of alternating stress obtained from the design fatigue curve
$S_{alt,k}$	Alternating equivalent stress for the k^{th} cycle.
$S_{y,\,k}$	Yield strength for the material evaluated at the k^{th} cycle
sup	Upper bound (EN-13445-3)
t	Calculation temperature (EN-13445-3)
t	Wall thickness (ASME VIII div. 2)
$t_{c\ max/min}$	Maximum and minimum temperature during an action cycle (EN-13445-3)
t_d	Design temperature (EN-13445-3)

Greek characters

$\Delta S_{n,k}$	Primary plus secondary equivalent stress range
$\Delta S_{P,k}$	Range of primary plus secondary plus peak equivalent stress range for the k^{th} cycle
$\Delta S_{LT,k}$	Local thermal equivalent stress for the k^{th} cycle
ΔT	Operating temperature range
ΔT_E	Effective number of changes in material temperature between any two adjacent points
$\Delta\sigma_{ij}$	Stress tensor range
$\Delta\sigma_{ij,k}$	Stress tensor range at the point under evaluation for the k^{th} cycle
$\Delta\varepsilon_{peq, k}$	Equivalent plastic strain range for the k^{th} cycle
$\Delta\varepsilon_{eff, k}$	Effective strain at the k^{th} cycle
$\Delta P_{ij, k}$	Change in plastic strain range components for th k^{th} cycle
Φ_B	Design factor for buckling (ASME VIII div. 2)
α	Thermal expansion coefficient of the material (ASME VIII div. 2)
α_1	Thermal expansion coefficient of material 1 evaluated at mean temperature of the cycle
α_2	Thermal expansion coefficient of material 2 evaluated at mean temperature of the cycle
α_{sl}	Material factor for the multi axial strain limit
B_{cr}	Capacity reduction factor (ASME VIII div. 2)
γ	Partial safety factor (EN-13445-3)
γ_1	True strain in the micro-strain region of the stress-strain curve (ASME VIII div. 2)
γ_2	True strain in the macro-strain region of the stress-strain curve (ASME VIII div. 2)
ε	Strain
ε_1	True plastic strain in the micro-strain region of the stress-strain curve (ASME VIII div. 2)
ε_2	True plastic strain in the macro-strain region of the stress-strain curve (ASME VIII div. 2)
ε_{cf}	Cold forming strain
ε_e	Elastic strain
ε_L	Limiting tri axial strain
ε_{LU}	Uniaxial strain limit
ε_p	Plastic strain
ε_p	Stress-strain curve fitting parameter (ASME VIII div. 2)
ε_t	Total true strain (ASME VIII div. 2)
ε_{ys}	0,2% engineering offsett strain (ASME VIII div. 2)
ρ	Mass density
ν	Poisson ratio
σ_t	True stress at which the strain will be evaluated (ASME VIII div. 2)
σ_{YS}	Engineering yield stress evaluated at the temperature of interest (ASME VIII div. 2)
σ_{UTS}	True ultimate tensile stress evaluated at the true ultimate tensile strain (ASME VIII div. 2)
σ_1	Principal stress in 1-direction, Maximum principal stress
σ_2	Principal stress in 2-direction, Middle principal stress
σ_3	Principal stress in 3-direction, Minimum principal stress

Abbreviations

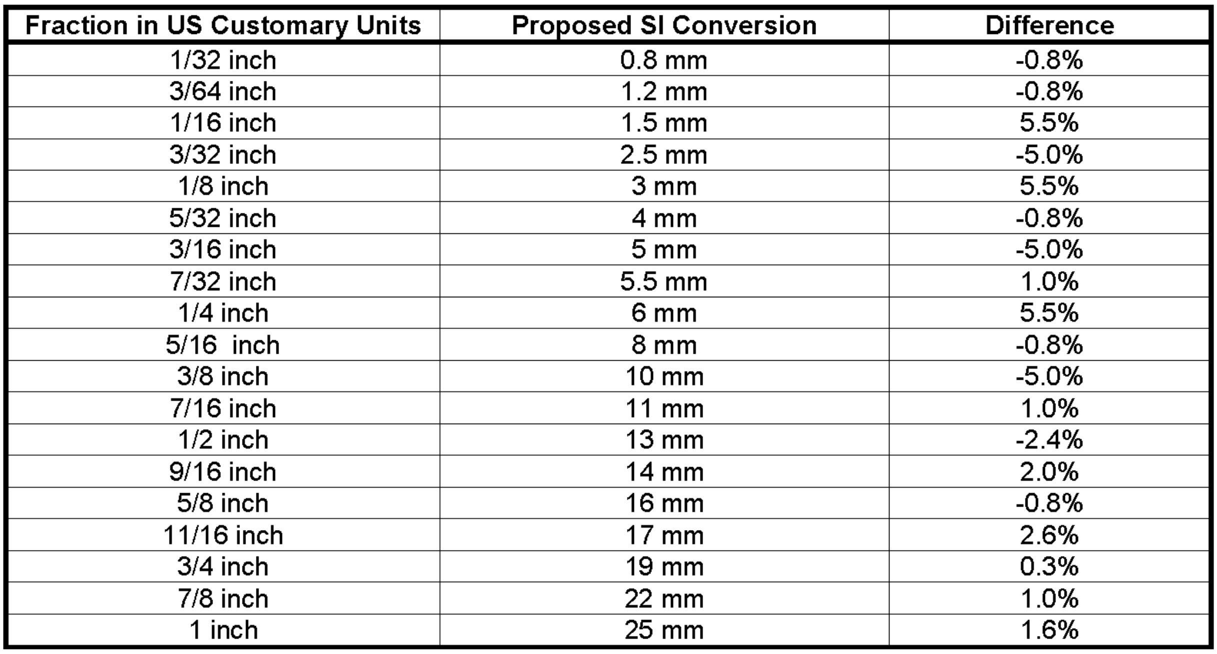
ASME	American Society of Mechanical Engineers
CAD	Computer assisted design
DBA	Design by Analysis
DBF	Design by Formula (see DBR)
DBR	Design by Rule
DOF	Degrees of Freedom
FEA	Finite Element Analysis
ID	Internal diameter
LRFD	Load Resistance Factor Design
NDT	Non Destructive Testing
NS-EN	Norwegian Standard – European Norm
OD	Outer diameter
SCL	Stress Classification Line

Tables

Useful conversion factors[1]			
US Customary units	**SI units**	**Conversion factor**	**Notes**
inches [in]	millimeters [mm]	25,4	
feet [ft]	meters [m]	0,3048	
[in^{2}]	[mm^{2}]	645,16	
[ft^{2}]	[m^{2}]	0,09290304	
[in^{3}]	[mm^{3}]	16,387064	
[ft^{3}]	[m^{3}]	0,02831685	
US Gallon [gal]	[m^{3}]	0,003785412	
pounds [lb]	Newton [N]	4,4482217	
pounds/ square inch [psi]	[MPa]	0,0068948	Used in equations.
[psi]	[kPa]	6,894757	Used for nameplates.
[°F]	[°C]	5/9(°F-32)	Not for temperature difference.
[°F]	[°C]	5/9(°F)	For temperature difference.
[lbm]	[kg]	0,4535924	
[in-lbs]	[N-mm]	112,98484	For use in equations.
[ft-lbs]	[N-m]	1,3558181	For use in text.
[lbs/ft^{3}]	[kg/m^{3}]	16,018463	
Other units			
[bar]	[kPa]	100	The unit bar is widely used in descriptions of pressure.

Figure 2: Useful conversion factors.

The common engineering size and thickness conversions for fractions including differences (e.g. 1 inch are 1,6% more than the proposed SI unit 25 mm) as suggested by, Ref /12/ are given in Figure 3**.**



Fraction in US Customary Units	Proposed SI Conversion	Difference
1/32 inch	0.8 mm	-0.8%
3/64 inch	1.2 mm	-0.8%
1/16 inch	1.5 mm	5.5%
3/32 inch	2.5 mm	-5.0%
1/8 inch	3 mm	5.5%
5/32 inch	4 mm	-0.8%
3/16 inch	5 mm	-5.0%
7/32 inch	5.5 mm	1.0%
1/4 inch	6 mm	5.5%
5/16 inch	8 mm	-0.8%
3/8 inch	10 mm	-5.0%
7/16 inch	11 mm	1.0%
1/2 inch	13 mm	-2.4%
9/16 inch	14 mm	2.0%
5/8 inch	16 mm	-0.8%
11/16 inch	17 mm	2.6%
3/4 inch	19 mm	0.3%
7/8 inch	22 mm	1.0%
1 inch	25 mm	1.6%

Figure 3: Size and thickness conversions.

---

[1] Table content adopted from Ref /1/.

CHAPTER 1 INTRODUCTION

1.1 Thesis Organization

Chapter 1 (Introduction):

The introduction contains the background information, some theory and examples to gain an illustrative understanding about the content of this thesis. The problem is stated followed by the purpose and scope of the thesis. A short thesis organization is also included (this section) to make navigation in the document simple for the reader.

Chapter 2 (Methods):

This section contains general information about the methods considered in this thesis. The procedures for using the different methods are presented in detail with references to the standard they are adopted from. Advantages and disadvantages for the different methods are included and examples are presented where clarification are required.

Chapter 3 (Design basis):

This section of the thesis includes the overall geometrical dimensions of the pressure vessel and the attached nozzle, a purposed CAD drawing and material properties. The loads and load cases are specified along with the acceptance criteria.

Chapter 4 (Analysis tools):

This section contains information regarding the Visual Vessel Design (VVD) calculation software and the FEA software ANSYS workbench. Calculation results from VVD are added for comparison reasons. The proposed 3D models of the vessels are presented in this section. The FEM model and the purposed mesh are also presented in this section. Model simplifications are explained.

Chapter 5 (Calculations):

This section consists of calculations carried out with respect to the relevant standard. The pressure vessel is checked for plastic deformation using different approaches, and the results are commented upon. Figures and graphs are used to increase the readers understanding of the different methods that are being used. All analysis results are combined in table format for easy comparison.

Chapter 6 (Comments):

This section contains comments regarding the use of different methods in pressure vessel design. Comments regarding each method are made based on the results and experience obtained during the work with this thesis.

Chapter 7 (Conclusions and further studies):

Here recommendations are given based on experience gained during the work with this thesis. Further studies are also mentioned for future research based on the results and descriptions in this thesis. Appendixes are added containing all relevant information not referred to in the reference list.

1.2 Background

1.2.1 Design methods

The pressure vessel design methods can be divided into two basic categories regardless of which code or standard they are inherent from, namely the Design by Rule (DBR) method and the Design by Analysis (DBA) method.

The main differences between the categories are:

- **Design by Rule**
  - Overall dimensions and loads are specified.
  - Wall thickness is calculated by predefined procedures, formulas and charts.
- **Design by Analysis**
  - The vessel geometry and overall dimensions are defined.
  - Allowable loads are evaluated through detailed structural analysis.

In this thesis the main focus will be directed towards the design by analysis approach. The structural analysis using this method is divided into elastic analysis and inelastic or plastic analysis depending on anticipated material behavior. Differences between the material behaviors will be explained with reference to the stress-strain diagram presented in Figure 4, Ref /1/.

- A material is said to exhibit linear elastic behavior if loading and unloading occurs along the line 0-A.
- Loading along line 0-L and offloading along L-M is also considered to be elastic behavior within the 0.2% offset range.
- Loading and offloading along the line 0-L is called non-linear elastic behavior.
- Loading along the line 0-J and offloading along J-K is called plastic response.
- Loading along the line 0-J and offloading along J-K-0 is called viscoelastic behavior.
- Loading along the line 0-J and offloading along J-K-M is called viscoplastic behavior.

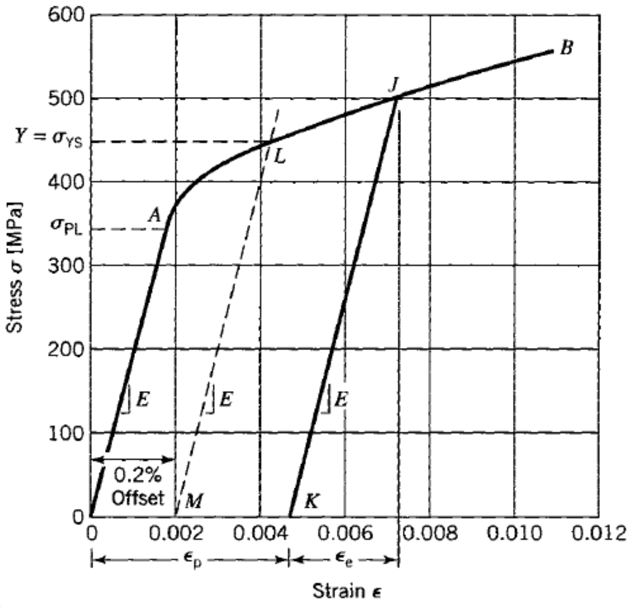


Figure 4: Engineering stress-strain diagram (expanded strain scale), Ref /1/.

1.2.2 Finite element analysis

Recently the finite element analysis (FEA) has entered the practical world of design engineering and stopped being regarded just as an analyst`s tool. Now most design by analysis is performed using the FEA method.

FEA using elastic or linear analysis method provides an acceptable approximation to real life characteristics for most problems related to design engineering. The elastic analysis assumes shell-type membrane and bending stress distributions. The method requires use of shell elements or stress linearization procedures. The thin shell element[2] and its degrees of freedom are shown in Figure 5.

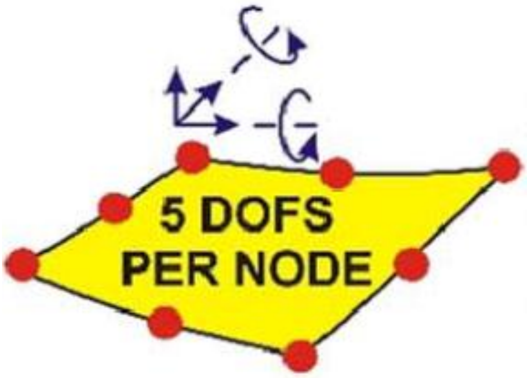


Figure 5: Thin shell element.

FEA using inelastic or non-linear analysis methods is called for when more challenging problems occur. The non-linear analysis may be based on shell elements or solid elements. Improved solution algorithms and powerful desktop computers have made it more feasible to conduct these analyses within reasonable time frame. The solid element[1] and its degrees of freedom are shown in Figure 6.

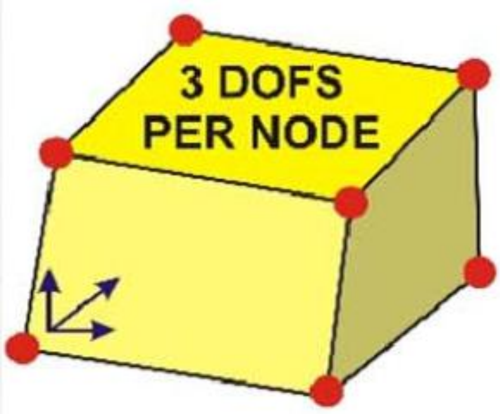


Figure 6: 3D solid element.

---

[2] The element types are adopted from CCOPPS Webinar «The interfacing of FEA with pressure vessel design codes»

1.2.3 Differences between linear and non-linear analysis

The fundamental difference between linear and non-linear (elastic and plastic) analysis lies in the term called “stiffness”.

The different factors affecting the stiffness, Ref /2/:

Shape: An I-beam has different stiffness than a channel or U-beam. Beam geometry differences are shown in Figure 7.

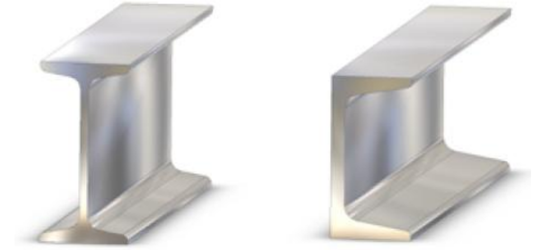


Figure 7: I-beam versus channel beam Ref /2/.

Material: A steel beam has higher stiffness than an iron beam of same dimensions. Material difference illustrated by means of Figure 8.

Figure 8: Steel beam versus iron beam Ref /2/.

Boundary conditions: A beam with simple support is less stiff than the same beam with both end supported as shown in Figure 9.

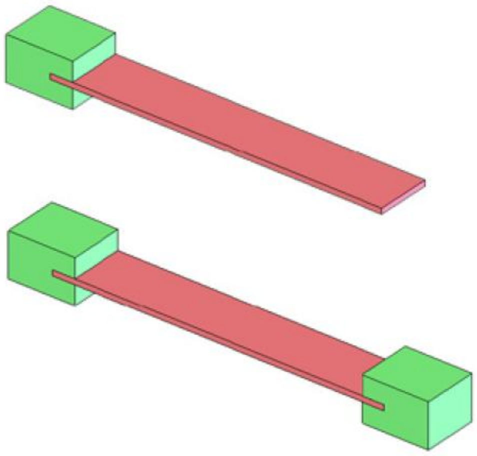


Figure 9: Boundary conditions for simple beam, Ref /2/.

If deformation of a structure occurs due to applied load the stiffness of the structure changes. However, if the change in stiffness is small it is reasonable to assume that neither the shape nor the material properties changes during the deformation process. This is the fundamental principle for linear analysis. Constant stiffness greatly simplifies the calculations required to obtain a numerical solution of a structural problem.

Consider the fundamental FEA equation:

$$[\boldsymbol{F}] = [\boldsymbol{K}] \cdot [\boldsymbol{d}]$$

Where:

$[\boldsymbol{F}]$ is the known vector of nodal loads.

$[\boldsymbol{K}]$ is the known stiffness matrix.

$[\boldsymbol{d}]$ is the unknown vector of nodal displacement.

This equation describes the behavior of FEM models. Depending on the model size it might contain several million linear algebraic equations. The stiffness matrix depends on the geometry, material properties and restrains. Under linear analysis assumptions this is constant and the set of equations are assembled and solved just once. The result is produced in the matter of seconds or minutes.

Considering the non-linear analysis the stiffness matrix must be updated as the non-linear solver progresses through an iterative solution process. Several iteration algorithms exist, among them the Newton-Raphson iterative algorithm and the Arc-length algorithm. The principal of the methods is shown in Figure 10 and Figure 11, Ref /5/.

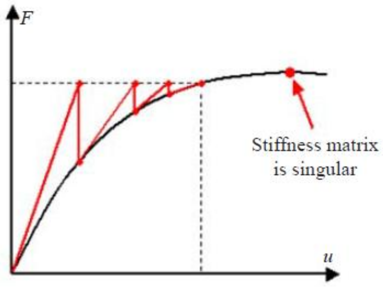


Figure 10: Newton-Raphson iterative algorithm, Ref /5/.

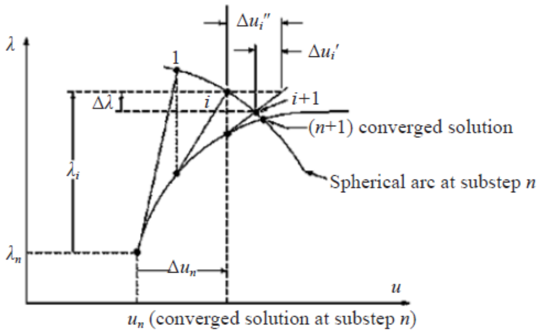


Figure 11: Arc-length algorithm, Ref /5/.

The numerical solution process might take considerable time for geometric complex models.

1.2.4 Large deformation effects

When considering large deformations in the analysis it is important to recognize that the load direction can change as the model deforms. Most of the existing FEA programs offer two choices to account for this direction change, namely the following and non-following load. The difference between following (non-conservative) and non-following (conservative) load are illustrated in Figure 12, Ref /2/.

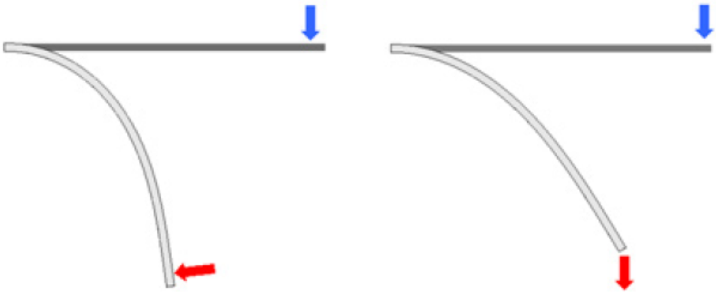


Figure 12: Following load versus non-following load, Ref /2/.

A pressure vessel subjected to very high internal pressure provides a good example of a situation where the following load should be utilized in the analysis. The pressure load always acts normal to the pressure vessels walls regardless of how much deformation the pressure vessel sustains.

Change of shape might cause change of stiffness. An example of this is an initially flat membrane with boundary conditions as illustrated in Figure 13, Ref /2/ deflecting under pressure loading.

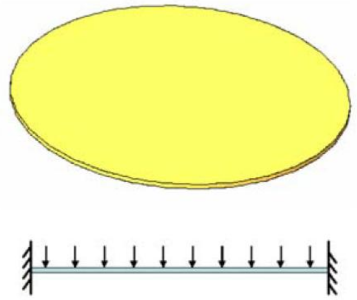


Figure 13: Flat membrane subjected to pressure loading, Ref /2/.

Initially the membrane resists the pressure loading by means of bending stiffness only. After the pressure has caused some deflection or curvature, the stiffness is increased with membrane stiffness in addition to the original bending stiffness as illustrated in Figure 14, Ref /2/.

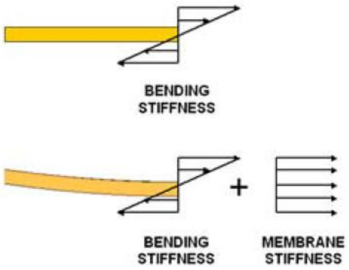


Figure 14: Membrane stiffness, Ref /2/.

1.2.5 Non-linear material

Changes in stiffness might occur due to changes in material properties under operating conditions. This problem is called material non-linearity.

A linear material model assumes stress to be proportional with strain. Another assumption for the linear material model is that it will return to its original shape when the load is removed. The stress-strain curve for the linear material model is show in Figure 15, Ref /2/.

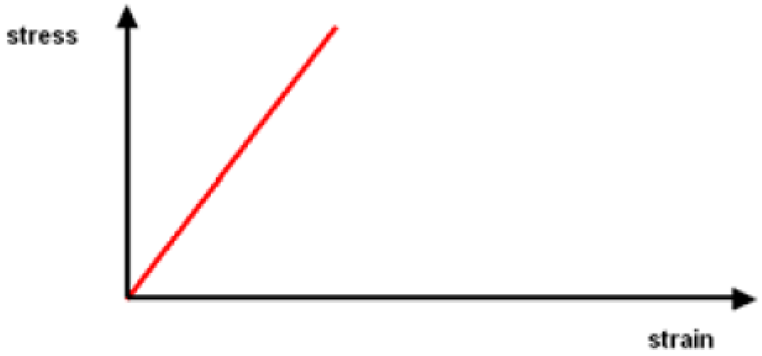


Figure 15: The linear material model, Ref /2/.

This simplification is in many cases acceptable, however if the loads are high enough to cause some permanent deformations the non-linear material model must be used in order to obtain more accurate analysis results.

The simplest of non-linear material models is the elastic-perfectly plastic model. This model represents a material that has lost all its ability to return to original shape after deformation has been initiated. The stress remains constant over a certain value of strain. This model is a good description of most cast iron material. The stress-strain curve of the elastic-perfectly plastic model is shown in Figure 16, Ref /2/.

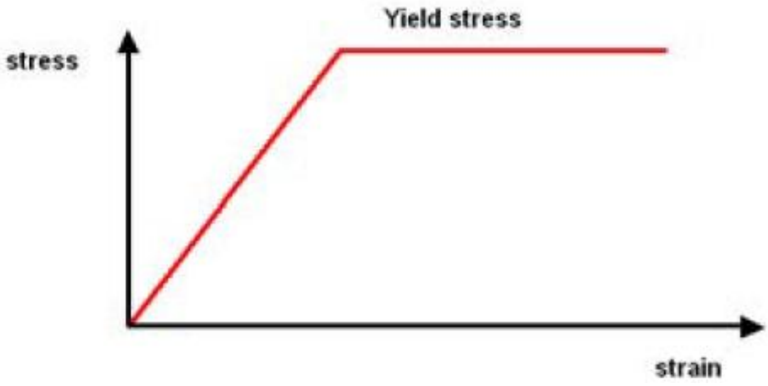


Figure 16: The elastic-perfectly plastic material model, Ref /2/.

1.2.6 Bulkhead analysis example

The following example is adopted from, Ref /2/ and used to illustrate the difference between FEA using the linear material model versus the elastic-perfectly plastic model.

The linear analysis using the material model presented in Figure 15 indicates a maximum von Mises stress of 614 MPa without considering the material yield strength of 206 MPa. So considering this analysis the bulkhead might fail. The result of the linear analysis is shown in Figure 17.

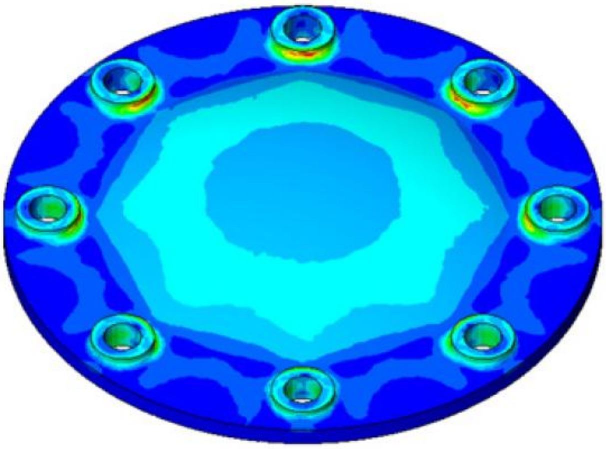


Figure 17: Linear solution of a bulkhead, Ref /2/.

The non-linear analysis using the material model presented in Figure 16 is used to find out how much of the material that consists of stresses in the plastic deformation area. The non-linear solution shows maximum stresses equal to yield stress. Plastic zones are still local which indicates that the bulkhead will not fall apart. The analysis result is shown in Figure 18.

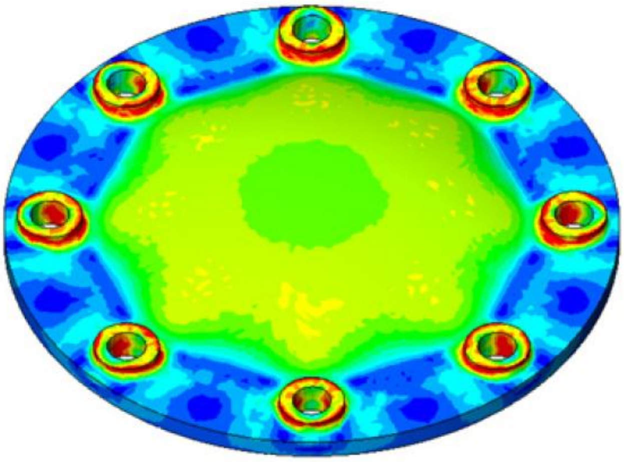


Figure 18: Non-linear solution using the elastic-perfectly plastic model, Ref /2/.

Careful engineering judgment is required to decide if the design is acceptable or if modifications are required.

Another example is the initial yielding of a cylinder with internal pressure, or a beam with increasing external loading. They have not failed until the formation of the plastic hinge is complete, or the entire cross section of the cylinder consists of stresses above the plastic limit as shown in Figure 19 and Figure 20, Ref /3/.

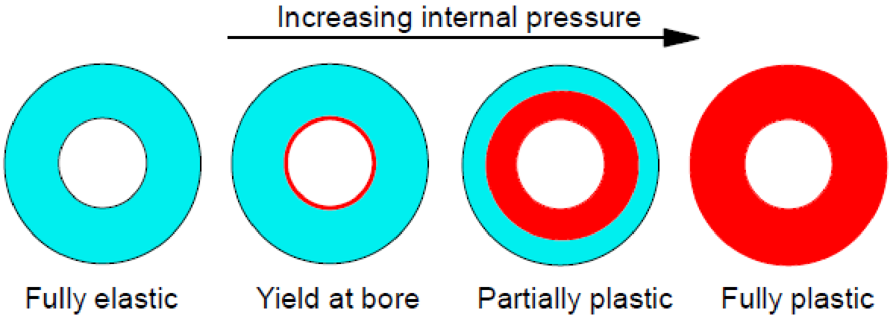


Figure 19: Cylinder with increasing internal pressure loading, Ref /3/.

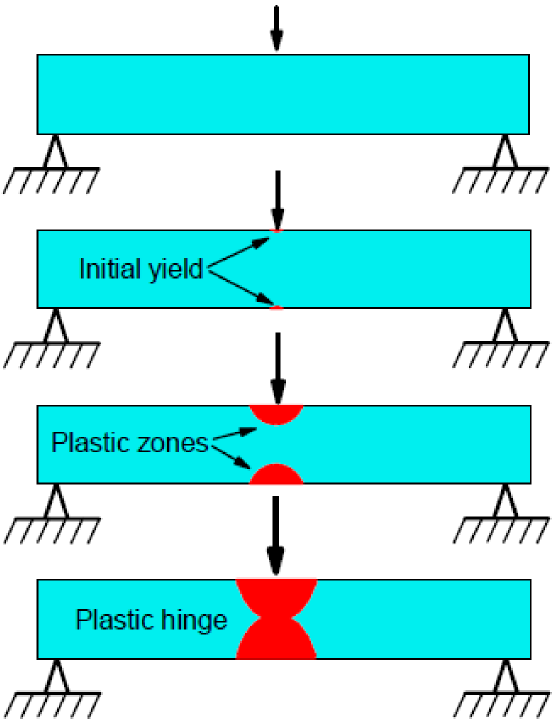


Figure 20: Beam with increasing external load, Ref /3/.

Again careful engineering judgment is called for to determine the acceptability criteria for these situations, which brings the problem statement of this thesis forward.

- When should the different standards and codes be used, and which code offers the most accurate description of the problem encountered?
- How should the results obtained from the different methods be interpreted?
- Can the use of different methods lead to serious design errors?

1.2.7 Serious design error example

The following example[3] presents the fatal failure of a brand new pressure vessel during the hydro-testing procedure. The vessel exploded during the test throwing metal pieces over a large area. Fortunately the test area was closed and no injuries occurred.

The root cause of this incident is at the moment not fully known but there are thoughts that hydro testing with “to cold water” was a contributing factor.

Questions regarding the design method used have also been noted, suggesting that non conservative results might have been overseen due to the “heavy wall construction”.

Specific recommendations

- Water temperature is critical when conducting hydro testing of pressure vessels. It is suggested by TOTAL that both metal and water temperature during pressure testing shall be maintained at least 10 °C above the impact test temperature of the metal.

Figure 21 shows pictures of the pressure vessel including the wall thickness, while Figure 22 shows a close up picture of the pressure vessel ruptured shell.

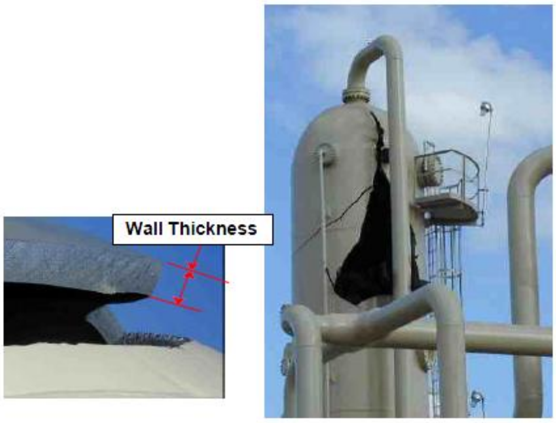


Figure 21: Pressure vessel failure and wall thickness.

Figure 22: Ruptured pressure vessel shell.

---

[3] The example is presented with permission from TOTAL.

1.3 Problem Statement

Several standards and codes are currently available for design and analysis of pressure vessels. Two of the main contributors are the American Society of Mechanical Engineers providing the ASME VIII rules for construction of pressure vessels, and the Technical Committee in Brussels providing the European Standard for unfired pressure vessels.

The ASME VIII code consists of three divisions covering different pressure ranges:

- Division 1: Up to 200 bar (3000 psi).
- Division 2: In general.
- Division 3: For pressure ranges above 690 bar (10000 psi).

Considering ASME division 2, Part 5 the following design by analysis methods are described:

- Elastic stress analysis method.
- Limit load analysis.
- Elastic-plastic analysis method.

The NS-EN 13445-3; 2009 provides two different methodologies for design by analysis, namely:

- The Direct route method.
- Method based on stress categories.

The method based on stress categories is similar to the Elastic Stress Analysis method from ASME VIII div.2.

Problems might arise when the various methods produces different answers to identical design problems.

When conducting design procedures a component could be designed:

- By rule.
- By classical interaction analysis.
- With an elastic FEA using a shell model.
- With an elastic FEA using a solid model and linearization procedures.
- By FEA limit analysis.
- By FEA plastic analysis.

Considering this list most designers might feel that the methods are presented in an order of increasing sophistication, indicating that they will reveal an increasing amount of information about the true structural behavior. One would therefore expect the more traditional methods to give conservative results.

However, experience from recent projects at IKM Ocean Design, as well as published research papers on the subject indicate that this might not be the case. Figure 23, Ref /4/ presents the results of an investigated pressure vessel design problem using various analysis methods.

Analysis Type	(Analysis Result) / (Design Limit) (> 1 fails)
WRC 107 and Section VIII Stress Index	0.97
Elastic Analysis, maximum PL	1.18
Averaged PL Elastic Analysis [8]	0.87
Limit Analysis	1.12
Plastic Analysis / von Mises	0.93

Figure 23: Margin of structure against various design limits, Ref /4/.

The maximum deviation between the results is about 30%.

- What should the designer or engineer do in this case?
- Should the traditional method (elastic analysis) be selected, knowing from the results above that it could be more conservative than a supposedly more “exact” one (plastic analysis)?
- Should a more sophisticated method be selected to improve the design even though traditionally designed vessels have performed satisfactorily in practice?

None of the existing codes or standards offers any guidance here.

1.4 Purpose and Scope

In this thesis differences between the different analysis methods will be investigated, and situations where there are advantages to use one type of analysis versus the other shall if possible be identified.

- **The result of an extensive literature review of the following methods, including both advantages and disadvantages shall be included for:**
    - Elastic Stress Analysis – ASME VIII div. 2; 2010
    - Elastic-plastic Analysis – ASME VIII div. 2; 2010
    - Direct Route – NS-EN 13445; 2009
- **A design basis for a typical pressure vessel shall be established at:**
    - 100 bar (thin wall construction)
    - 200 bar (heavy wall construction)
    - The pressure vessel to be used in the study shall be generic. A cylindrical tank with a nozzle can be a good option.
- **Calculations using the different methods and different available analysis programs such as:**
    - ANSYS Workbench.
    - Visual Vessel Design.
- Comparison of design **results** for both heavy wall and thin wall pressure vessels using NS-EN 13445; 2009 Direct Route and ASME VIII div.2; 2010 Elastic-plastic Analysis.
    - Evaluate the suitability of Elastic Stress Analysis for a heavy wall pressure vessel.
- **Recommendations** on Design by Analysis methods for different applications shall be included in this thesis.
- Present “A simplified guide to pressure vessel design”.

CHAPTER 2 METHODS

2.1 Direct Route – NS-EN 13445; 2009

2.1.1 General information

The Design by Analysis – Direct Route (DBA-DR) included in EN-13445-3:2002, Ref /6/ establishes a set of design rules for any component under any action.

It might be used:

- As an alternative to the common Design by Formulas route (DBF).
- For cases that are not covered by the DBF route, such as for superposition of environmental actions.
- For cases where the quality related manufacturing tolerances are exceeded.
- When the local authorities, due to a high risk project require more detailed investigations.

Main advantages of the DBA-DR route, Ref/7/ are:

- The elimination of the problems associated with the stress categorization route.
- Direct addressing of the failure modes thus better insight into the critical failure modes and the relevant safety margins. This might improve the design philosophy.
- Direct assessment of other actions than pressure, such as thermal and environmental effects.

Disadvantages of the DBA-DR route, Ref /7/:

- Non-linear calculations are required and this influences the calculation time directly.
- Linear superposition is not possible anymore.
- The method requires good knowledge of the underlying theories.

With the computer software available today it is quite easy to obtain Finite Element results, but obtaining reasonable correct results is not that easy. The process of setting up the model and defining the relevant boundary conditions requires experience and good knowledge of the theory of structures.

The DBA-DR requires a great amount of expertise, and it should therefore be used with care. Because of the advanced approach the following warning remark was introduced in the standard EN-13445-3 Annex B, Section B1, Ref /6/.

"Due to the advanced methods applied, until sufficient in-house experience can be demonstrated, the involvement of an independent body, appropriately qualified in the field of DBA, is required in the assessment of the design (calculations) and the potential definition of particular NDT requirements."

All checks considered in this thesis are for normal operating loads. For testing loads reference is made to EN-13445-3:2002 Appendix A, Ref /6/.

2.1.2 Notations

A clear distinction between principle and application rules is given in the standard, Ref /6/. Principles consist of general statements, definitions and requirements that are absolute, thus no alternative is permitted. The application rules on the other hand are generally recognized rules, where alternatives are allowed provided that the alternative can be demonstrated and verified to be in confirmation with the principle.

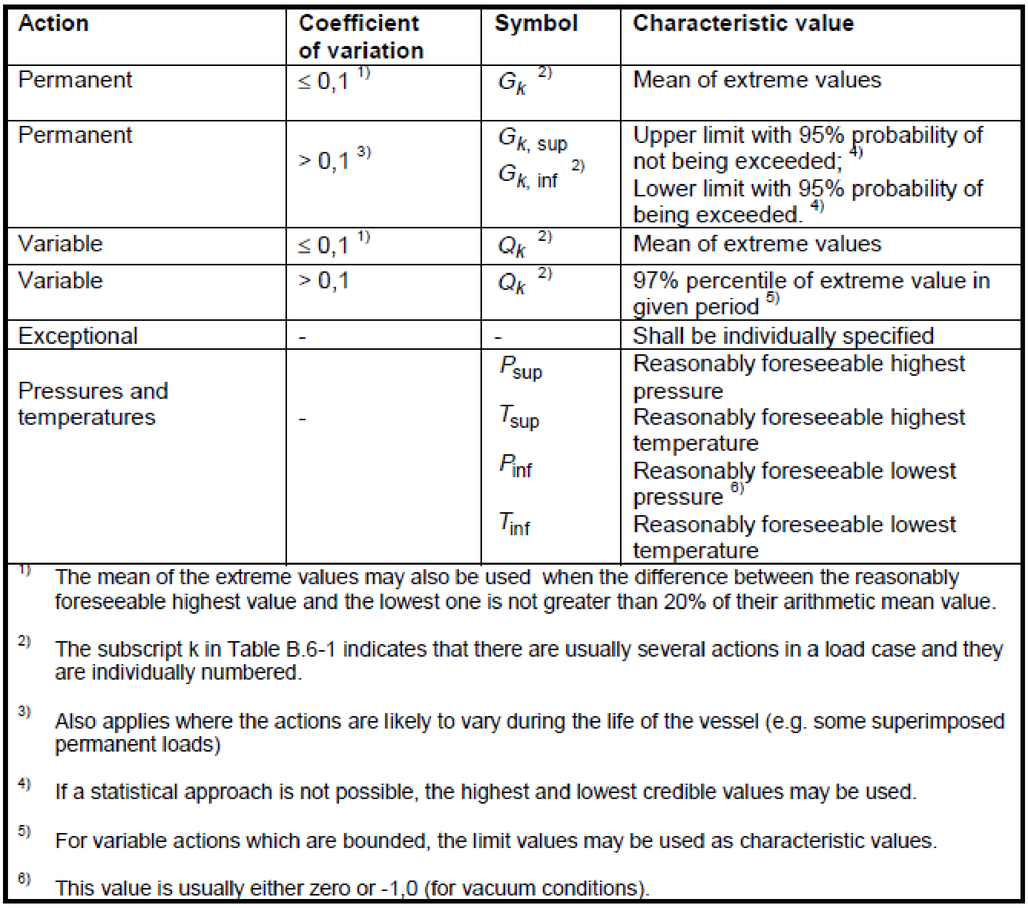
The term action denotes all quantities imposed on the structure that causes stress or strain, like forces, pressure, displacement and temperature.

The action types are classified by their variation in time:

- Permanent actions ***(G)***
- Variable actions ***(Q)***
- Exceptional actions ***(E)***
- Operating pressures and temperatures ***(p,T)***

There is usually a strong correlation between pressure and temperature. Therefore they shall be considered to act simultaneously.

Figure 24 illustrates the characteristic values of the different action types as they were presented in table B.6-1, Ref /6/.



Action	**Coefficient of variation**	**Symbol**	**Characteristic value**
Permanent	≤ 0,1 [1]	G_k [2]	Mean of extreme values
Permanent	> 0,1 [3]	$G_{k,\text{ sup}}$ $G_{k,\text{ inf}}$ [2]	Upper limit with 95% probability of not being exceeded; [4] Lower limit with 95% probability of being exceeded. [4]
Variable	≤ 0,1 [1]	Q_k [2]	Mean of extreme values
Variable	> 0,1	Q_k [2]	97% percentile of extreme value in given period [5]
Exceptional	-	-	Shall be individually specified
Pressures and temperatures	-	P_{sup} T_{sup} P_{inf} T_{inf}	Reasonably foreseeable highest pressure Reasonably foreseeable highest temperature Reasonably foreseeable lowest pressure [6] Reasonably foreseeable lowest temperature

[1] The mean of the extreme values may also be used when the difference between the reasonably foreseeable highest value and the lowest one is not greater than 20% of their arithmetic mean value.

[2] The subscript k in Table B.6-1 indicates that there are usually several actions in a load case and they are individually numbered.

[3] Also applies where the actions are likely to vary during the life of the vessel (e.g. some superimposed permanent loads)

[4] If a statistical approach is not possible, the highest and lowest credible values may be used.

[5] For variable actions which are bounded, the limit values may be used as characteristic values.

[6] This value is usually either zero or -1,0 (for vacuum conditions).

Figure 24: Characteristic values for different types of actions, Ref /6/.

The coefficient of variation is a statistical measure of the dispersion of data points in a data series around the mean value. It is defined as:

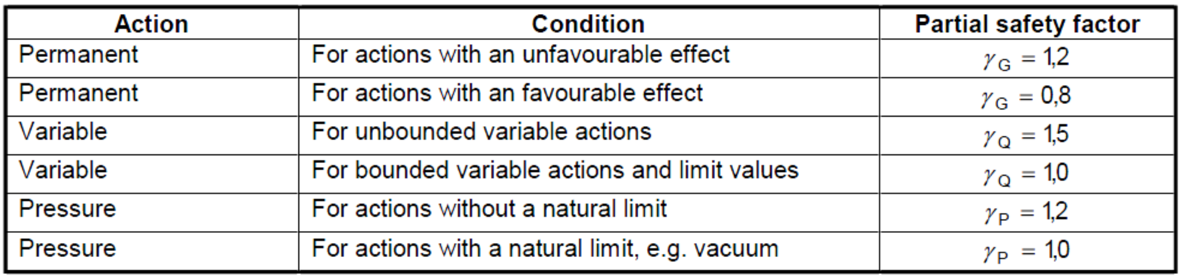
$$\boldsymbol{Coefficient\ of\ variation = \frac{Standard\ deviation}{Mean\ value}}$$

The characteristic values identified are used to establish the design values of the different action effects.

2.1.3 Partial Safety factors

The design values for actions are obtained by multiplying the characteristic values with partial safety factors. The partial safety factors of the actions depend on the action type, and whether the action is favorably or non-favorably. Favorable effect is when the action in a given load case acts opposite to the governing action, e.g. weight acting opposite to pressure. If the governing action is not obvious separate load cases are required.

The partial safety factors for actions and normal operations from table B.8-1, Ref /6/, are presented in Figure 25.

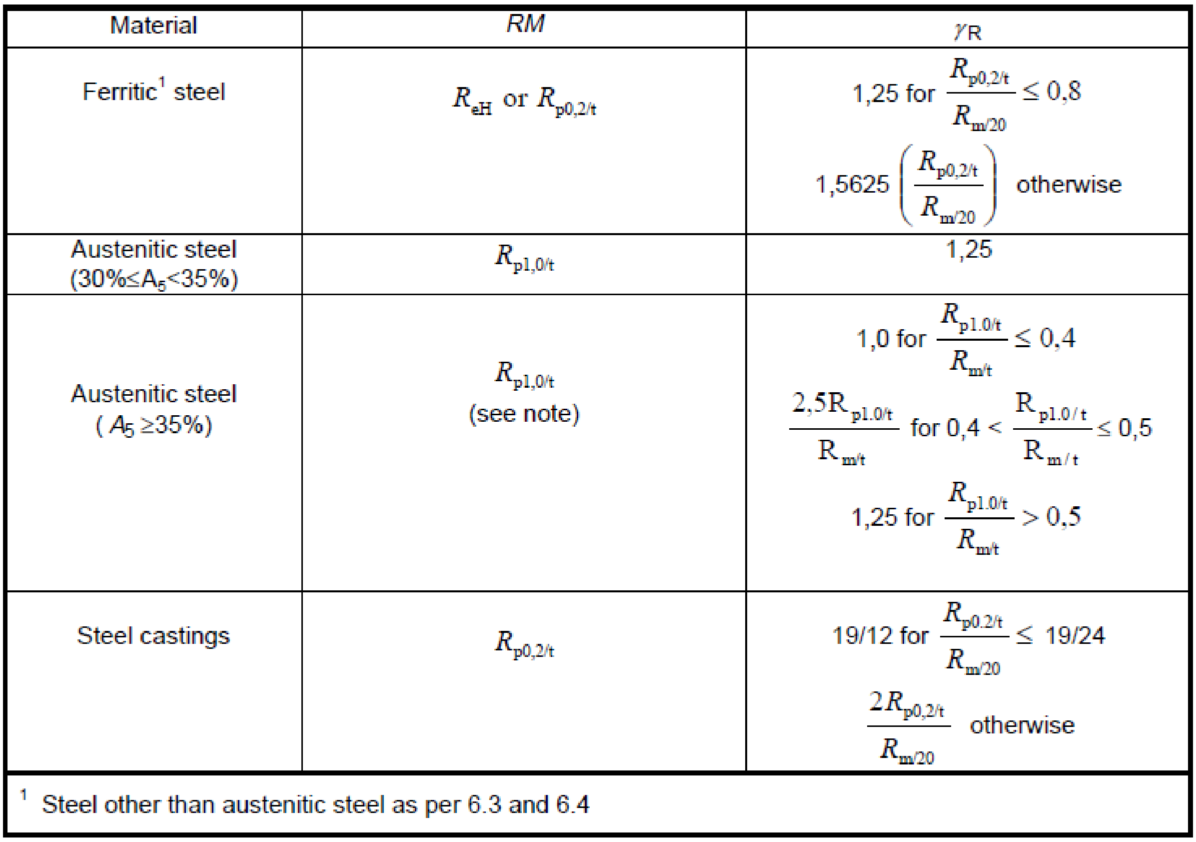


Action	Condition	Partial safety factor
Permanent	For actions with an unfavourable effect	γ_G = 1,2
Permanent	For actions with an favourable effect	γ_G = 0,8
Variable	For unbounded variable actions	γ_Q = 1,5
Variable	For bounded variable actions and limit values	γ_Q = 1,0
Pressure	For actions without a natural limit	γ_P = 1,2
Pressure	For actions with a natural limit, e.g. vacuum	γ_P = 1,0

Figure 25: Partial safety factors for actions and normal operation cases, Ref /6/.

The design values for material resistance are obtained by dividing the material strength parameter with the relevant partial safety factor. The partial safety factors for material depend on the material type, the dispersion in material parameters, uncertainties of the relationship between material test parameters and those materials to be used in the real structure.

The partial safety factors for different material types under normal operations and load cases from table B.8-2, Ref /6/, are presented in Figure 26.



Material	*RM*	γ_R
Ferritic[1] steel	R_{eH} or $R_{p0,2/t}$	1,25 for $\frac{R_{p0,2/t}}{R_{m/20}} \le 0,8$ $1,5625\left(\frac{R_{p0,2/t}}{R_{m/20}}\right)$ otherwise
Austenitic steel (30%≤A_5<35%)	$R_{p1,0/t}$	1,25
Austenitic steel (A_5 ≥35%)	$R_{p1,0/t}$ (see note)	1,0 for $\frac{R_{p1.0/t}}{R_{m/t}} \le 0,4$ $\frac{2,5R_{p1.0/t}}{R_{m/t}}$ for $0,4 < \frac{R_{p1.0/t}}{R_{m/t}} \le 0,5$ 1,25 for $\frac{R_{p1.0/t}}{R_{m/t}} > 0,5$
Steel castings	$R_{p0,2/t}$	19/12 for $\frac{R_{p0.2/t}}{R_{m/20}} \le$ 19/24 $\frac{2R_{p0,2/t}}{R_{m/20}}$ otherwise

[1] Steel other than austenitic steel as per 6.3 and 6.4

Figure 26: Partial safety factors for different material types used for pressure vessels, Ref /6/.

Note: The deformations at this material strength may be very large for austenitic steels, and it is therefore advisable to check for leakages at bolted connections.

Considering the action types, the characteristic values of each action and the partial safety factors for the given action type, the design effects are calculated. The design effects are usually the stresses in the structure.

The expression for design effects, Ref /7/:

$$E_d = E(\gamma_P \times P, \gamma_G \times G, \gamma_Q \times Q \ldots)$$

Considering the material strength parameter and its corresponding partial safety factor the expression for design resistance is obtained.

The expression for design resistance, Ref /7/:

$$R_d = R(\frac{RM}{\gamma_R})$$

2.1.4 Design checks

Design checks are investigations of the construction`s safety when subjected to specified combinations of actions with respect to different limit states. A limit state is defined as a structural condition beyond the point where the design requirements are satisfied. The limit states are divided into ultimate limit state and serviceability limit state.

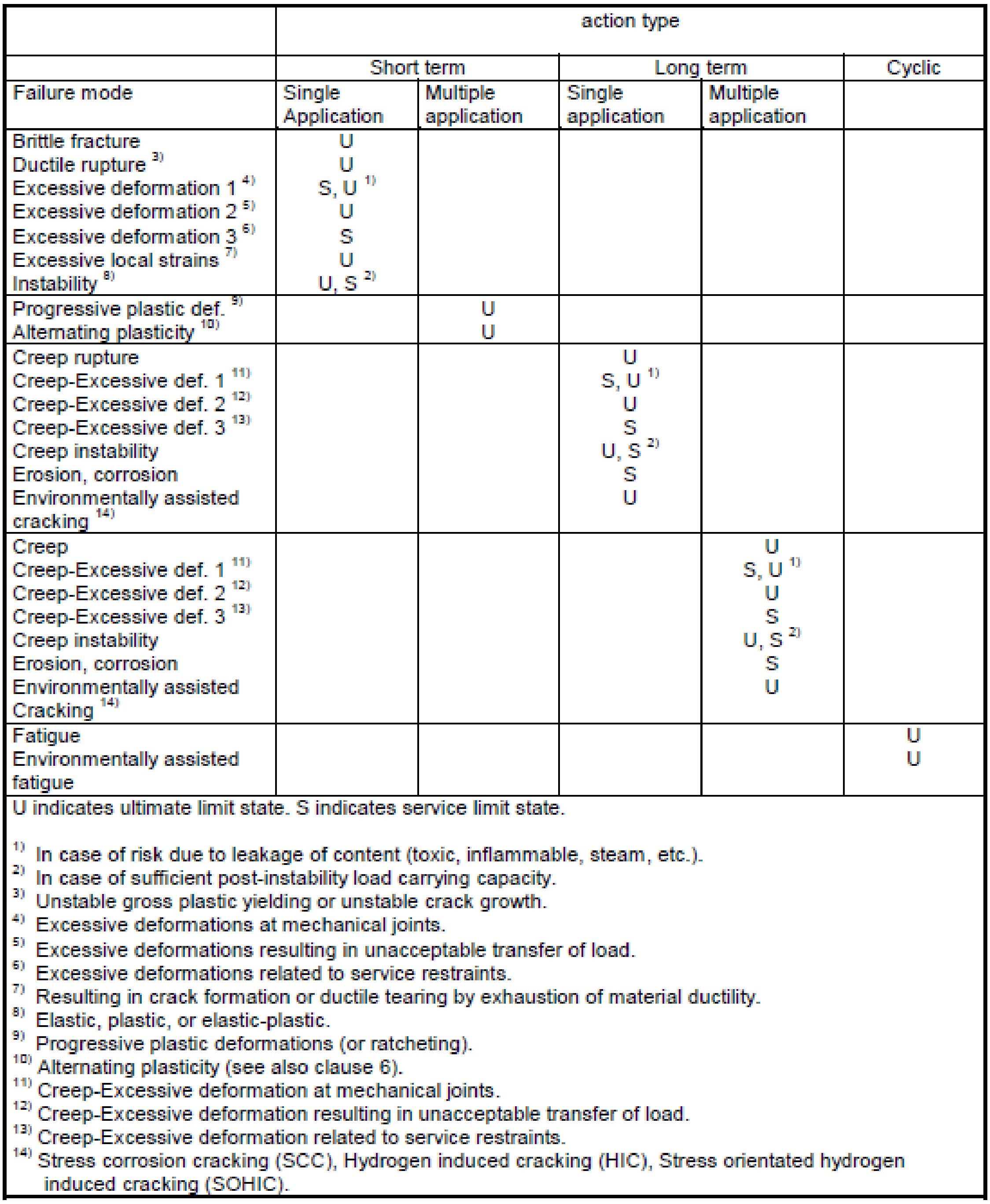
Ultimate limit states as defined in, Ref /6/:

- Failure by gross plastic deformation.
- Rupture caused by fatigue.
- Collapse caused by instability.
- Loss of equilibrium.
- Overturning or displacement of a rigid body.
- Leakage affecting safety.

Serviceability limit states as defined in, Ref /6/:

- Deformation or deflection which affects the normal use of the vessel, or causes damage to structural or non-structural elements.
- Leakage which affects efficient use of the vessel but does not compromise safety or causes unacceptable environmental damage.

The failure modes considered are listed in Table B.4-1, Ref /6/ and presented in Figure 27.



	action type				
	Short term		Long term		Cyclic
Failure mode	Single Application	Multiple application	Single application	Multiple application	
Brittle fracture	U				
Ductile rupture 3)	U				
Excessive deformation 1 4)	S, U 1)				
Excessive deformation 2 5)	U				
Excessive deformation 3 6)	S				
Excessive local strains 7)	U				
Instability 8)	U, S 2)				
Progressive plastic def. 9)		U			
Alternating plasticity 10)		U			
Creep rupture			U		
Creep-Excessive def. 1 11)			S, U 1)		
Creep-Excessive def. 2 12)			U		
Creep-Excessive def. 3 13)			S		
Creep instability			U, S 2)		
Erosion, corrosion			S		
Environmentally assisted cracking 14)			U		
Creep				U	
Creep-Excessive def. 1 11)				S, U 1)	
Creep-Excessive def. 2 12)				U	
Creep-Excessive def. 3 13)				S	
Creep instability				U, S 2)	
Erosion, corrosion				S	
Environmentally assisted Cracking 14)				U	
Fatigue					U
Environmentally assisted fatigue					U

U indicates ultimate limit state. S indicates service limit state.

1) In case of risk due to leakage of content (toxic, inflammable, steam, etc.).
2) In case of sufficient post-instability load carrying capacity.
3) Unstable gross plastic yielding or unstable crack growth.
4) Excessive deformations at mechanical joints.
5) Excessive deformations resulting in unacceptable transfer of load.
6) Excessive deformations related to service restraints.
7) Resulting in crack formation or ductile tearing by exhaustion of material ductility.
8) Elastic, plastic, or elastic-plastic.
9) Progressive plastic deformations (or ratcheting).
10) Alternating plasticity (see also clause 6).
11) Creep-Excessive deformation at mechanical joints.
12) Creep-Excessive deformation resulting in unacceptable transfer of load.
13) Creep-Excessive deformation related to service restraints.
14) Stress corrosion cracking (SCC), Hydrogen induced cracking (HIC), Stress orientated hydrogen induced cracking (SOHIC).

Figure 27: Classification of failure modes and limit states, Ref /6/.

To each failure mode, there exists a single design check (DC). The design checks are named after the main failure mode they deal with.

Design checks to be considered are, Ref /6/:

- Gross Plastic Deformation Design Check (GPD-DC).
- Progressive Plastic Deformation Design Check (PD-DC).
- Instability Design Check (I-DC).
- Fatigue Design Check (F-DC).
- Static Equilibrium Design Check (SE-DC).

2.1.5 Procedure

1. All design checks listed above shall be considered. Thus the failure mode and corresponding limit state of the construction must be identified.
2. All relevant load cases shall be considered for each design check.
3. For each design check appropriate application rule shall be selected, or the principle shall be used directly. If the principle is not satisfied, repeat the design check with different loading, geometry or material.
4. For each design check the fulfillment of the principle shall be shown.
   a. Specification of design check, load case and corresponding actions.
   b. Determination of the characteristic values of the actions.
   c. Calculation of the design values for the given action.
   d. Calculate the effect of the actions.
   e. Calculate the resistance of the component.
   f. Check the fulfillment of the principle.
   g. Statement confirming if the principle is fulfilled for the load case considered.

2.1.6 Gross Plastic Deformation (GPD)

Nominal values shall be used for all dimensions with the exception of thickness. For thickness effective value shall be used. That is the nominal value minus allowance for material tolerances, allowance for possible thinning during manufacture, and corrosion allowance.

Principle

For each load case, the design value of an action shall be carried entirely by the design model using:

- A linear-elastic ideal-plastic constitutive law.
- The Tresca yield condition (maximum shear stress condition) and associated flow rule.
  - The flow rule determines the relationship between stress and plastic strain. In other words it describes how the material behavior is beyond the yield point.
  - Von Mises` condition might be used, but then the design strength parameter shall be multiplied by $\sqrt{3}/_2$.
- Material strength parameters and partial safety factors as given in Figure 26.
- Maximum absolute value of the principal structural strains for proportional increase of all actions and a stress free initial state shall be less than:
  - ***5%*** in normal operating load cases.
  - ***7%*** in testing load cases.
- In exceptional load cases this strain limitation requirement does not apply.

Application rule

The Lower bound limits approach state that if it can be verified that any lower bound limit value of the applied action, determined with the design model specified in the principle, is reached without violating the strain limit the principle is considered fulfilled.

Problems

The use of Tresca`s yield condition in the constitutive law of the design model poses a problem for usual soft- and hardware as they do not provide for elasto-plastic calculations with that criteria. On the other hand experience has shown that the usage of Von Mises` yield condition with the reduced design yield strength gives quite acceptable results, Ref /9/.

2.1.7 Progressive Plastic Deformation (PD)

The design model established for the GPD design check may also be used in the PD design check. Characteristic values of permanent actions and combinations of temperature/pressure shall be combined with the most unfavorable variable action in the action cycle.

Principle

For all relevant load cases applied to the model in specified repeated action cycles no progressive plastic deformation shall occur for:

- First order theory.
- A linear-elastic ideal-plastic constitutive law.
- Von Mises` yield condition (maximum distortion energy criterion) and associated flow rule.
- Material design strength parameters as given in Figure 26 at a temperature which shall not be less than $\mathbf{0{,}75 \times t_{c\,max} + 0{,}25 \times t_{c\,min}}$ (lowest and highest temperature at the position during the action cycle)
- All partial safety factors equal to unity ***(1)*** in this design check.

Application rule 1

The principle is considered fulfilled if it can be verified that the maximum absolute value of the principal structural strains is less than 5% after the application of the number of cycles specified for the considered load case. If the number of cycles is not specified, then a reasonable number, no less than 500 cycles, shall be assumed.

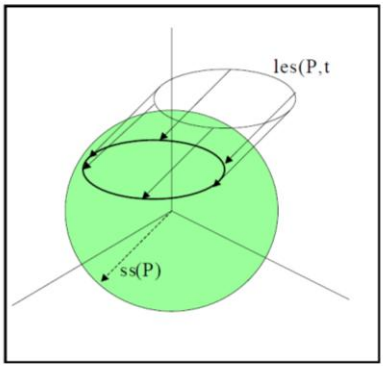
Application rule 2

According to the shakedown approach the principle is considered fulfilled if the structure shakes down to linear elastic behavior, or purely alternating plasticity. Two simple and effective tools are available for the shakedown approach, the Melan`s Shakedown theorem Ref /9/, and the Deviatoric map, Ref /10/.

The Deviatoric map is a map of the stress tensor, represented as an isometric view of the principal stress vector in the principal stress space. It illustrates a structure`s behavior at specific points with regard to ideal plasticity.

A problem worth mentioning is that some of the information value are lost if the principal stress directions change during the considered action cycle.

The Deviatoric map is shown in Figure 28, Ref /10/.



The coordinate system axis are defined as the principal stresses;

- $\sigma_1, \sigma_2, \sigma_3$

Stress caused by constant action;

- $ss(P)$

Stress caused by time varying cyclic action;

- $les(P, t)$

Figure 28: The Deviatoric map, Ref /10/.

Melan`s shakedown theorem reads, Ref /9/:

- A structure will shake down to linear elastic behavior under a given action cycle, if there exists a time-invariant self-equilibrating stress field $\boldsymbol{\sigma_{ij}^*(P)}$ such as **the sum of this** stress field and where the stress field $\boldsymbol{\sigma_{ij}^E(P,t)}$**,** determined with the linear elastic constitutive law for the cyclic action, nowhere and at no time exceeds the material yield strength.

The theorem sounds complicated, but the difference between the stress field calculated for a specific action with the elasto-plastic constitutive law and the stress field for the same action but for the linear-elastic law is in fact ,very convenient, a self-equilibrating stress field.

The self-equilibrating stress field is self-equilibrating in the sense that it fulfils all internal and external equilibrium conditions such as vanishing imposed forces and all external forces which are not reactions. In other words for all forces which contribute to work in a virtual deformation. Reaction forces due to kinematic boundary conditions may be different from zero. Only the forces in points with dynamic boundary conditions, where surface forces are specified, must be zero for a stress field to be self-equilibrating.

Different material behavior with respect to stress and strain is illustrated in Figure 29, Ref /11/.

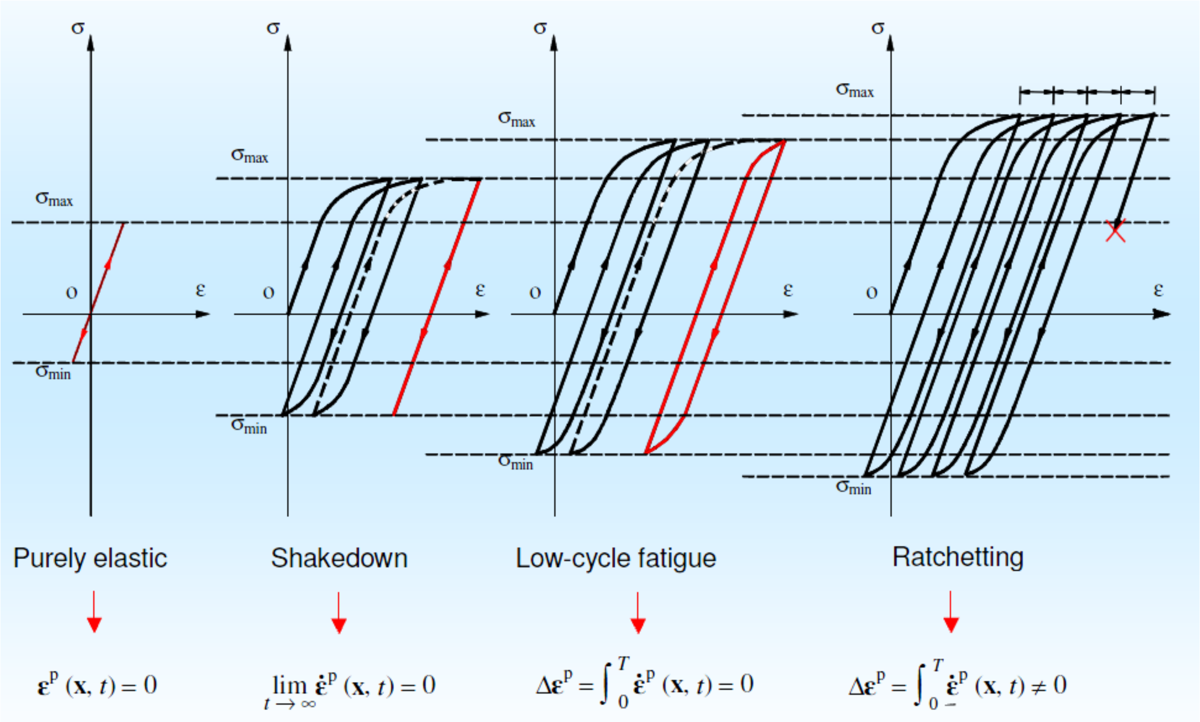


Figure 29: Material behavior considering stress versus strain, Ref /11/.

- Purely elastic behavior.
    - No plastic deformation.
- Shakedown.
    - Structure shows elastic behavior after a given number of cycles.
- Low-cycle fatigue or alternating plasticity.
    - Consider the risk of failure due to fatigue.
- Ratchetting.
    - Material failure after a given number of cycles. Ratchetting is not permitted in pressure vessel design according to the standard.

2.1.8 Instability (I)

Principle

For each considered load case, the design value of an action or of a combination of actions shall not exceed the design value of the corresponding buckling strength. The value of the principal structural strains shall not exceed 5%.

The design model shall be according to:

- Pre-deformation according to EN 13445-4:2002 or per specification on the drawings.
- A linear-elastic ideal plastic constitutive law.
- Von Mises`s yield condition and associated flow rule.
- Design strength parameters and partial safety factors as given in Figure 26.
- Partial safety factors for actions as given in Figure 25.
  - Partial safety factor for temperature action shall be equal to unity (1).
  - All actions shall be applied with proportional increase.
- Stress-free initial state.
- The design value for buckling strength shall be determined by division by the safety factor γ_R.
  - 1, 25 if pressure test is to be carried out.
  - 1, 5 otherwise.

Application rule 1

This rule may be used if experimental results for the specific load cases are available.

The principle is considered fulfilled if the design value of an action is below a lower bound limit of the range of failure values based on experimental observations.

Application rule 2

Fulfillment of all the other design checks mentioned here is sufficient as a stability check for pure pressure action.

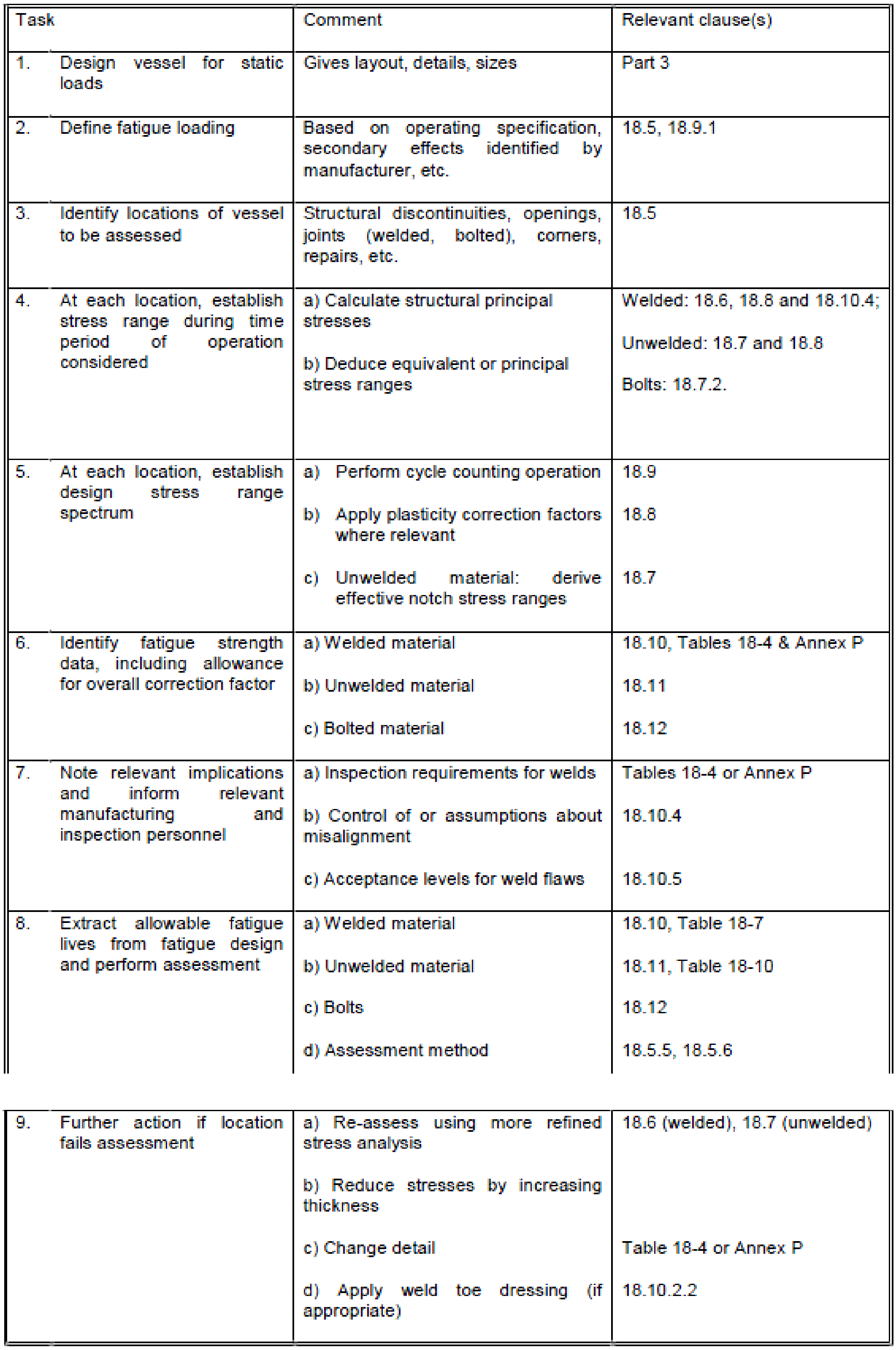
2.1.9 Fatigue failure (F)

Principle

The design value of the damage indicator obtained for the design function of pressure/temperature and variable actions shall not exceed 1.

Application rule

The requirements given in NS-EN 13445-3, section 18 are sufficient as a check against failure. A summary of the requirements are presented in Figure 30, Ref /6/. As an additional requirement, cladding should be considered with respect to the thermal analysis and the stress analysis. If the cladding thickness is less than 10% of the total thickness of the component it can be neglected and the model can be simplified to base metal geometry.



Task	Comment	Relevant clause(s)
1. Design vessel for static loads	Gives layout, details, sizes	Part 3
2. Define fatigue loading	Based on operating specification, secondary effects identified by manufacturer, etc.	18.5, 18.9.1
3. Identify locations of vessel to be assessed	Structural discontinuities, openings, joints (welded, bolted), corners, repairs, etc.	18.5
4. At each location, establish stress range during time period of operation considered	a) Calculate structural principal stresses b) Deduce equivalent or principal stress ranges	Welded: 18.6, 18.8 and 18.10.4; Unwelded: 18.7 and 18.8 Bolts: 18.7.2.
5. At each location, establish design stress range spectrum	a) Perform cycle counting operation b) Apply plasticity correction factors where relevant c) Unwelded material: derive effective notch stress ranges	18.9 18.8 18.7
6. Identify fatigue strength data, including allowance for overall correction factor	a) Welded material b) Unwelded material c) Bolted material	18.10, Tables 18-4 & Annex P 18.11 18.12
7. Note relevant implications and inform relevant manufacturing and inspection personnel	a) Inspection requirements for welds b) Control of or assumptions about misalignment c) Acceptance levels for weld flaws	Tables 18-4 or Annex P 18.10.4 18.10.5
8. Extract allowable fatigue lives from fatigue design and perform assessment	a) Welded material b) Unwelded material c) Bolts d) Assessment method	18.10, Table 18-7 18.11, Table 18-10 18.12 18.5.5, 18.5.6
9. Further action if location fails assessment	a) Re-assess using more refined stress analysis b) Reduce stresses by increasing thickness c) Change detail d) Apply weld toe dressing (if appropriate)	18.6 (welded), 18.7 (unwelded) Table 18-4 or Annex P 18.10.2.2

Figure 30: Summary of the fatigue assessment process for pressure vessels according to Ref /6/.

2.1.10 Static equilibrium (SE)

Principle

The design effect of the destabilizing actions shall be smaller than the design effect of the stabilizing actions.

- Partial safety factors for actions as given in Figure 25.
- Stabilizing actions shall be represented by lower design value, while destabilizing actions shall be represented by highest design value.
- Self-weights of construction elements shall be treated as separate permanent actions.
- Favorable effects of a variable action shall not be considered.

2.2 Elastic Stress Analysis – ASME VIII div. 2; 2010

2.2.1 General information

In the elastic stress analysis, the material is assumed to be linear-elastic. The plastic hardening of the material is not considered. This method is also known as “the stress categorization method” due to the classification procedure of the elastic stresses. The stresses are classified into the primary stress, secondary stress and peak stress.

The stresses are limited to allowable values that have been established such that plastic collapse will not occur. The requirements given in the code are based on protection against different failure modes.

The failure modes considered are, Ref /12/:

- Protection against plastic collapse.
- Protection against local failure.
- Protection against collapse from buckling.
- Protection against failure from cyclic loading or fatigue.
    - Ratcheting assessment.

Disadvantages of the elastic stress analysis method, Ref /14/:

- The major problem is the stress categorization. The stresses must be defined and sorted in their respective categories.
- The linearization of stresses over certain regions of the pressure vessel has no theoretical justification.
- In areas where the stress distribution is complex the designer might have problems assigning the stresses to the appropriate category as significant knowledge and expert judgment are required.
- The method cannot be used for geometries with significant non-linear variation in through-thickness stress. (E.g. thick/heavy-wall pressure vessels).

Section 5.2.1.3, Ref /12/ states:

“The use of elastic stress analysis combined with stress classification procedures to demonstrate structural integrity for heavy-wall $(\boldsymbol{R}/\boldsymbol{t} \leq \mathbf{4})$ *pressure containing components may produce non-conservative results and is not recommended”.*

Advantages of the elastic stress analysis method:

- A useful aspect of linear elastic analysis is that superposition of combined action effects are allowed.
- The method is widely used in the pressure vessel industry and is therefore well known and accepted.
- ASME VIII div. 2 offers a detailed and easy to follow approach for use of this method.
- For design purposes quick calculation time is often desired and the use of this method reduces the calculation times significantly.

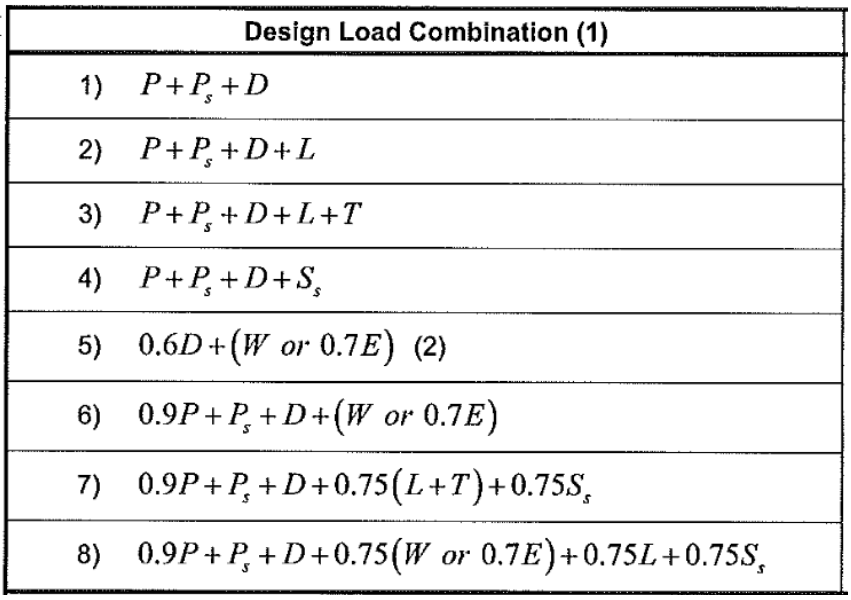
2.2.2 Loading conditions

All loads applied on the component shall be considered in the analysis. Combination of loads and loads that are varying in time must also be considered. For time varying loads a load histogram shall be developed to show the time variation of each specific load.

For normal operation the following load cases are considered:

- Dead weight of the vessel, content, and appurtenances at the location of interest ***(D).***
- Live loading including the effect of fluid momentum ***(L).***
- Internal and external specified design pressure ***(P).***
- Static head from liquid or bulk materials ***(P_S).***
- Earthquake loads ***(E).***
- Wind loads ***(W).***
- Snow loads ***(S_S).***
- Self-restraining loads like thermal loads and applied displacements ***(T).***

Suggested load case combinations are provided in the code, Ref /12/ and presented in Figure 31.



	Design Load Combination (1)
1)	$P+P_s+D$
2)	$P+P_s+D+L$
3)	$P+P_s+D+L+T$
4)	$P+P_s+D+S_s$
5)	$0.6D+(W \text{ or } 0.7E)$ (2)
6)	$0.9P+P_s+D+(W \text{ or } 0.7E)$
7)	$0.9P+P_s+D+0.75(L+T)+0.75S_s$
8)	$0.9P+P_s+D+0.75(W \text{ or } 0.7E)+0.75L+0.75S_s$

Figure 31: Load case combinations, Ref /12/.

2.2.3 Material properties

Physical properties like; Young`s modulus, thermal expansion coefficient, thermal conductivity, thermal diffusivity, density and Poisson`s ratio are found in Part 3, Ref /12/

Strength parameters like allowable stress ***(S),*** minimum specified yield strength and minimum specified tensile strength are found in ASME II Part D; 2007, Boiler and Pressure Vessel Code, Material properties, Ref /13/.

2.2.4 Protection against plastic collapse

To evaluate a structure`s protection against plastic collapse using the elastic stress analysis method, the results from an elastic analysis of the component subjected to the loading conditions given above are categorized and compared to a limiting value.

The equivalent stress must be calculated at selected locations in the component and compared to allowable values to determine if the component is suitable for intended design conditions.

The von Mises` (maximum distortion energy) yield criterion shall be used to establish the equivalent stress:

$$\boldsymbol{S_e = \sigma_e = \frac{1}{\sqrt{2}}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]^{0.5}}$$

The use of elastic stress analysis method for structural evaluation against plastic collapse provides an approximated value. For more accurate estimates the elastic-plastic analysis method should be used to develop limits and plastic collapse loads.

Stress categories

The tree basic primary categories and limits that are to be satisfied for plastic collapse are defined as, Ref /12/:

- General Primary Membrane equivalent stress ***(P_M).***
    - Average stress across the thickness of a section.
    - Excluding discontinuities and stress concentrations.
    - Produced by the design internal pressure and other specified mechanical loads.
- Local Primary Membrane equivalent stress ***(P_L).***
    - Average stress across any solid section.
    - Produced by mechanical loads.
    - Including discontinuities and excluding stress concentrations.
    - A region is considered local if the distance where the equivalent stress exceeds $\boldsymbol{1,1 \times S}$ is more than $\boldsymbol{\sqrt{\mathbf{R} \times \mathbf{t}}}$.
    - Regions of local primary membrane stress that exceeds $\boldsymbol{1.1 \times S}$ shall be separated by a distance greater than $\boldsymbol{1,25 \times \sqrt{(\mathbf{R_1} + \mathbf{R_2})(\mathbf{t_1} + \mathbf{t_2})}}$.
- Primary membrane (general or local) plus primary bending equivalent stress ***(P_L+P_b).***
    - Derived from the highest value across the thickness of a section.
    - Obtained by superposition of the membrane and bending stresses.
    - The decomposition of a nonlinear stress field into equivalent membrane stress and equivalent bending stress is shown graphically in Figure 32, Ref /14/.

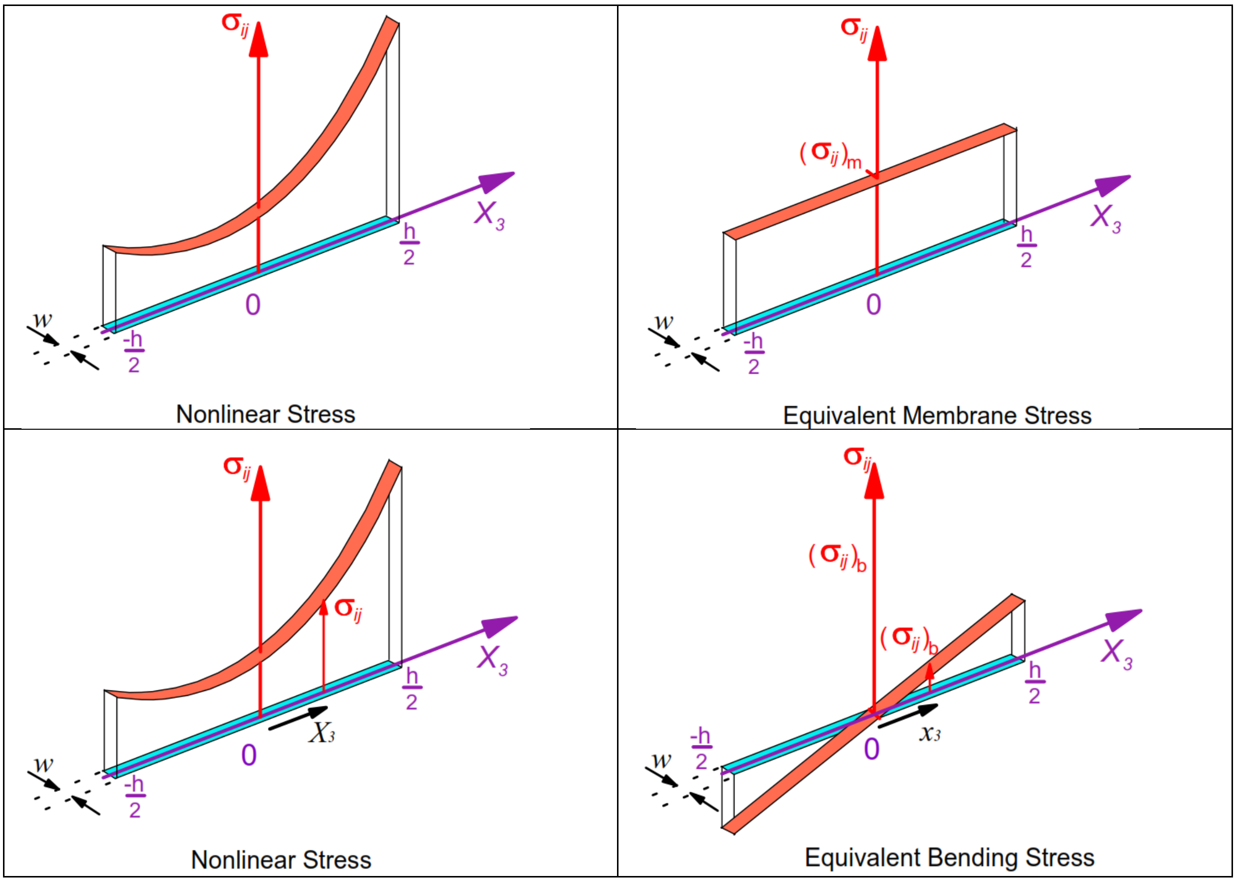


Figure 32: Decomposition of a nonlinear stress field, Ref /14/.

The procedure mentioned above is presented mathematically in section 5.A.4.1.2, Ref /12/. The numerical work required to perform this by hand is beyond the scope of this thesis but a graphical example is provided for the reader to understand the required procedure.

The numerical calculations are to be conducted for each value of x (i.e. distance through the cross section along the stress classification line). The results as a function of cross section thickness are plotted in a graph as illustrated in Figure 33, Ref /14/.

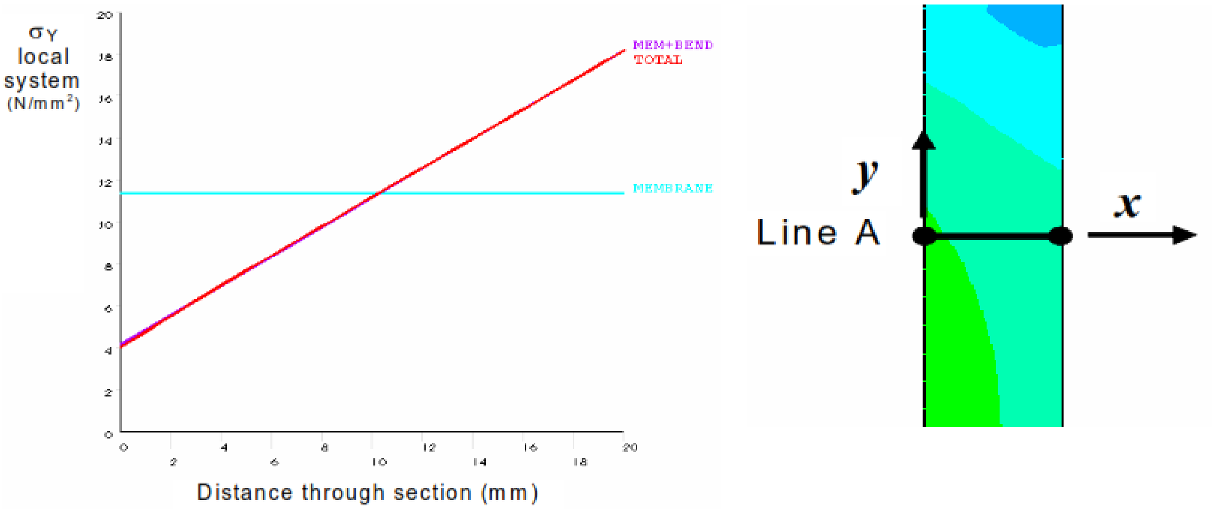
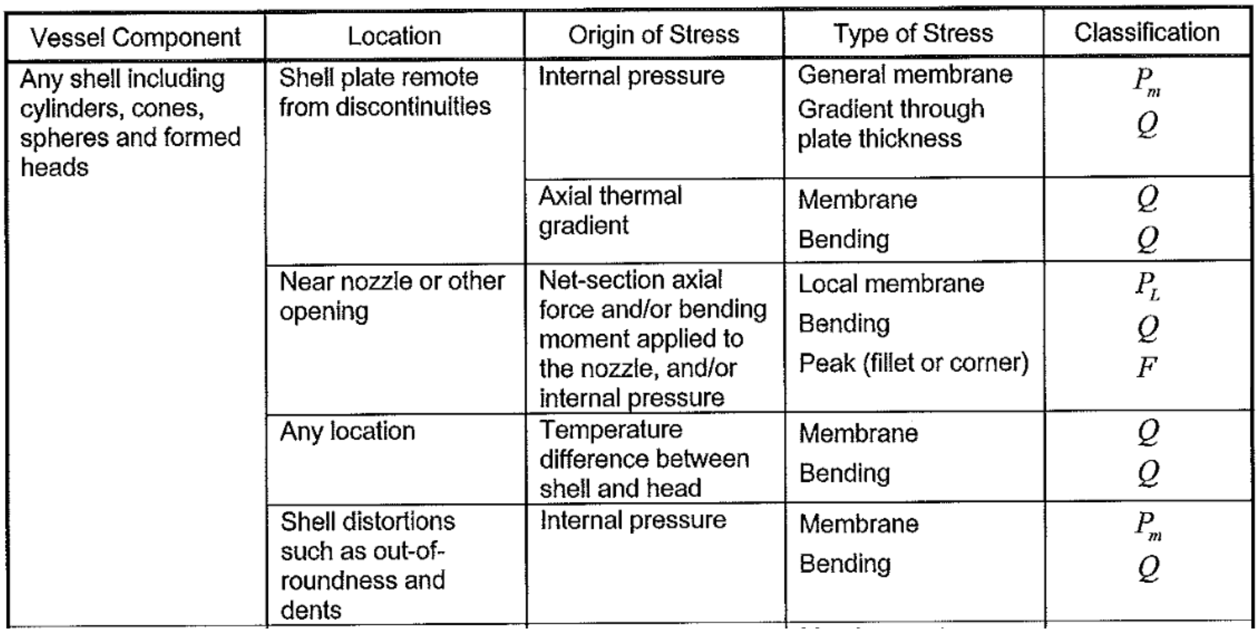


Figure 33: Example of the stress categorizing method result plot, Ref /14/.

Two secondary categories also exist, but they are not needed to evaluate protection against plastic collapse. However these components are needed for fatigue and ratcheting evaluations. They are defined as:

- Secondary equivalent stress ***(Q).***
    - Self-equilibrating stress necessary to satisfy continuity of structure.
    - Occurs at structural discontinuities.
    - Caused by mechanical loads or thermal expansion effects.
- Peak or stress concentration ***(F).***
    - Increment added to a primary or secondary stress by a concentration.
    - Certain thermal stresses that may cause fatigue, but not distortion of the vessel shape.

A table including examples of stress classification is included in the code, Ref /12/ as a helping aid in the stress categorizing procedure. A short summary of the table is presented in Figure 34 while the reader is referred to the code for the full content of the table.



Vessel Component	Location	Origin of Stress	Type of Stress	Classification
Any shell including cylinders, cones, spheres and formed heads	Shell plate remote from discontinuities	Internal pressure	General membrane Gradient through plate thickness	P_m Q
		Axial thermal gradient	Membrane Bending	Q Q
	Near nozzle or other opening	Net-section axial force and/or bending moment applied to the nozzle, and/or internal pressure	Local membrane Bending Peak (fillet or corner)	P_L Q F
	Any location	Temperature difference between shell and head	Membrane Bending	Q Q
	Shell distortions such as out-of-roundness and dents	Internal pressure	Membrane Bending	P_m Q

Figure 34: Examples of stress classification, Ref /12/.

Assessment procedure

The following procedure is used to calculate and categorize the equivalent stress at a given point in the structure, and to determine the acceptability of the resulting stress state.

Step 1:

Determine all types of loads including the different load combinations that are acting on the component. Suggested load combinations are presented in Figure 31.

Step 2:

At the point of interest, the stress tensor must be calculated for each type of load. The computed stress tensors shall be categorized as defined in the stress categories section.

- General primary membrane equivalent stress ***(P_m).***
- Local primary membrane equivalent stress ***(P_L).***
- Primary bending equivalent stress ***(P_b).***
- Secondary equivalent stress ***(Q).***
- Additional stress over and above the nominal ***(P+Q)*** level ***(F).***

Step 3:

The stress tensors assigned to each equivalent stress category shall be summarized. The summarizing is done on component basis and the final result is a stress tensor representing the combined effect of all loads assigned to the respective stress category. When using numerical methods such as finite element analysis the combinations of P_L+P_b and P_L+P_b+Q+F are usually provided directly.

- For a load case that include only “load-controlled” loads, that is pressure and weight effects, the computed equivalent stress shall be used to directly represent the P_m, P_L+P_b or P_L+P_b+Q.
- For a load case that include only “strain-controlled” loads such as thermal gradients the computed equivalent stress represent the ***Q*** only.
- For cases where stress is produced by a stress concentration or thermal stress the quantity ***F*** is the additional stress exceeding the nominal membrane stress plus the bending stress.

Step 4:

For each category the principal stresses of the sum of the stress tensors need to be calculated. Then the von Mises` (maximum distortion energy) yield criterion shall be used to calculate the equivalent stress at the location being investigated.

Step 5:

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

- $\boldsymbol{P_m \leq S}$
- $\boldsymbol{P_L \leq 1{,}5 \times S}$
- $\boldsymbol{(P_L + P_b) \leq 1{,}5 \times S}$

If these conditions are satisfied protection against plastic collapse is considered fulfilled.

2.2.5 Protection against local failure

The use of the elastic stress analysis method provides an approximation of the protection against local failure. In addition to the fulfillment of the conditions given in the protection against plastic collapse the following criterion shall be satisfied for each point in the component.

$$(\boldsymbol{\sigma_1} + \boldsymbol{\sigma_2} + \boldsymbol{\sigma_3}) \leq \mathbf{4} \times \boldsymbol{S}$$

The sum of the local primary membrane plus bending principal stresses ***(P_L+P_b)*** shall be used to check this criterion.

2.2.6 Protection against collapse from buckling

The design criteria for protection against plastic collapse shall be satisfied along with a design factor for protection against collapse from buckling. The protection against buckling criteria shall prevent buckling of components subjected to a compressive stress field under applied design loads.

If a bifurcation buckling analysis is performed using the elastic stress analysis without geometric nonlinearities in the solution to obtain the buckling load the minimum design factor shall be:

$$\boldsymbol{\Phi_B} = \frac{\mathbf{2}}{\boldsymbol{\beta_{cr}}}$$

The following capacity reduction factors shall be used:

- For un-stiffened or ring stiffened cylinders and cones under axial compression:

$$\boldsymbol{\beta_{cr}} = \mathbf{0.207}\ \boldsymbol{for}\ \frac{\boldsymbol{D_0}}{\boldsymbol{t}} \geq \mathbf{1247}$$

$$\boldsymbol{\beta_{cr}} = \frac{\mathbf{338}}{\mathbf{389} + {}^{\boldsymbol{D_0}}/_{\boldsymbol{t}}}\ \boldsymbol{for}\ \frac{\boldsymbol{D_0}}{\boldsymbol{t}} < 1247$$

- For un-stiffened and ring stiffened cylinders and cones under external pressure:

$$\boldsymbol{\beta_{cr}} = \mathbf{0,80}$$

- For spherical, tori-spherical and elliptical heads under external pressure:

$$\boldsymbol{\beta_{cr}} = \mathbf{0,124}$$

For this analysis the pre-stress in the component shall be established based on the load combinations given in Figure 31. For the numerical analysis all possible buckling modes shall be considered in the determination of the minimum buckling load. The geometric model should be carefully examined to ensure that model simplifications do not result in the exclusion of a critical buckling mode.

The capacity reduction factor is used to consider the effects of possible shell imperfections. The difference between collapse load and the bifurcation buckling point is illustrated in Figure 35, Ref /15/.

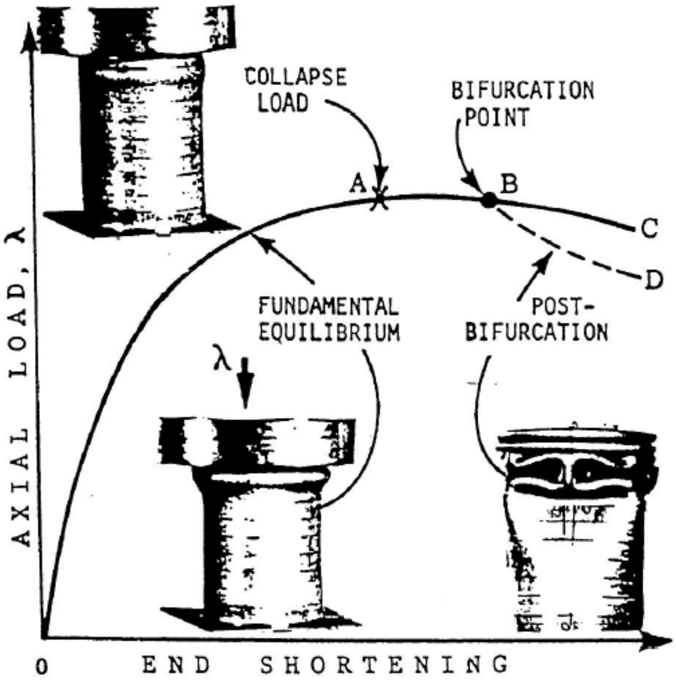


Figure 35: Load vs. end shortening with collapse (A) and bifurcation buckling (B) points, Ref /15/.

2.2.7 Protection against failure from cyclic loading

Fatigue analysis is required if the component is subjected to cyclic loading operation. To determine if a fatigue analysis is required screening criteria are provided. For design cases where the specified number of cycles is greater than 10^6, screening criteria are not applicable and a fatigue analysis is required.

Screening criteria for fatigue analysis

Screening based on experience with comparable equipment:

If successful experience is obtained with comparable equipment, and can be documented to follow a similar loading histogram, then a fatigue analysis is not required as part of the vessel design.

Fatigue analysis screening, Method A:

This simplified screening method can only be used for materials with a specified tensile strength less than or equal to 552 MPa (80000 psi). For materials with tensile strength above specified limit the reader is referred to Method B, section 5.5.2.4, Ref /12/. Method B is more elaborating and will not be considered in this thesis.

Step 1:

Determine a detailed load history of all cyclic operation loads and events that are applied to the component.

Step 2:

Based on the load history, determine the expected number of full-range pressure cycles including startup and shut down. This value shall be designated as $\boldsymbol{N_{\Delta FP}}$.

Step 3:

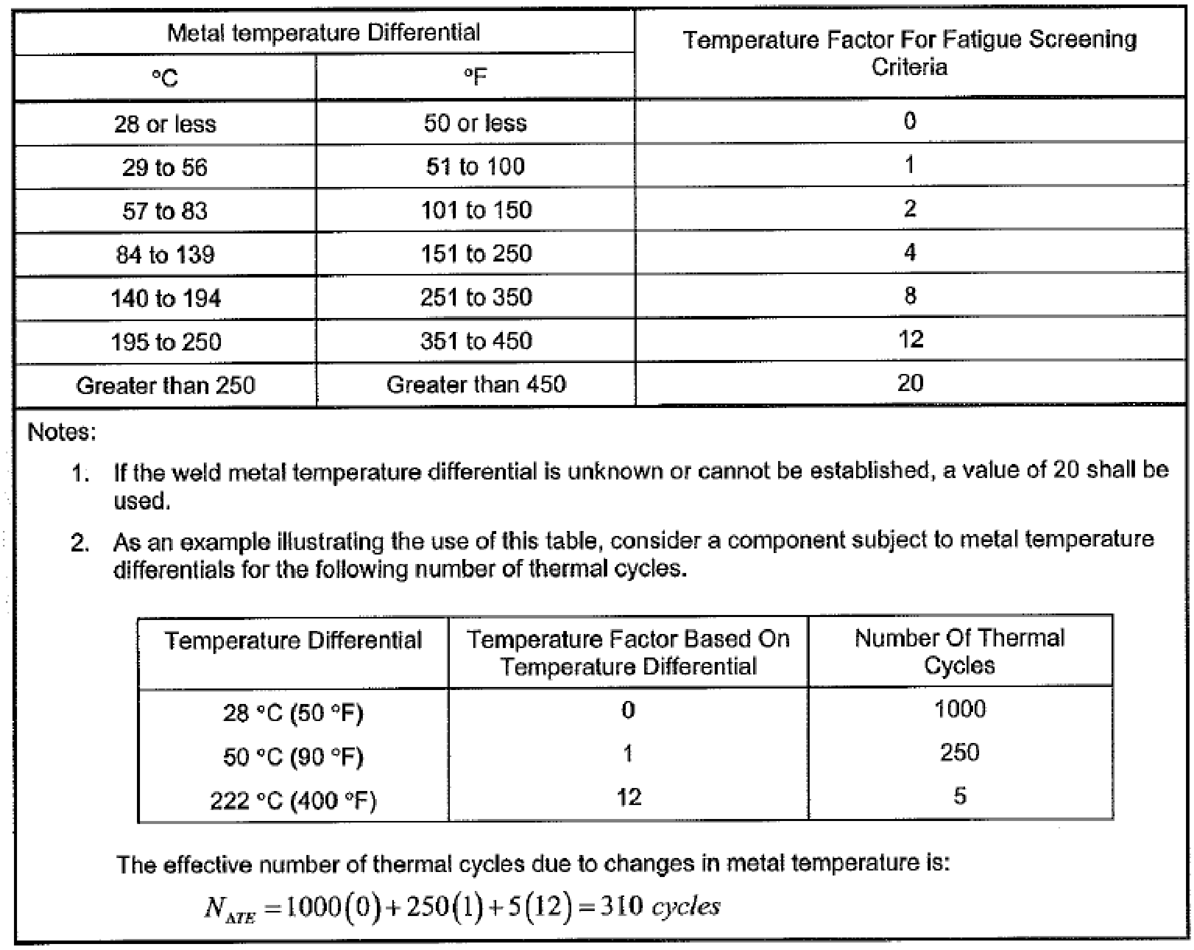
Based on the load history, determine the expected number of operating cycles where the pressure range **exceeds** the design pressure by:

- 20% for integral construction.
- 15% for non-integral construction.

This value shall be designated as $\boldsymbol{N_{\Delta P0}}$.

Step 4:

Based on the load history, determine the effective number of changes in material temperature between any two adjacent points. The number is determined by multiplying the number of changes in material temperature difference of a certain magnitude by a factor given in Figure 36, Ref /12/ and then adding the resulting numbers. The value shall be designated as $\boldsymbol{N_{\Delta TE}}$.



Metal temperature Differential		Temperature Factor For Fatigue Screening Criteria
°C	°F	
28 or less	50 or less	0
29 to 56	51 to 100	1
57 to 83	101 to 150	2
84 to 139	151 to 250	4
140 to 194	251 to 350	8
195 to 250	351 to 450	12
Greater than 250	Greater than 450	20

Notes:

1. If the weld metal temperature differential is unknown or cannot be established, a value of 20 shall be used.
2. As an example illustrating the use of this table, consider a component subject to metal temperature differentials for the following number of thermal cycles.

Temperature Differential	Temperature Factor Based On Temperature Differential	Number Of Thermal Cycles
28 °C (50 °F)	0	1000
50 °C (90 °F)	1	250
222 °C (400 °F)	12	5

The effective number of thermal cycles due to changes in metal temperature is:

$$N_{ATE} = 1000(0) + 250(1) + 5(12) = 310 \ cycles$$

Figure 36: Temperature factors for fatigue screening criteria, Ref /12/.

The length between adjacent points considered shall be:

- $L = 2,5\sqrt{Rt}$, for shells and dished heads.
- $L = 3,5a$, for flat plates.
- For through-the-thickness temperature differences any two points on a line normal to the surface of the component are defined as adjacent.

Step 5:

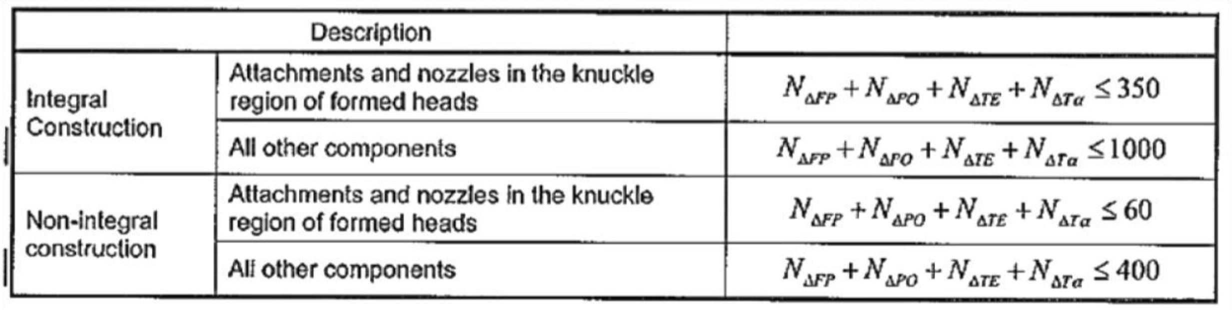
Based on the load history, determine the number of temperature cycles for welded components where the materials thermal expansion coefficients differ causing:

$$(\alpha_1 - \alpha_2)\Delta T > 0,00034$$

Designate this value as $N_{\Delta T\alpha}$.

Step 6:

Compare all values obtained in Step 2, 3, 4 and 5 with the criterion presented in Figure 37, Ref /12/.



Description		
Integral Construction	Attachments and nozzles in the knuckle region of formed heads	$N_{\Delta FP} + N_{\Delta PO} + N_{\Delta TE} + N_{\Delta T\alpha} \le 350$
	All other components	$N_{\Delta FP} + N_{\Delta PO} + N_{\Delta TE} + N_{\Delta T\alpha} \le 1000$
Non-integral construction	Attachments and nozzles in the knuckle region of formed heads	$N_{\Delta FP} + N_{\Delta PO} + N_{\Delta TE} + N_{\Delta T\alpha} \le 60$
	All other components	$N_{\Delta FP} + N_{\Delta PO} + N_{\Delta TE} + N_{\Delta T\alpha} \le 400$

Figure 37: Fatigue screening criteria for method A, Ref /12/.

If the criterion presented above is satisfied no fatigue analysis is required as part of the vessel design.

Fatigue assessment using the elastic stress analysis method

The controlling stress to be used in the fatigue evaluation is the effective total stress amplitude, defined as:

$$0.5(P_L + P_b + Q + F)$$

Assessment procedure

Step 1:

Determine load history including all significant operation loads and events that are applied to the component.

Step 2:

Determine the individual stress-strain cycles and define the number as ***M***. Methods for counting cycles are available in Annex 5.B, Ref /12/.

Step 3:

Determine the equivalent stress range for the $\boldsymbol{k^{th}}$ cycle that is counted in step 2. The code offers two different approaches to do this. Only one method is presented in this thesis.

The stress tensor $\left({}^{m}\boldsymbol{\sigma}_{ij,k}\,,\ {}^{n}\boldsymbol{\sigma}_{ij,k}\right)$ for the start and end points need to be determined (time points ${}^{m}\boldsymbol{t}$ and ${}^{n}\boldsymbol{t}$ for the $\boldsymbol{k^{th}}$ cycle). The local thermal stress $\left({}^{m}\boldsymbol{\sigma}^{LT}_{ij,k},\ {}^{n}\boldsymbol{\sigma}^{LT}_{ij,k}\right)$ at the same time points must also be determined.

Calculate the following quantities:

$$\Delta\sigma_{ij,k} = \left({}^{m}\sigma_{ij,k} - {}^{m}\sigma^{LT}_{ij,k}\right) - \left({}^{n}\sigma_{ij,k} - {}^{n}\sigma^{LT}_{ij,k}\right)$$

$$\left(\Delta S_{p,k} - \Delta S_{LT,k}\right) = \frac{1}{\sqrt{2}}\begin{bmatrix} \left(\Delta\sigma_{11,k} - \Delta\sigma_{22,k}\right)^2 + \left(\Delta\sigma_{11,k} - \Delta\sigma_{33,k}\right)^2 \\ +\left(\Delta\sigma_{22,k} - \Delta\sigma_{33,k}\right)^2 + 6\left(\Delta\sigma^2_{12,k} + \Delta\sigma^2_{13,k} + \Delta\sigma^2_{23,k}\right)\end{bmatrix}^{0.5}$$

$$\Delta\sigma^{LT}_{ij,k} = {}^{m}\sigma^{LT}_{ij,k} - {}^{n}\sigma^{LT}_{ij,k}$$

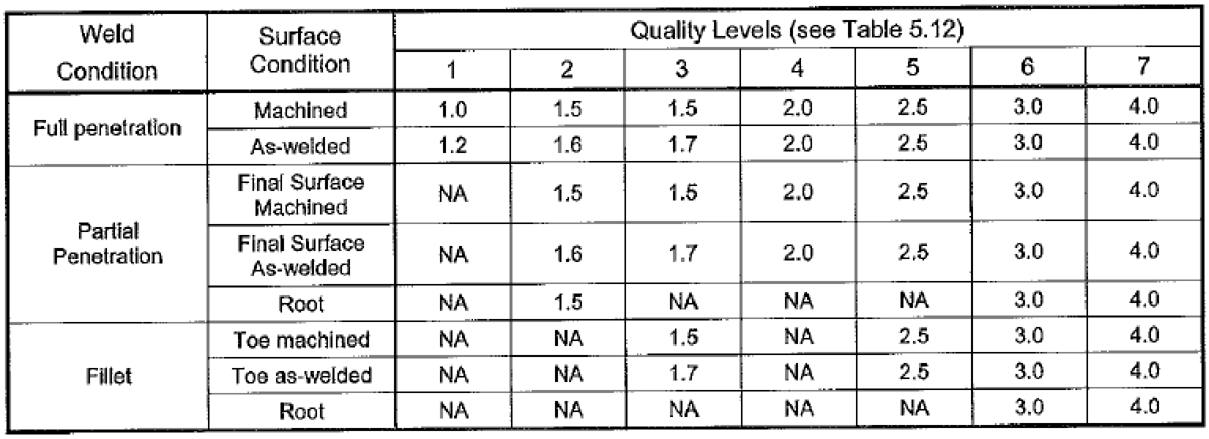
$$\Delta S_{LT,k} = \frac{1}{\sqrt{2}}\left[\left(\Delta\sigma^{LT}_{11,k} - \Delta\sigma^{LT}_{22,k}\right)^2 + \left(\Delta\sigma^{LT}_{11,k} - \Delta\sigma^{LT}_{33,k}\right)^2 + \left(\Delta\sigma^{LT}_{22,k} - \Delta\sigma^{LT}_{33,k}\right)^2\right]^{0.5}$$

Step 4:

Determine the effective alternating equivalent stress amplitude for the $\boldsymbol{k^{th}}$ cycle by means of the quantities established in step 3.

$$S_{alt,k} = \frac{K_f \times K_{e,k} \times \left(\Delta S_{p,k} - \Delta S_{LT,k}\right) + K_{v,k} \times \Delta S_{LT,k}}{2}$$

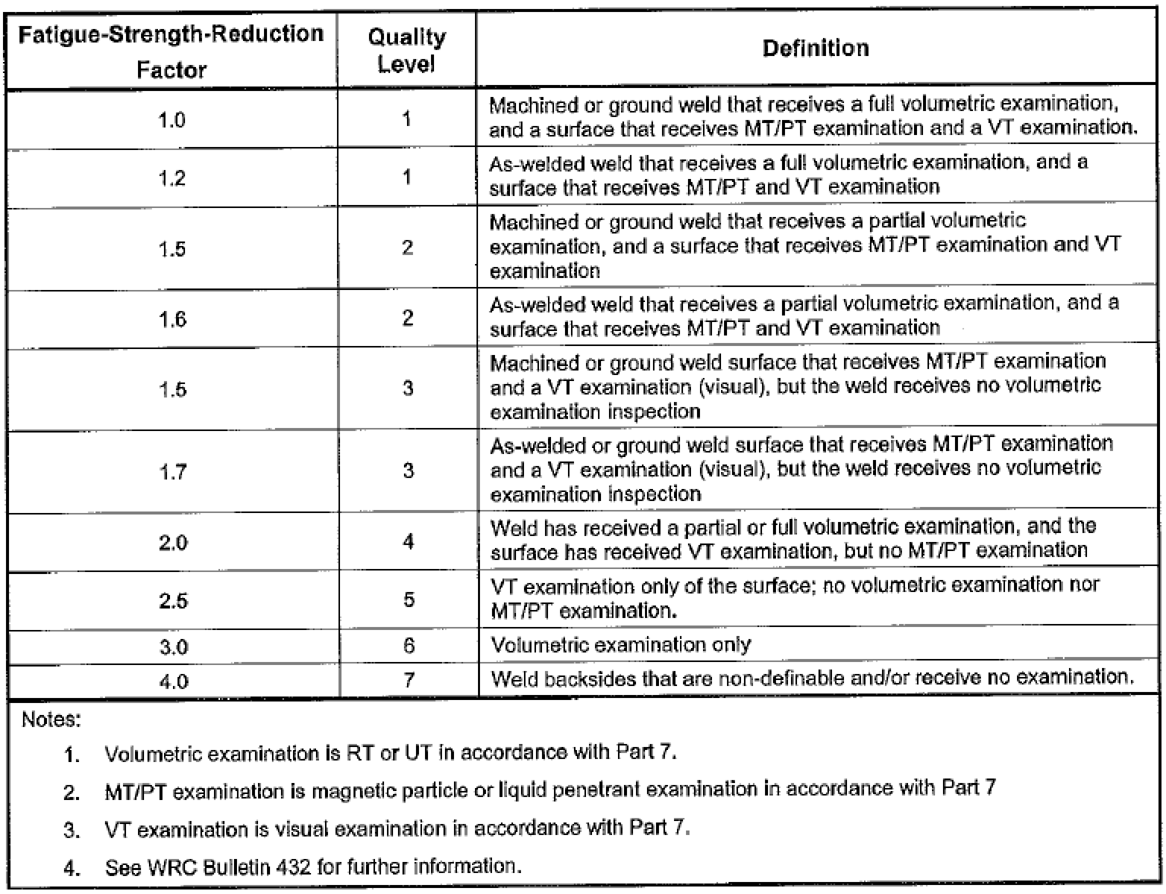
If effects from local notch and welds are included in the numerical model, then the strength reduction factor is $\boldsymbol{K_f=1.0}$. If not values for the strength reduction factor are provided in Figure 38 (table 5.11) and Figure 39 (table 5.12), Ref /12/.



Weld Condition	Surface Condition	Quality Levels (see Table 5.12)						
		1	2	3	4	5	6	7
Full penetration	Machined	1.0	1.5	1.5	2.0	2.5	3.0	4.0
	As-welded	1.2	1.6	1.7	2.0	2.5	3.0	4.0
Partial Penetration	Final Surface Machined	NA	1.5	1.5	2.0	2.5	3.0	4.0
	Final Surface As-welded	NA	1.6	1.7	2.0	2.5	3.0	4.0
	Root	NA	1.5	NA	NA	NA	3.0	4.0
Fillet	Toe machined	NA	NA	1.5	NA	2.5	3.0	4.0
	Toe as-welded	NA	NA	1.7	NA	2.5	3.0	4.0
	Root	NA	NA	NA	NA	NA	3.0	4.0

Figure 38: Weld surface fatigue strength reduction factors, Ref /12/.

The quality levels obtained from Figure 38 shall be used in Figure 39 below to establish the strength reduction factor.



Fatigue-Strength-Reduction Factor	Quality Level	Definition
1.0	1	Machined or ground weld that receives a full volumetric examination, and a surface that receives MT/PT examination and a VT examination.
1.2	1	As-welded weld that receives a full volumetric examination, and a surface that receives MT/PT and VT examination
1.5	2	Machined or ground weld that receives a partial volumetric examination, and a surface that receives MT/PT examination and VT examination
1.6	2	As-welded weld that receives a partial volumetric examination, and a surface that receives MT/PT and VT examination
1.5	3	Machined or ground weld surface that receives MT/PT examination and a VT examination (visual), but the weld receives no volumetric examination inspection
1.7	3	As-welded or ground weld surface that receives MT/PT examination and a VT examination (visual), but the weld receives no volumetric examination inspection
2.0	4	Weld has received a partial or full volumetric examination, and the surface has received VT examination, but no MT/PT examination
2.5	5	VT examination only of the surface; no volumetric examination nor MT/PT examination.
3.0	6	Volumetric examination only
4.0	7	Weld backsides that are non-definable and/or receive no examination.

Notes:

1. Volumetric examination is RT or UT in accordance with Part 7.
2. MT/PT examination is magnetic particle or liquid penetrant examination in accordance with Part 7
3. VT examination is visual examination in accordance with Part 7.
4. See WRC Bulletin 432 for further information.

Figure 39: Weld surface fatigue strength reduction factors, Ref /12/.

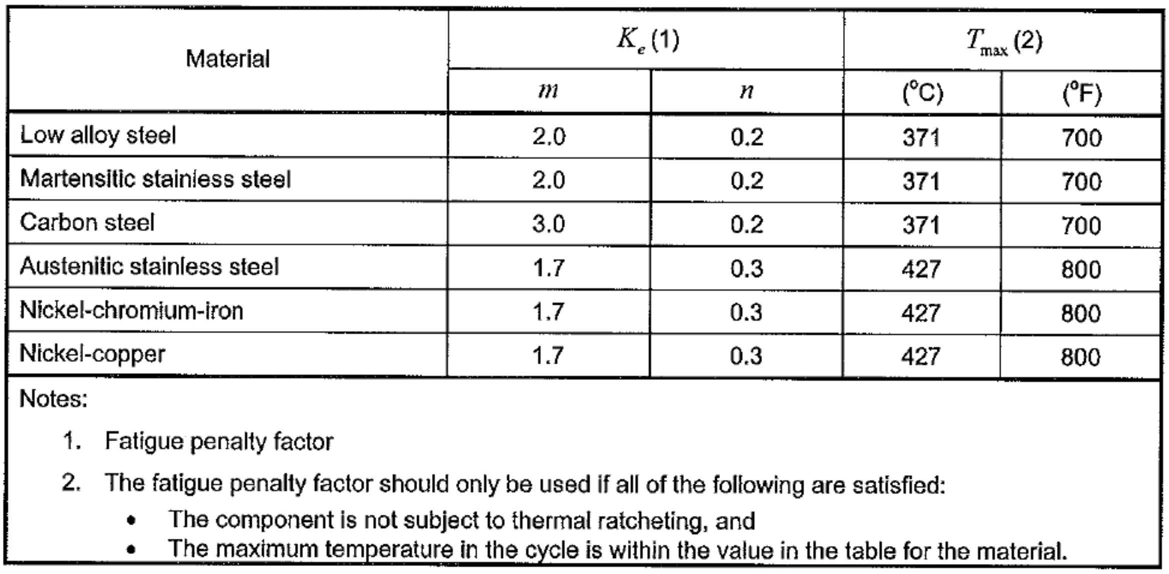
The fatigue penalty factor $K_{e,k}$ is calculated using the following equations:

$$K_{e,k} = 1.0 \; for \; \Delta S_{n,k} \le S_{PS}$$

$$K_{e,k} = 1.0 + \frac{(1-n)}{n(m-1)}\left(\frac{\Delta S_{n,k}}{S_{PS}} - 1\right) for \; S_{PS} < \Delta S_{n,k} < mS_{PS}$$

$$K_{e,k} = \frac{1}{n}\ for\ \Delta S_{n,k} \geq mS_{PS}$$

The factors **m** and **n** are determined from Figure 40, (table 5.13), Ref /12/ and the values for S_{PS} and $\Delta S_{n,k}$ are defined under Ratcheting assessment.



Material	K_e (1)		T_{max} (2)	
	m	n	(°C)	(°F)
Low alloy steel	2.0	0.2	371	700
Martensitic stainless steel	2.0	0.2	371	700
Carbon steel	3.0	0.2	371	700
Austenitic stainless steel	1.7	0.3	427	800
Nickel-chromium-iron	1.7	0.3	427	800
Nickel-copper	1.7	0.3	427	800

Notes:

1. Fatigue penalty factor
2. The fatigue penalty factor should only be used if all of the following are satisfied:
    - The component is not subject to thermal ratcheting, and
    - The maximum temperature in the cycle is within the value in the table for the material.

Figure 40: Fatigue penalty factors, Ref /12/.

The Poisson correction factor $K_{v,k}$ is calculated using the following equation:

$$K_{e,k} = (\frac{1 - \nu_e}{1 - \nu_p})$$

$$\nu_p = max\left[0.5 - 0.2\left(\frac{S_{y,k}}{S_{a,k}}\right), \nu_e\right]$$

Step 5:

Determine the number of permissible cycles, N_k for the alternating equivalent stress computed in step 4. The value is obtained from fatigue curves that are provided in Annex 3.F, Ref /12/.

Step 6:

Determine the fatigue damage factor for the k^{th} cycle, where the actual number of repetitions of the cycle is n_k.

$$D_{f,k} = \frac{n_k}{N_k}$$

Step 7:

Step 3 through step 6 shall be repeated for all stress ranges identified in step 2.

Step 8:

Finally the total accumulated fatigue damage for all stress ranges must be computed. The location investigated for fatigue damage is acceptable for continued operation if the following equation is satisfied:

$$\boldsymbol{D_f = \sum_{k}^{M} D_{f,k} \leq 1.0}$$

Step 9:

Repeat this check for each point in the structure subjected to a fatigue evaluation.

2.2.8 Ratcheting assessment

When using the elastic approach for ratcheting analysis the following limit shall be satisfied:

$$\Delta \boldsymbol{S_{n,k}} \leq \boldsymbol{S_{PS}}$$

The quantity $\Delta \boldsymbol{S_{n,k}}$ is the equivalent stress range, derived from the highest value across the thickness of a section of the $(\boldsymbol{P_L} + \boldsymbol{P_b} + \boldsymbol{Q})$**.** (See the stress categories section).

The allowable limit $\boldsymbol{S_{PS}}$ is computed as the largest of:

- Three times the average of the ***S*** value for the material from Annex 3.A for the highest and lowest temperature during the operation cycle.
- Two times the $\boldsymbol{S_y}$ values for the material from Annex 3.D at the highest and lowest temperature during the operation cycle.

2.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010

2.3.1 General information

In the elastic-plastic stress analysis a collapse load is derived considering both the loading and the deformation characteristics of the component. The allowable load is established by applying design factors to the calculated plastic collapse load. The requirements given in the code are based on protection against different failure modes.

The main failure modes considered in ASME VIII div. 2; 2007/2010, Boiler and Pressure Vessel Code are, Ref /12/:

- Protection against plastic collapse.
- Protection against local failure.
- Protection against collapse from buckling.
- Protection against failure from cyclic loading or fatigue.
    - Ratcheting assessment.

Disadvantages of the elastic-plastic stress analysis method, Ref /14/:

- The elastic-plastic analysis is more complex and the calculations will require more computation time.
- The analyst is required to define appropriate service criteria for the structure which require significant experience and excellent understanding of structural behavior.

Advantages of the elastic-plastic stress analysis method:

- Inelastic analysis eliminates the problem with categorizing stresses.
- Applicable for tree-dimensional stress fields and models with complex geometry.
- The actual structure behavior is more accurate presented by using this method.
- The method allows for improvements to the conventional design.
- Provides a more flexible approach to customized engineered solutions and details.

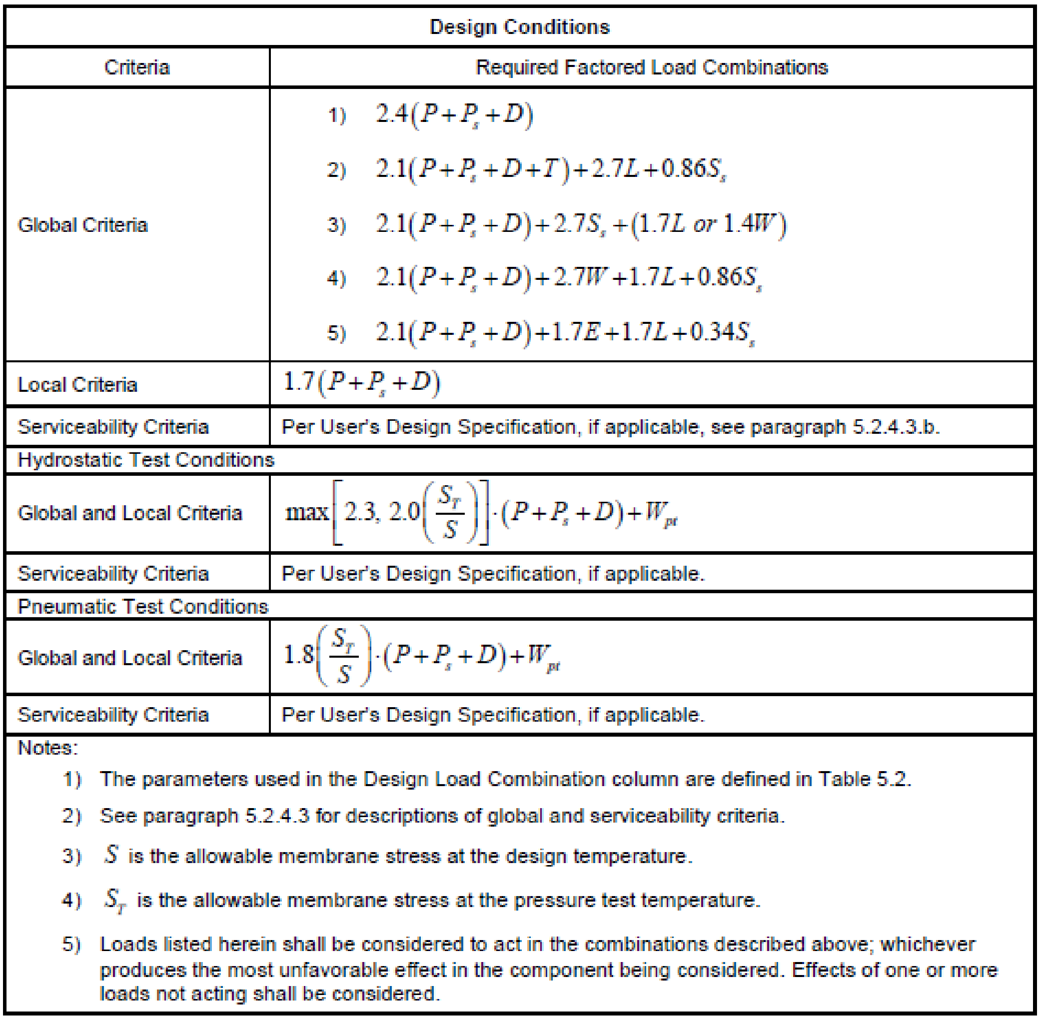
2.3.2 Loading conditions

All loads applied on the component shall be considered in the analysis. Combination of loads and loads that are varying in time must also be considered. For time varying loads a load histogram shall be developed to show the time variation of each specific load.

For normal operation the following load cases are considered:

- Dead weight of the vessel, content, and equipment at the location of interest ***(D).***
- Live loading including the effect of fluid momentum ***(L).***
- Internal and external specified design pressure ***(P).***
- Static head from liquid or bulk materials ***(P_S).***
- Earthquake loads ***(E).***
- Wind loads ***(W).***
- Snow loads ***(S_S).***
- Self-restraining loads like thermal loads and applied displacements ***(T).***

Suggested design load case combinations for an elastic-plastic analysis are provided in the code (table 5.5), Ref /12/ and presented in Figure 41.



Design Conditions	
Criteria	Required Factored Load Combinations
Global Criteria	1) $2.4(P+P_s+D)$ 2) $2.1(P+P_s+D+T)+2.7L+0.86S_s$ 3) $2.1(P+P_s+D)+2.7S_s+(1.7L \text{ or } 1.4W)$ 4) $2.1(P+P_s+D)+2.7W+1.7L+0.86S_s$ 5) $2.1(P+P_s+D)+1.7E+1.7L+0.34S_s$
Local Criteria	$1.7(P+P_s+D)$
Serviceability Criteria	Per User's Design Specification, if applicable, see paragraph 5.2.4.3.b.
Hydrostatic Test Conditions	
Global and Local Criteria	$\max\left[2.3,\ 2.0\left(\frac{S_T}{S}\right)\right]\cdot(P+P_s+D)+W_{pt}$
Serviceability Criteria	Per User's Design Specification, if applicable.
Pneumatic Test Conditions	
Global and Local Criteria	$1.8\left(\frac{S_T}{S}\right)\cdot(P+P_s+D)+W_{pt}$
Serviceability Criteria	Per User's Design Specification, if applicable.

Notes:

1) The parameters used in the Design Load Combination column are defined in Table 5.2.
2) See paragraph 5.2.4.3 for descriptions of global and serviceability criteria.
3) S is the allowable membrane stress at the design temperature.
4) S_T is the allowable membrane stress at the pressure test temperature.
5) Loads listed herein shall be considered to act in the combinations described above; whichever produces the most unfavorable effect in the component being considered. Effects of one or more loads not acting shall be considered.

Figure 41: Load case combinations, Ref /12/.

2.3.3 Material properties

Physical properties like; Young`s modulus, thermal expansion coefficient, thermal conductivity, thermal diffusivity, density and Poisson`s ratio are found in Part 3, Ref /12/

Strength parameters like allowable stress ***(S),*** minimum specified yield strength and minimum specified tensile strength for different materials at different temperatures are found in, Table 5A, Ref /13/. A sample of table 5A containing the first six lines is provided in Figure 42.

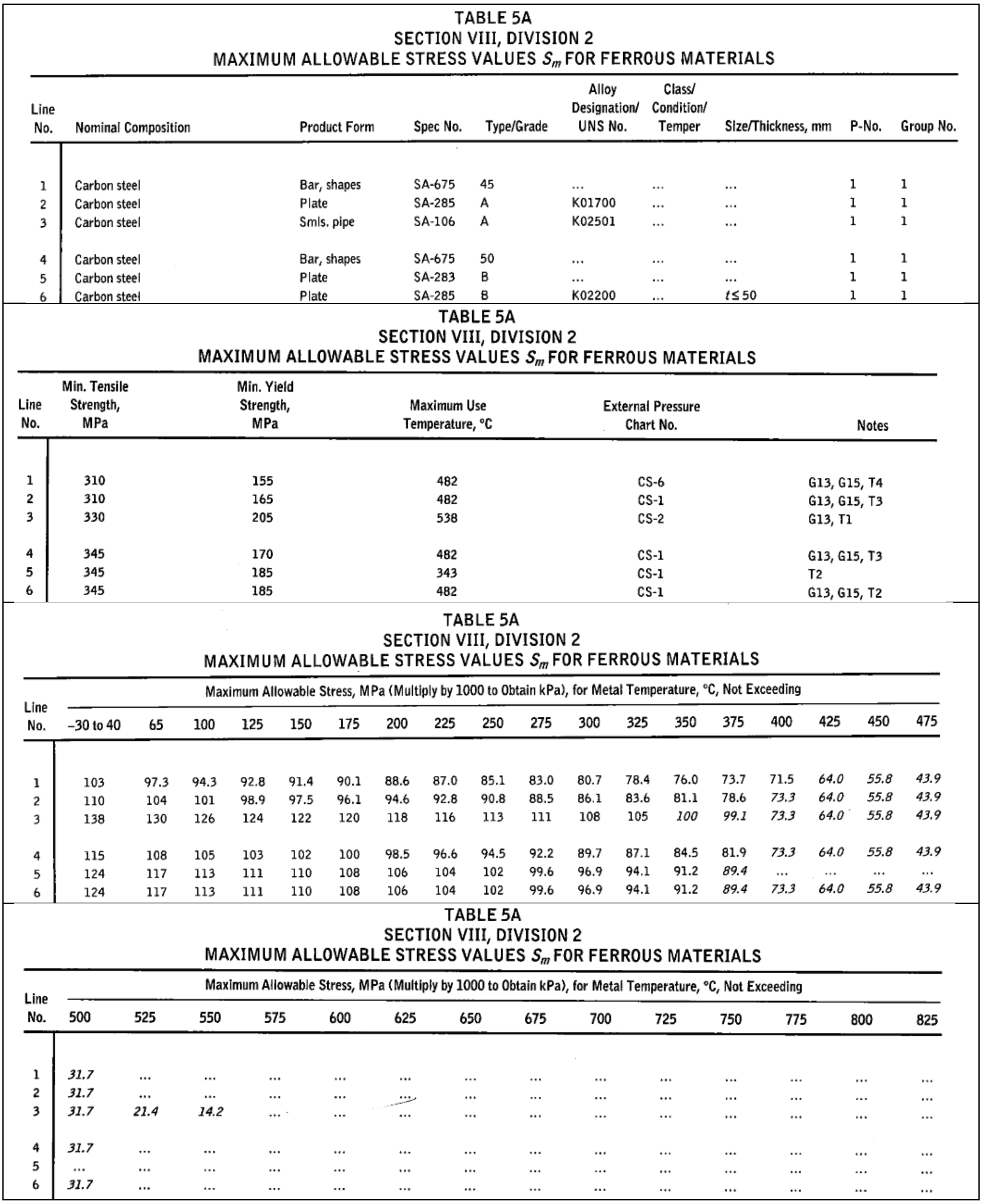


TABLE 5A
SECTION VIII, DIVISION 2
MAXIMUM ALLOWABLE STRESS VALUES S_m FOR FERROUS MATERIALS

Line No.	Nominal Composition	Product Form	Spec No.	Type/Grade	Alloy Designation/ UNS No.	Class/ Condition/ Temper	Size/Thickness, mm	P-No.	Group No.
1	Carbon steel	Bar, shapes	SA-675	45	...	...	...	1	1
2	Carbon steel	Plate	SA-285	A	K01700	...	...	1	1
3	Carbon steel	Smls. pipe	SA-106	A	K02501	...	...	1	1
4	Carbon steel	Bar, shapes	SA-675	50	...	...	...	1	1
5	Carbon steel	Plate	SA-283	B	...	...	...	1	1
6	Carbon steel	Plate	SA-285	B	K02200	...	$t \leq 50$	1	1

TABLE 5A
SECTION VIII, DIVISION 2
MAXIMUM ALLOWABLE STRESS VALUES S_m FOR FERROUS MATERIALS

Line No.	Min. Tensile Strength, MPa	Min. Yield Strength, MPa	Maximum Use Temperature, °C	External Pressure Chart No.	Notes
1	310	155	482	CS-6	G13, G15, T4
2	310	165	482	CS-1	G13, G15, T3
3	330	205	538	CS-2	G13, T1
4	345	170	482	CS-1	G13, G15, T3
5	345	185	343	CS-1	T2
6	345	185	482	CS-1	G13, G15, T2

TABLE 5A
SECTION VIII, DIVISION 2
MAXIMUM ALLOWABLE STRESS VALUES S_m FOR FERROUS MATERIALS

Line	Maximum Allowable Stress, MPa (Multiply by 1000 to Obtain kPa), for Metal Temperature, °C, Not Exceeding																	
No.	−30 to 40	65	100	125	150	175	200	225	250	275	300	325	350	375	400	425	450	475
1	103	97.3	94.3	92.8	91.4	90.1	88.6	87.0	85.1	83.0	80.7	78.4	76.0	73.7	71.5	*64.0*	*55.8*	*43.9*
2	110	104	101	98.9	97.5	96.1	94.6	92.8	90.8	88.5	86.1	83.6	81.1	78.6	*73.3*	*64.0*	*55.8*	*43.9*
3	138	130	126	124	122	120	118	116	113	111	108	105	*100*	*99.1*	*73.3*	*64.0*	*55.8*	*43.9*
4	115	108	105	103	102	100	98.5	96.6	94.5	92.2	89.7	87.1	84.5	81.9	*73.3*	*64.0*	*55.8*	*43.9*
5	124	117	113	111	110	108	106	104	102	99.6	96.9	94.1	91.2	*89.4*	...	...	...	...
6	124	117	113	111	110	108	106	104	102	99.6	96.9	94.1	91.2	*89.4*	*73.3*	*64.0*	*55.8*	*43.9*

TABLE 5A
SECTION VIII, DIVISION 2
MAXIMUM ALLOWABLE STRESS VALUES S_m FOR FERROUS MATERIALS

Line	Maximum Allowable Stress, MPa (Multiply by 1000 to Obtain kPa), for Metal Temperature, °C, Not Exceeding													
No.	500	525	550	575	600	625	650	675	700	725	750	775	800	825
1	*31.7*	...	...	...	...	...	...	...	...	...	...	...	...	...
2	*31.7*	...	...	...	...	...	...	...	...	...	...	...	...	...
3	*31.7*	*21.4*	*14.2*	...	...	...	...	...	...	...	...	...	...	...
4	*31.7*	...	...	...	...	...	...	...	...	...	...	...	...	...
5	...	...	...	...	...	...	...	...	...	...	...	...	...	...
6	*31.7*	...	...	...	...	...	...	...	...	...	...	...	...	...

Figure 42: Sample of table 5A, Ref /13/.

2.3.4 Protection against plastic collapse

The plastic collapse load can be established using a numerical analysis like the finite element method. An elastic-plastic model needs to be incorporated to obtain a solution. The effect of non-linear geometry shall be included as well as geometric strengthening/weakening. The plastic collapse load is defined as the load that causes overall structural instability. This point is indicated by the inability to obtain equilibrium conditions for the solution for a small increase in load. For the numerical calculation it is the point where the solution will not converge.

Acceptance criteria

Two acceptance criteria are presented for use with an elastic-plastic analysis, namely the global criteria and the service criteria.

The global criterion uses the global plastic collapse load as a limiting value. The plastic collapse load is defined as the load, or load combination which causes overall structural instability. As an alternative to the rigorous calculations and iterations needed to obtain the plastic collapse load for a structure, the concept of Load and Resistance Factor Design might be used (LRFD).

The LRFD procedure uses factored loads as presented in Figure 41 including a design factor to account for uncertainty. Then the resistance of the component to these factored loads is determined using the elastic-plastic analysis.

The service criteria states that every location in the structure shall be within the limit for acceptable performance when the component is subjected to design loads. The plastic collapse criteria may be satisfied but the component could have large deformations causing unsatisfactory performance. In this case the design loads may have to be reduced based on deformation or deflection criterion.

Assessment procedure

The following procedure is used to determine the acceptability of component using an elastic-plastic stress analysis.

Step 1:

An accurate numerical model of the component needs to be constructed. The model shall contain all relevant geometry properties, boundary conditions and applied loads of the component. Areas of complex geometry might acquire additional numerical models to obtain an accurate description of the stresses and strains.

Step 2:

All relevant loads and combination of loads shall be determined. The loads to be considered are defined and presented in the chapter called loading conditions.

Step 3:

A material model that includes hardening or softening, or an elastic-perfectly plastic model might be used. If plasticity is expected the von Mises yield function and associated flow rule shall be used. The effect of non-linear geometry shall be considered. Appendix 3.D, Ref /12/ contains true stress-strain curve models that consider temperature dependent hardening. When using this model the hardening behavior shall be considered up to the true ultimate stress limit and perfect plasticity behavior shall be used beyond this limit. Perfect plasticity behavior is equivalent to no slope of the stress-strain curves.

Step 4:

Determine the different load case combinations to use in the analysis from Figure 41 (table 5.5). Each of the load cases shall be investigated and evaluated.

Step 5:

Perform an elastic-plastic analysis for each of the load cases established in step 4. If convergence is obtained the component is stable under the applied load case. If the analysis is unable to reach convergence the component configuration shall be modified or the applied loads reduced and the analysis repeated.

Convergence for the solution of the elastic-plastic analysis is considered as protection against plastic collapse.

2.3.5 Protection against local failure

The following procedure shall be used to evaluate protection against local failure using the elastic-plastic method.

Step 1:

An elastic-plastic stress analysis based on the load combinations given in Figure 31 shall be performed. The combinations for local criteria shall be used along with the effects of non-linear geometry (large deflections).

Step 2:

For the location to be investigated the principal stresses shall be determined. The von Mises` (maximum distortion energy) yield criterion shall be used to establish the equivalent stress:

$$\boldsymbol{\sigma_e = \frac{1}{\sqrt{2}}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]^{0.5}}$$

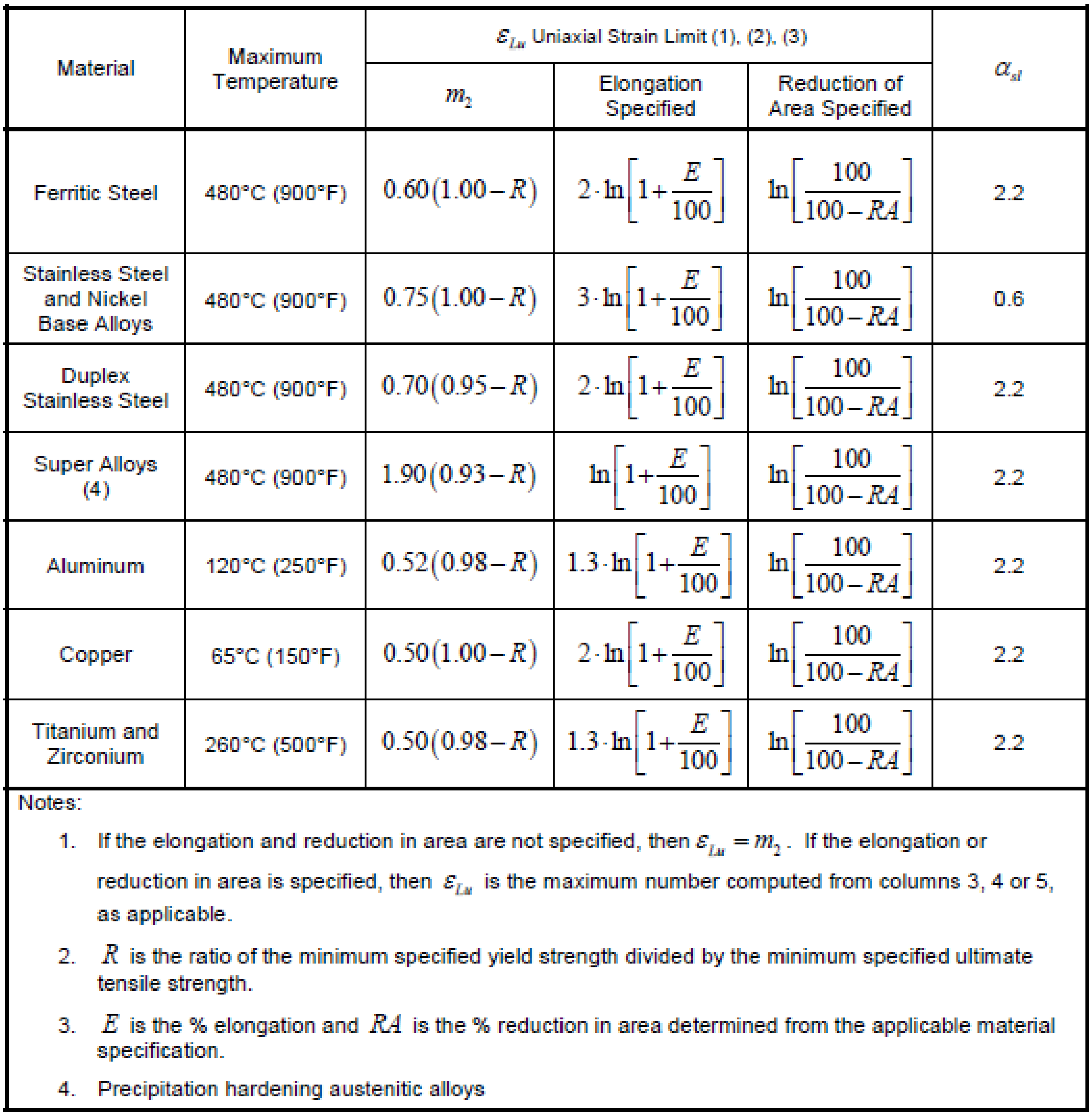
The total equivalent plastic strain shall be obtained from the equivalent stress and denoted as $\boldsymbol{\varepsilon_{\mathrm{peq}}}$.

Step 3:

The limiting tri-axial strain is determined by the following equation:

$$\boldsymbol{\varepsilon_L = \varepsilon_{Lu} \times exp\left[-\left(\frac{\alpha_{sl}}{1+m_2}\right)\left(\left\{\frac{\sigma_1+\sigma_2+\sigma_3}{3\sigma_e}\right\} - \frac{1}{3}\right)\right]}$$

The quantities **($\boldsymbol{\varepsilon_{Lu}}$, $\boldsymbol{m_2}$ and $\boldsymbol{\alpha_{sl}}$)** are determined from (table 5.7) Figure 43 , Ref /12/.



Material	Maximum Temperature	ε_{Lu} Uniaxial Strain Limit (1), (2), (3)			α_{sl}
		m_2	Elongation Specified	Reduction of Area Specified	
Ferritic Steel	480°C (900°F)	$0.60(1.00-R)$	$2\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2
Stainless Steel and Nickel Base Alloys	480°C (900°F)	$0.75(1.00-R)$	$3\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	0.6
Duplex Stainless Steel	480°C (900°F)	$0.70(0.95-R)$	$2\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2
Super Alloys (4)	480°C (900°F)	$1.90(0.93-R)$	$\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2
Aluminum	120°C (250°F)	$0.52(0.98-R)$	$1.3\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2
Copper	65°C (150°F)	$0.50(1.00-R)$	$2\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2
Titanium and Zirconium	260°C (500°F)	$0.50(0.98-R)$	$1.3\cdot\ln\left[1+\frac{E}{100}\right]$	$\ln\left[\frac{100}{100-RA}\right]$	2.2

Notes:

1. If the elongation and reduction in area are not specified, then $\varepsilon_{Lu} = m_2$. If the elongation or reduction in area is specified, then ε_{Lu} is the maximum number computed from columns 3, 4 or 5, as applicable.
2. R is the ratio of the minimum specified yield strength divided by the minimum specified ultimate tensile strength.
3. E is the % elongation and RA is the % reduction in area determined from the applicable material specification.
4. Precipitation hardening austenitic alloys

Figure 43: Uniaxial strain limit, Ref /12/.

Step 4:

The forming strain based on the materials and fabrication method in Part 6, Ref /12/ shall be determined and denoted as ε_{cf}. If heat treatment is performed the forming strain may be assumed to be zero.

Step 5:

If the following equation is satisfied at the location being investigated the component is considered protected against local failure.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

A procedure for evaluating protection against local failure for a component subjected to a sequence of loading is also provided. In this procedure the load sequence is divided into ***k*** load increments. Then the principal stresses for each increment:

$$\boldsymbol{\sigma_{1,k}},\ \boldsymbol{\sigma_{2,k}},\ \boldsymbol{\sigma_{3,k}}$$

The equivalent stress for each increment is calculated with von Mises formula as presented in step 2. And then the change in equivalent stress and change in the equivalent plastic strain from the previous load increment will be:

$$\Delta\boldsymbol{\sigma}_{\mathbf{e,k}}$$

$$\boldsymbol{\Delta\varepsilon}_{\mathbf{peq,k}}$$

The strain limit for the $\boldsymbol{k^{th}}$ load increment is calculated using the following formula:

$$\boldsymbol{\varepsilon_{L,k} = \varepsilon_{Lu} \times exp\left[-\left(\frac{\alpha_{sl}}{1+m_2}\right)\left(\left\{\frac{\sigma_{1,k}+\sigma_{2,k}+\sigma_{3,k}}{3\sigma_{e,k}}\right\}-\frac{1}{3}\right)\right]}$$

Here the quantities $\boldsymbol{\varepsilon_{Lu}}$**,** $\boldsymbol{m_2}$ **and** $\boldsymbol{\alpha_{sl}}$ are determined from (table 5.7) Figure 43, Ref /12/.

The strain limit damage for each increment is calculated using:

$$\mathbf{D}_{\boldsymbol{\varepsilon},\mathbf{k}} = \frac{\boldsymbol{\Delta\varepsilon}_{\mathbf{peq,k}}}{\boldsymbol{\varepsilon}_{\mathbf{L,k}}}$$

The strain limit damage from forming is calculated by:

$$\boldsymbol{D_{\varepsilon,form} = \frac{\varepsilon_{cf}}{\varepsilon_{Lu} \times exp\left[-0.67\left(\frac{\alpha_{sl}}{1+m_2}\right)\right]}}$$

If heat treatment is performed according to Part 6, Ref /12/ the strain limit damage from forming is assumed to be zero.

If the following equation is satisfied the location in the component is acceptable for the specified loading sequence:

$$\boldsymbol{D_{\varepsilon} = D_{\varepsilon,form} + \sum_{k=1}^{M} D_{\varepsilon,k} \le 1.0}$$

2.3.6 Protection against collapse from buckling

The design criteria for protection against plastic collapse shall be satisfied along with a design factor for protection against collapse from buckling. The protection against buckling criteria shall prevent buckling of components subjected to a compressive stress field under applied design loads.

If a bifurcation buckling analysis is performed using the elastic-plastic stress analysis with the effect of geometric nonlinearities in the solution to obtain the buckling load the minimum design factor shall be:

$$\boldsymbol{\Phi_B = \frac{1.667}{\beta_{cr}}}$$

The following capacity reduction factors shall be used:

- For un-stiffened or ring stiffened cylinders and cones under axial compression:

$$\boldsymbol{\beta_{cr} = 0.207\, for\, \frac{D_0}{t} \geq 1247}$$

$$\boldsymbol{\beta_{cr} = \frac{338}{389 + {D_0}/{t}}\, for\, \frac{D_0}{t}} < 1247$$

- For un-stiffened and ring stiffened cylinders and cones under external pressure:

$$\boldsymbol{\beta_{cr} = 0,80}$$

- For spherical, tori-spherical, elliptical heads under external pressure:

$$\boldsymbol{\beta_{cr} = 0,124}$$

For this analysis the pre-stress in the component shall be established based on the load combinations presented for the elastic analysis. For the numerical analysis all possible buckling modes shall be considered in the determination of the minimum buckling load. The geometric model should be carefully examined to ensure that model simplifications do not result in the exclusion of a critical buckling mode.

If a plastic collapse analysis is conducted as presented in the section “protection against plastic collapse” using the factored load combinations given in Figure 31, the design factor for buckling is accounted for.

2.3.7 Protection against failure from cyclic loading

Fatigue analysis is required if the component is subjected to cyclic loading operation. To determine if a fatigue analysis is required the screening criteria presented in the section containing the elastic analysis method might be used. For design cases where the specified numbers of cycles are greater than 10^6, screening criteria are not applicable and a fatigue analysis is required anyway.

When using the elastic plastic stress analysis in the fatigue damage evaluation a parameter known as the effective strain range is used. Two methods are suggested for this calculation, namely the cycle-by-cycle analysis or the twice yield method. The cycle-by-cycle method require a cyclic plasticity algorithm with kinematic hardening while the twice yield method can be used with an analysis program without cyclic plasticity capability.

The twice yield method is based on a specified cyclic*"stress range-strain range"* curve with a load representing the cycle. The cyclic curve can be obtained by material testing or curves that are known to be more conservative can be used. Cyclic stress-strain curves for certain materials and temperatures are also provided in Annex 3. D, Ref/12/.

Assessment procedure

Step 1:

Determine all significant operating loads and events the component might be subjected to.

Step 2:

Determine the individual stress-strain cycles for the location being investigated and define the number as ***M***. Methods for counting cycles are available in Annex 5.B, Ref /12/.

Step 3:

For each of the cycles counted in step 2, determine the loadings at the start and end points. Use this data to calculate the loading ranges (the differences between the loadings at the start and end point) for the k^{th} cycle.

Step 4:

Perform an elastic-plastic stress analysis for the $\mathbf{k}^{\mathbf{th}}$ cycle. For the cycle-by-cycle analysis a constant amplitude loading is cycled using the plasticity algorithm with kinematic hardening. For the twice yield method, the loading at the start point is considered to be zero while the loading at the end point is the range determined in step 3. Cyclic stress-strain curves from Annex 3.D, Ref /12/ may be used.

Step 5:

Calculate the effective strain range for the k^{th} cycle using the following equation:

$$\Delta\varepsilon_{eff,k} = \frac{\Delta S_{P,k}}{E_{ya,k}} + \Delta\varepsilon_{peq,k}$$

Where the effective von Mises stress range is calculated by:

$$\boldsymbol{\Delta S_{P,k}} = \frac{\mathbf{1}}{\sqrt{\mathbf{2}}}\begin{bmatrix} (\Delta\sigma_{11,k} - \Delta\sigma_{22,k})^2 + (\Delta\sigma_{11,k} - \Delta\sigma_{33,k})^2 \\ +(\Delta\sigma_{22,k} - \Delta\sigma_{33,k})^2 + 6(\Delta\sigma_{12,k}^2 + \Delta\sigma_{13,k}^2 + \Delta\sigma_{23,k}^2) \end{bmatrix}^{0.5}$$

And equivalent plastic strain range for the $\boldsymbol{k^{th}}$ step is calculated by:

$$\boldsymbol{\Delta\varepsilon_{peq,k}} = \frac{\sqrt{2}}{\mathbf{3}}\begin{bmatrix} (\Delta P_{11,k} - \Delta P_{22,k})^2 + (\Delta P_{22,k} - \Delta P_{33,k})^2 + (\Delta P_{33,k} - \Delta P_{11,k})^2 \\ +1.5(\Delta P_{12,k}^2 + \Delta P_{23,k}^2 + \Delta P_{31,k}^2) \end{bmatrix}^{0.5}$$

The component stress and plastic strain ranges for the $\boldsymbol{k^{th}}$ cycle are designated as $\boldsymbol{\Delta\sigma_{ij,k}}$ and $\boldsymbol{\Delta P_{ij,k}}$ respectively.

Using the twice yield method the equivalent plastic strain range and the von Mises equivalent stress range are outputs that can be obtained directly from a stress analysis.

Step 6:

Calculate the effective alternating stress for the $\boldsymbol{k^{th}}$ cycle using:

$$\boldsymbol{S_{alt,k}} = \frac{\boldsymbol{E_{y,k}} \times \Delta\boldsymbol{\varepsilon_{eff,k}}}{\mathbf{2}}$$

Step 7:

Determine the number of permissible cycles, $\boldsymbol{N_k}$ for the alternating equivalent stress computed in step 6. The value is obtained from fatigue curves that are provided in Annex 3.F, Ref /12/.

Step 8:

Determine the fatigue damage factor for the $\boldsymbol{k^{th}}$ cycle, where the actual number of repetitions of the cycle is $\boldsymbol{n_k}$.

$$\boldsymbol{D_{f,k}} = \frac{\boldsymbol{n_k}}{\boldsymbol{N_k}}$$

Step 9:

Step 3 through step 8 shall be repeated for all stress ranges identified in step 2.

Step 10:

The total accumulated fatigue damage for all stress ranges must be computed. The location investigated for fatigue damage is acceptable for continued operation if the following equation is satisfied:

$$\boldsymbol{D_f} = \sum_{k}^{M} \boldsymbol{D_{f,k}} \leq \mathbf{1.0}$$

Step 11:

Repeat step 2 through 10 for each point in the component subjected to the fatigue evaluation.

2.3.8 Ratcheting assessment

Under special combinations of steady state loads and cyclic loadings there is a possibility for ratcheting. A proper evaluation of ratcheting will require an elastic-plastic analysis of the component.

When using the elastic-plastic method for evaluating protection against ratcheting the loadings are applied and removed in a cycle. If the progression of the stress-strain hysteresis loop is stabilized protection against ratcheting is considered fulfilled.

Assessment procedure

Step 1:

A numerical model including all relevant geometry, applied loads and boundary conditions shall be developed.

Step 2:

All relevant loads and load cases shall be defined.

Step 3:

An elastic-perfectly plastic material model shall be used in the analysis. The effect of non-linear geometry (large deflections) shall be considered. The yield strength for definition of the plastic limit shall be the minimum yield strength at specified temperature.

Step 4:

The elastic-plastic analysis for the loading event that has the highest probability of ratcheting shall be performed for a number of repetitions. See Annex 5. B, Ref /12/.

Step 5:

The following ratcheting criteria shall be evaluated after a minimum of three complete repetition of the applied load cycle. Additional cycles might be required to demonstrate solution convergence. If any of the criteria presented below is met, ratcheting protection is considered fulfilled.

- No plastic action or zero plastic strains in the component.
- The core in the primary load bearing boundary of the component is elastic.
- There is no permanent change in the overall dimensions of the component.
    - Verified by a plot of relevant component dimensions versus cycle time.

CHAPTER 3 DESIGN BASIS

3.1 General

In order to obtain comparable analysis results using the different methods presented in this thesis simple pressure vessel geometry is required. A cylindrical tank with one nozzle is chosen. The nozzle is selected from standard components and attached perpendicular to the centerline of the vessel.

The comparison of heavy wall construction and thin wall construction require two different pressure vessel models. The criteria ($\boldsymbol{{}^{R}\!/_{t} \leq 4}$) establishes the limit between heavy wall construction and thin wall construction.

3.2 Geometrical dimensions

3.2.1 Overall dimensions

- Length: 1000 mm.
- Internal diameter without corrosion and thinning allowance: 500 mm.
- Corrosion allowance: 1,0 mm.
- Material thinning allowance: 1,0 mm.
- Wall thickness:
  - Thin wall configuration: $t < 62{,}5\ mm$.
    - **35 mm** is selected as initial value.
  - Heavy wall configuration: $t \geq 62{,}5\ mm$.
    - **70 mm** is selected as initial value.
- Nozzle data, thin wall configuration:
  - Size: 3” Class 900 lbs
  - Outer diameter: 127 mm
  - Internal diameter: 76,22 mm
  - Wall thickness: 25,39 mm
  - Nozzle standout from vessel OD: 200 mm
  - Reinforcement pad:
    - Diameter: 229 mm
    - Thickness: 25 mm
- Nozzle data, heavy wall configuration:
  - Size: 3” Class 1500 lbs
  - Outer diameter: 133,3 mm
  - Internal diameter: 76.16 mm
  - Wall thickness: 28,57 mm
  - Nozzle standout from vessel OD: 200 mm
  - Reinforcement pad:
    - Diameter: 235,3 mm
    - Thickness: 40 mm

3.2.2 Drawings

The dimensions for the thin wall pressure vessel including corrosion and thinning allowance are illustrated by the drawing[4] presented in Figure 44, while the dimensions for the heavy wall pressure vessel are illustrated by the drawing[4] in Figure 45. These drawings were used as basis for the full set of construction drawings that are presented in APPENDIX A.

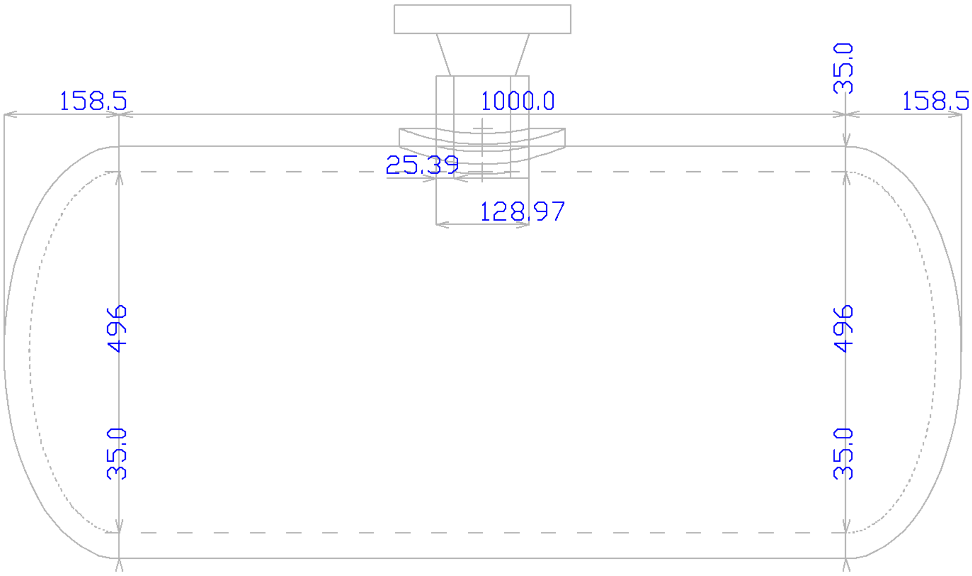


Figure 44: Drawing of thin wall pressure vessel.

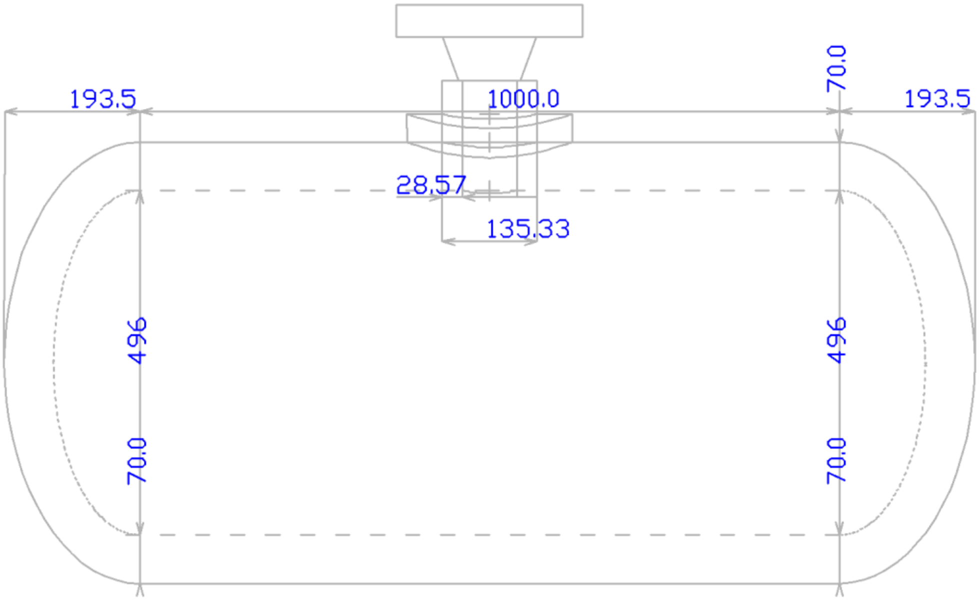


Figure 45: Drawing of heavy wall pressure vessel.

---

[4] The drawings are produced with AutoCAD Mechanical 2011.

3.3 Material properties

The material considered used is SA-516 Grade 70, UNS K02700 with main properties as specified in Figure 46. Material data are obtained from, Ref /13/ table 5A.

Young`s Modulus, E [GPa]	210
Poisson`s ratio, ν [-]	0,3
Mass density, ρ [kg/m^3]	7850
Min. Yield strength, S_y [MPa]	260
Min. Ultimate strength, S_u [MPa]	485

Figure 46: Material properties.

3.4 Loads and load cases

The load cases to be considered during the analyses are listed in Figure 47.

Description	Thin wall vessel	Heavy wall vessel
Internal operating pressure	100 bar (10 MPa)	200 bar (20 MPa)
Nozzle load (unfavorable)	30 000 N	30 000 N
Pressure high	100 bar (10 MPa)	200 bar (20 MPa)
Pressure low	P_{ATM} = 1 bar (0,1 MPa)	P_{ATM} = 1 bar (0,1 MPa)
Temperature high	100 °C	100 °C
Temperature low	20 °C	20 °C
Max number of cycles	500	500

Figure 47: Loads and load cases.

3.5 Acceptance Criteria

In order to obtain proper comparison basis between the different analysis methods the results of each individual test shall be given as the utilization factor:

$$\frac{\boldsymbol{Analysis\ Result}}{\boldsymbol{Design\ Limit}}\ \boldsymbol{if}\ (>1\ fails)$$

Both the analysis result and the corresponding design limit are obtained using the same method. The respective design check of the method being investigated is accepted if the ratio is less than one (1). The final result including all methods shall be presented in table format for easy comparison.

CHAPTER 4 ANALYSIS TOOLS

4.1 Visual vessel design

Visual Vessel Design (VVD) computer software is used for mechanical design of pressure vessels. The program utilizes calculation formulas included in the following national codes and standards, Ref /16/:

- American Standard, ASME VIII Div. 1
- British Standard, PD5500
- European Code, EN13445
- Swedish Standard, TKN
- Norwegian Standard, TBK2

Visual vessel design is meant to be a time saving tool for the design by formula route. In this thesis the program is used to verify and check the parameters included in the design basis section. The minimum required wall thickness might also be calculated with VVD.

The established design basis and vessel geometry are used to construct the finite element model for the design by analysis methods considered in this thesis.

4.1.1 Thin wall configuration

The maximum utilization factor is calculated to 0,597 for the nozzle. Hence the thin wall configuration is acceptable according to the DBF route. The utilization chart for the components is presented in Figure 48, while the complete calculation report is available in APPENDIX B.

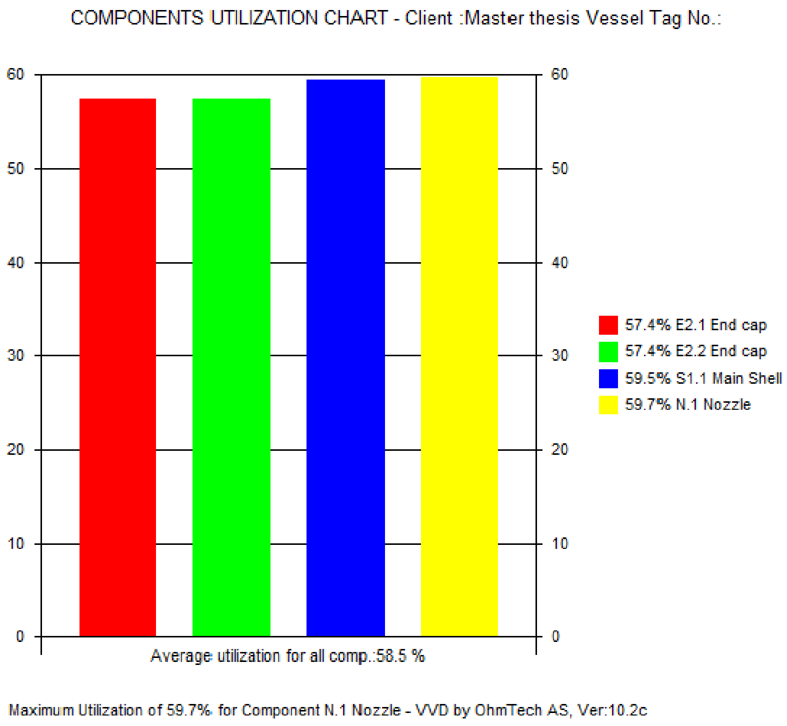


Figure 48: Component utilization chart (thin wall).

4.1.2 Heavy wall configuration

The maximum utilization factor is calculated to 0,595 for the main shell. Hence the heavy wall configuration is acceptable according to the DBF route. The utilization chart for the components is presented in Figure 49, while the complete calculation report is available in APPENDIX C.

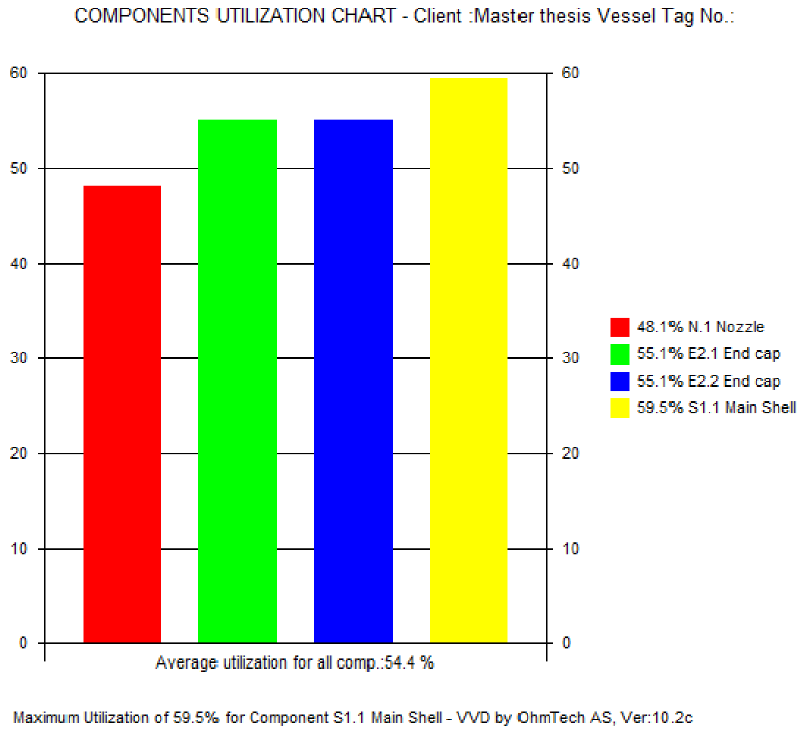


Figure 49: Component utilization chart (heavy wall).

Another important notice to consider is that due to the heavy wall configuration of this pressure vessel the condition of applicability for the equations needed for nozzle load calculations is NOT fulfilled.

$$\mathbf{0,001} < \frac{\boldsymbol{ea}}{\boldsymbol{D}} < 0{,}1$$

Where the quantity $(\frac{ea}{D})$ is the shell analysis thickness divided by the mean diameter of the vessel. The analysis thickness considers the corrosion allowance.

$$\frac{\mathbf{70-1-1}}{\mathbf{500}+(\boldsymbol{ea})} = \mathbf{0.119}$$

Outside this range the effects of torsion moment are significant, Ref /6/.

In this case the analysis is considered valid due to:

- There is no torsion applied to the nozzle.
- The condition of applicability is barely over the limit.

4.2 ANSYS workbench

In order to conduct analysis of the pressure vessel in ANSYS workbench a 3D model of the vessel geometry must be constructed. A 3D model of both thin and heavy wall configuration is required. The model should be as accurate as possible to obtain result close to the real structural behavior under the imposed loads.

A 3D model of the pressure vessels is presented in Figure 50. The thin wall configuration is shown on top, while the heavy wall configuration is shown at the bottom.

The 3D models are constructed with Autodesk Inventor 2011, Ref /17/.

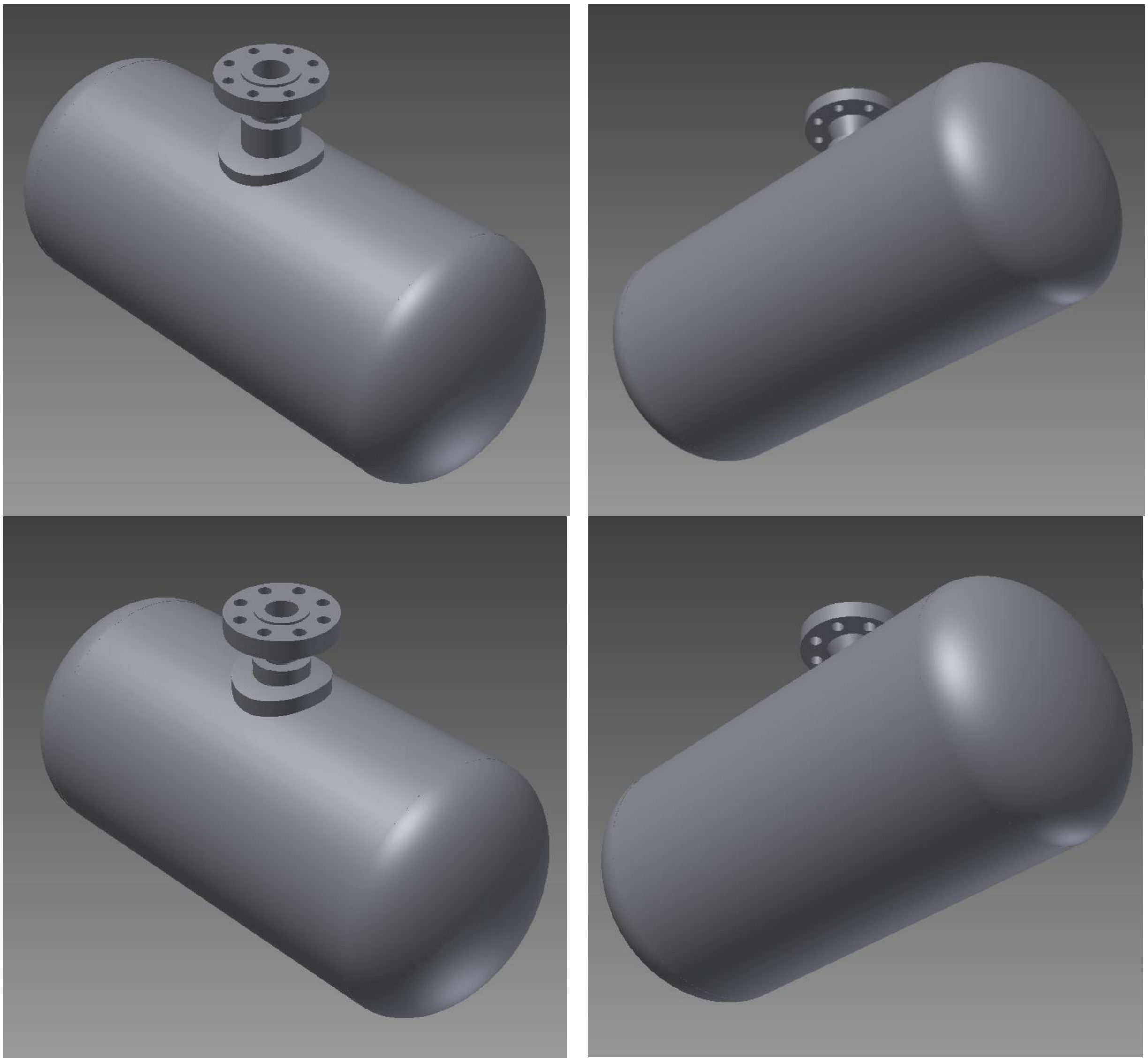


Figure 50: 3D model of the pressure vessels.

Welds were added between the reinforcement pad and the main shell, between the reinforcement pad and the nozzle and between the nozzle and the flange to further enhance the model accuracy. The weldment details are shown in Figure 51 for the heavy wall pressure vessel and in Figure 52 for the thin wall pressure vessel.

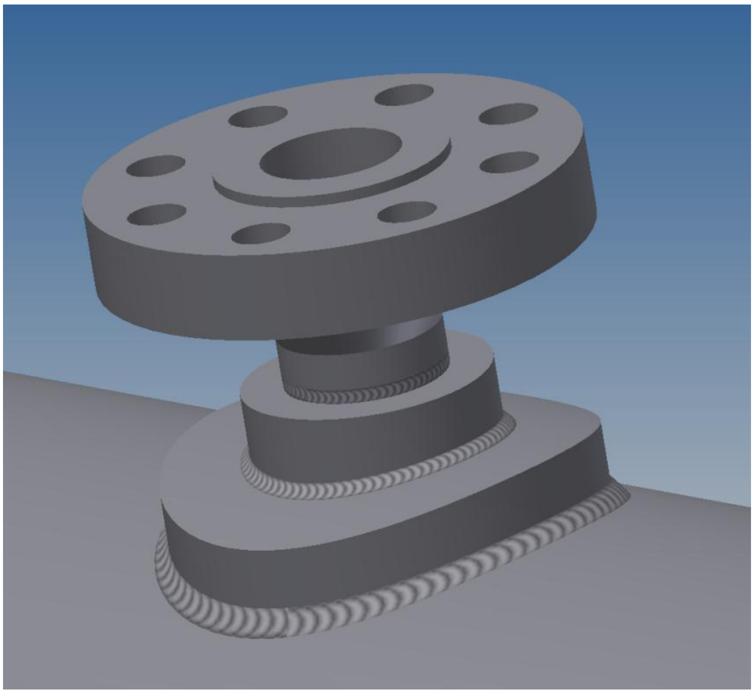


Figure 51: Weldment details for the heavy wall pressure vessel.

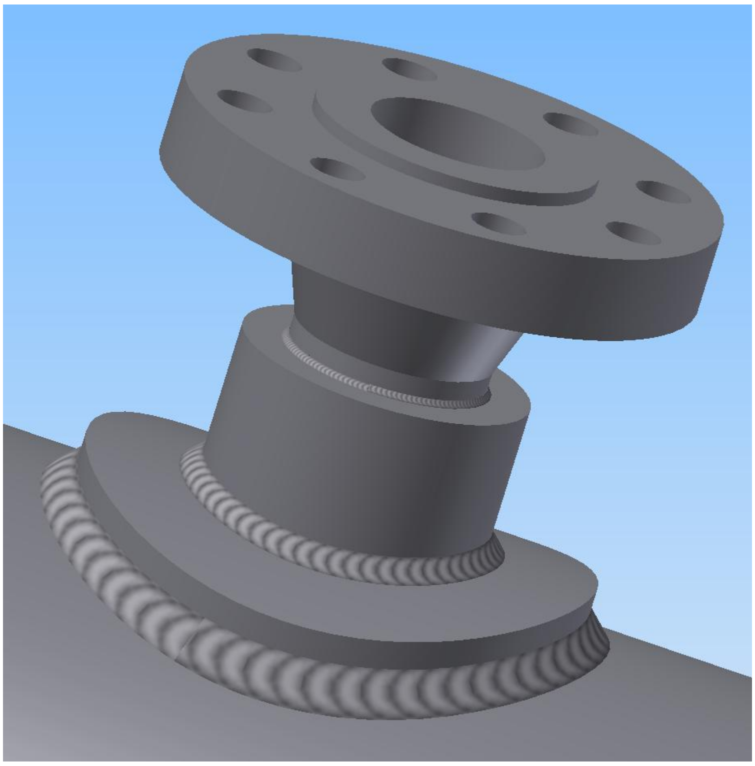


Figure 52: Weldment details for the thin wall pressure vessel.

These models are then used to create the finite element models for further analysis in ANSYS Workbench. The process of building a finite element model is called meshing. The initial mesh for the thin wall pressure vessel is shown in Figure 53, and the mesh for the heavy wall pressure vessel is shown in Figure 54.

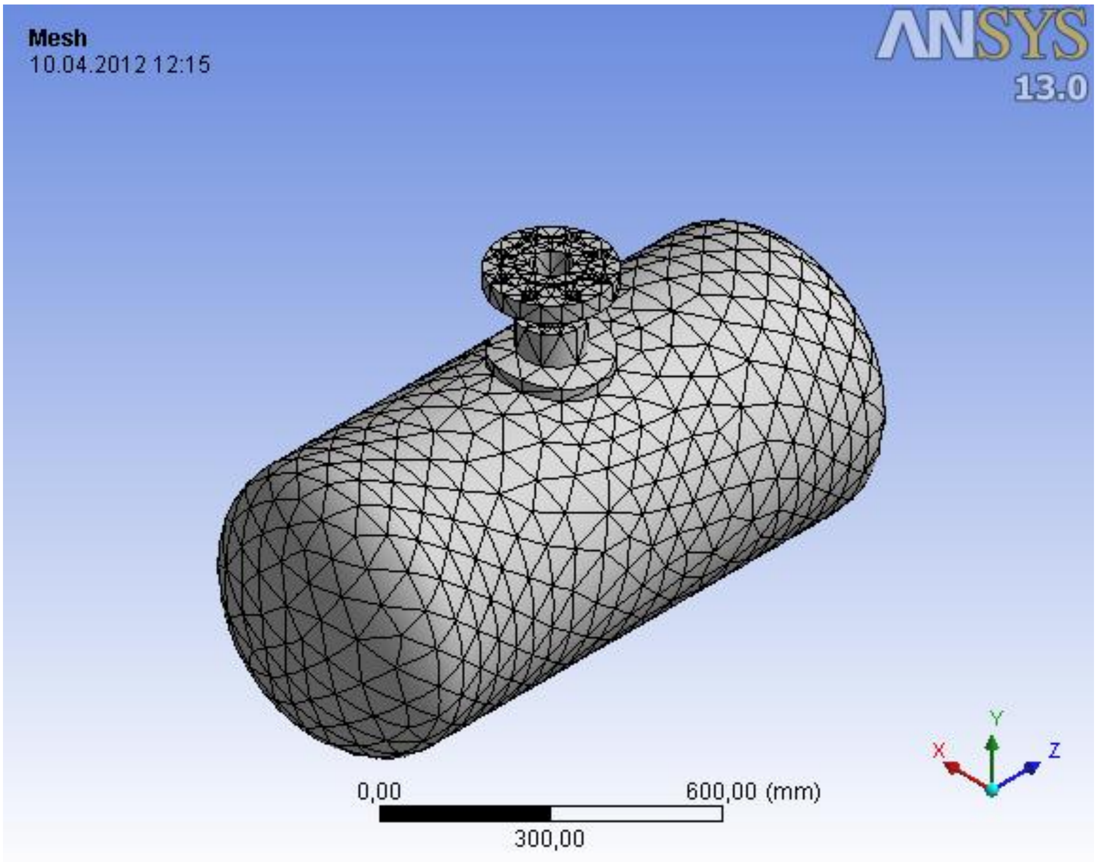


Figure 53: Mesh for the thin wall configuration generated in ANSYS workbench.

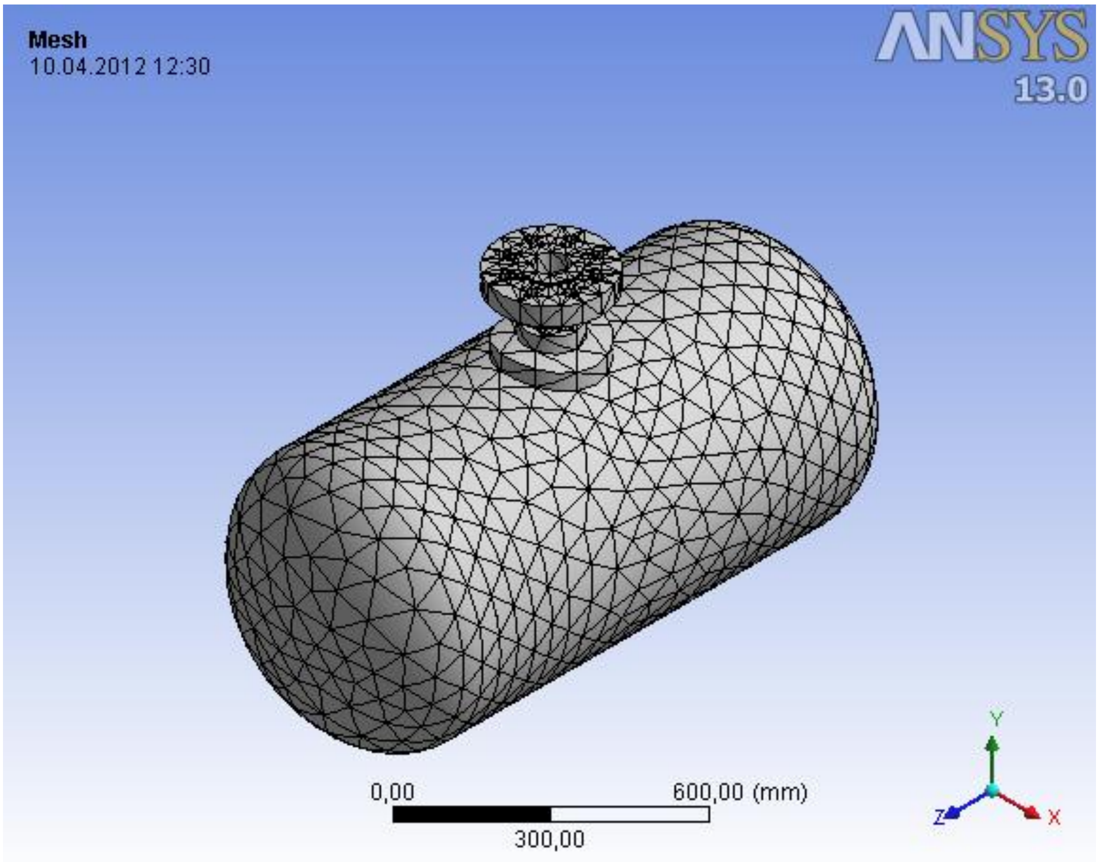


Figure 54: Mesh for the heavy wall configuration generated in ANSYS workbench.

In order to reduce the numerical calculations required to solve the finite element model symmetry and boundary conditions were applied. The model was cut in half and the mesh refined as shown in Figure 55 and Figure 56 for the thin wall pressure vessel and in Figure 57 and Figure 58 for the heavy wall pressure vessel.

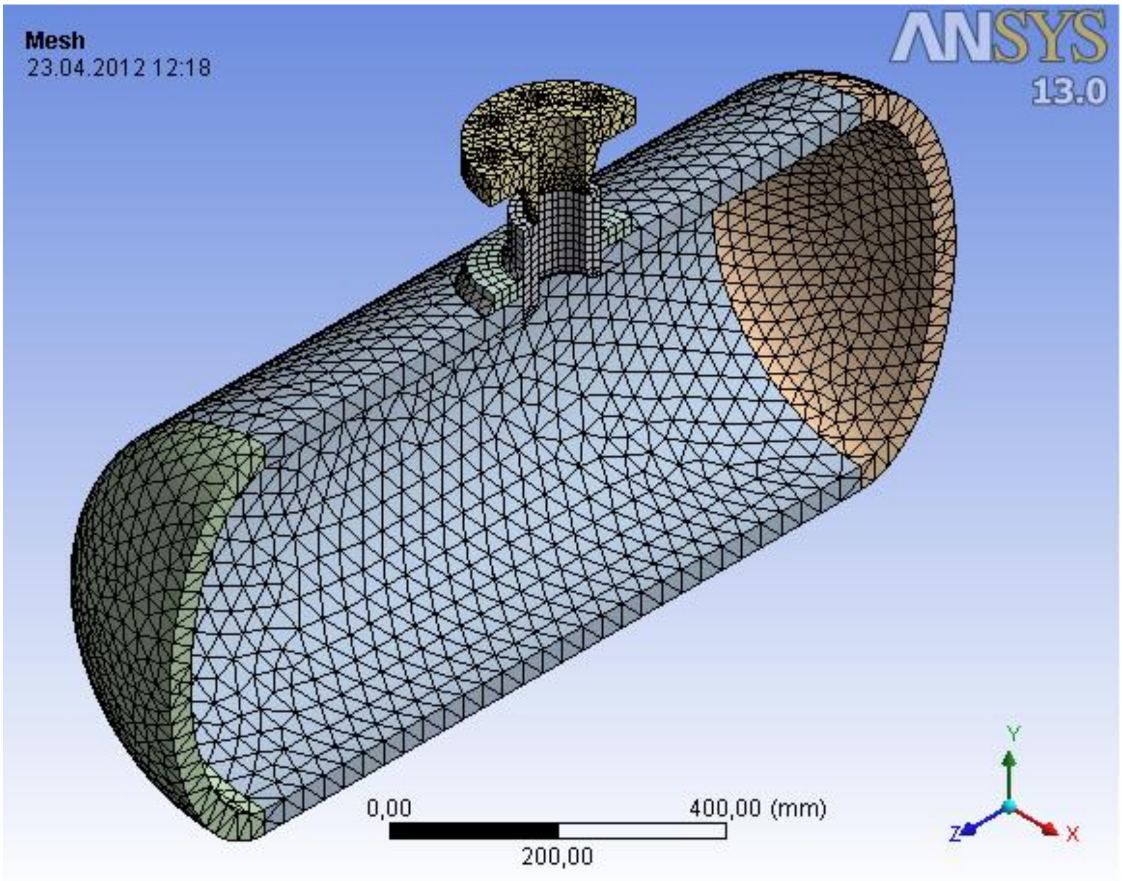


Figure 55: Refined mesh for the thin wall pressure vessel.

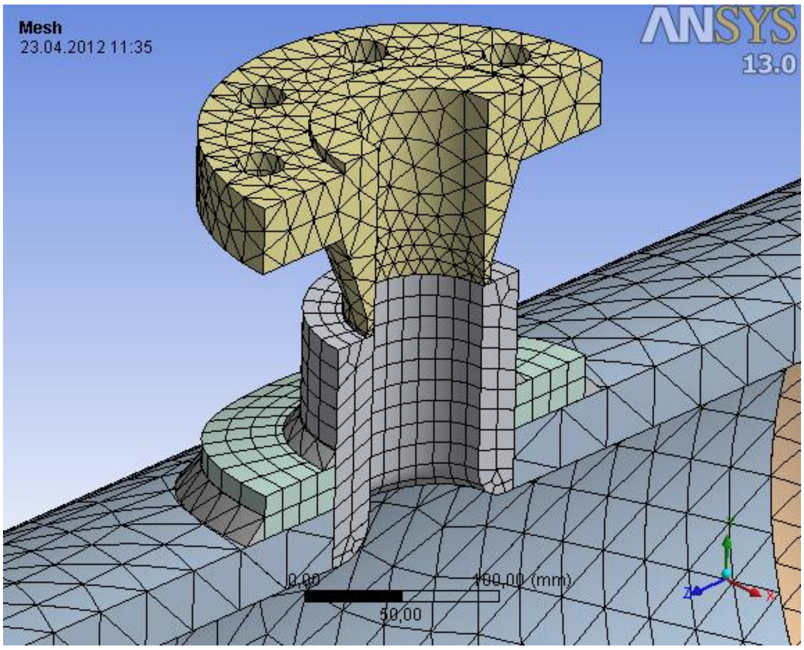


Figure 56: Mesh detail for the nozzle geometry including welds for the thin wall pressure vessel.

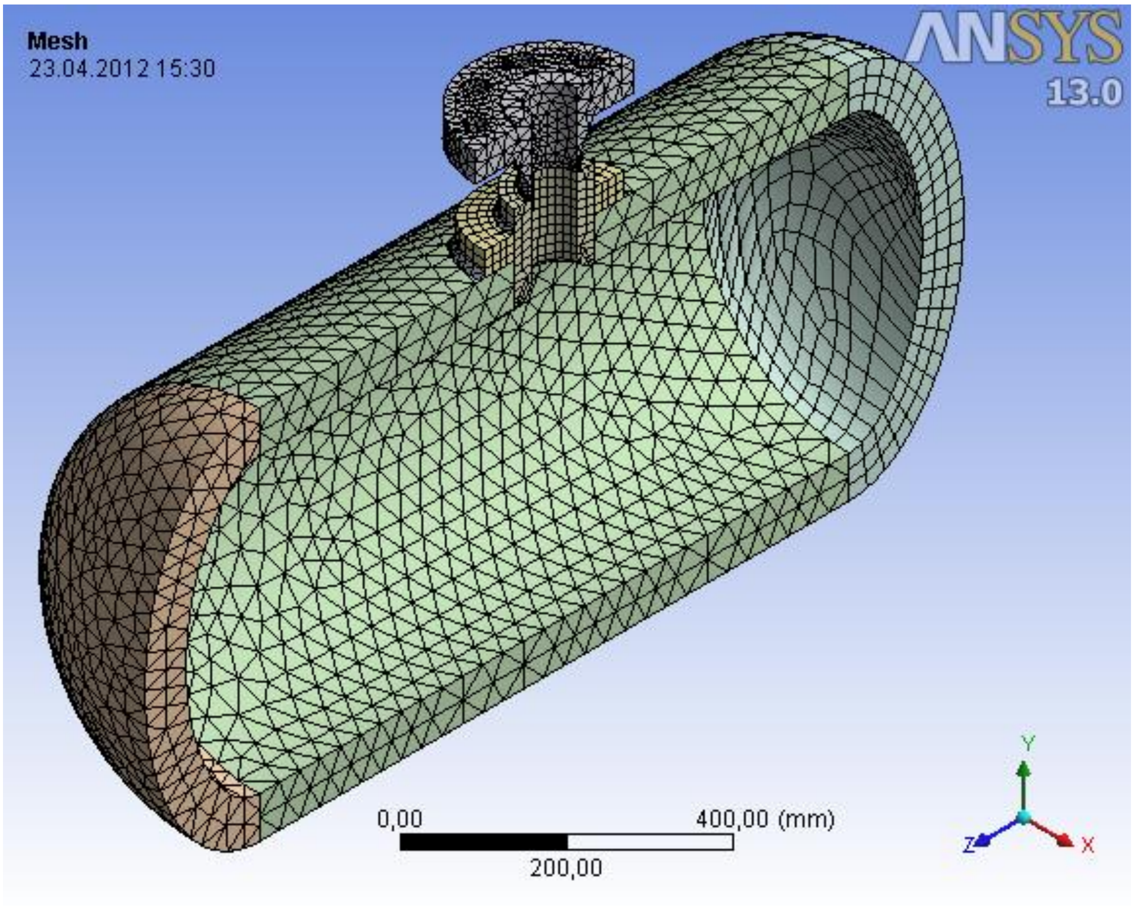


Figure 57: Refined mesh for the heavy wall pressure vessel.

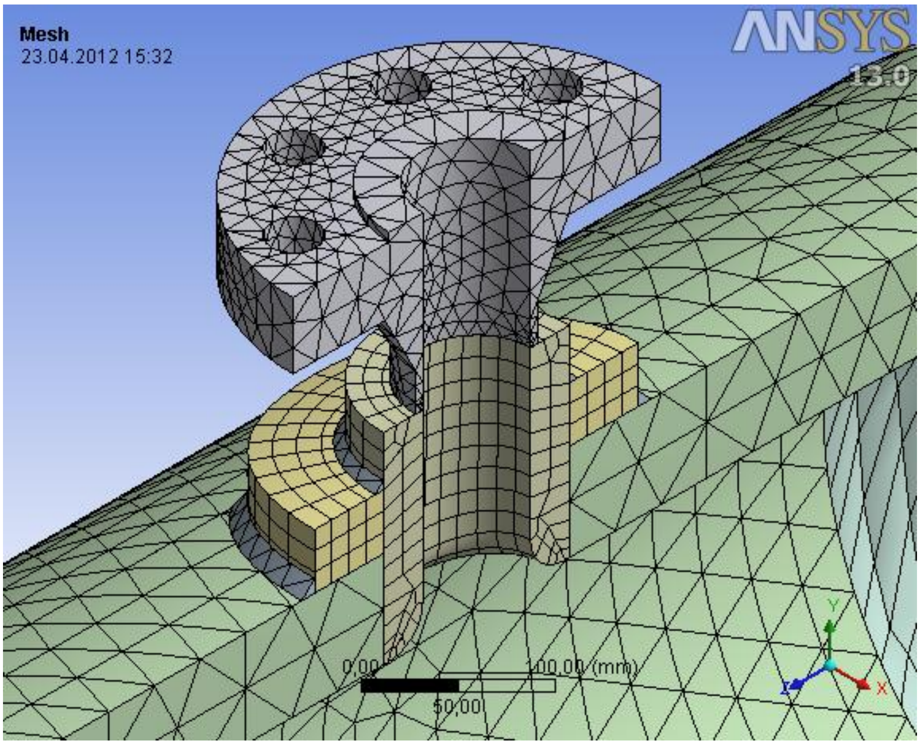


Figure 58: Mesh detail for the nozzle geometry including welds for the heavy wall pressure vessel.

CHAPTER 5 CALCULATIONS

5.1 Direct Route – NS-EN 13445; 2009

The calculations are carried out according to section "2.1 Direct Route – NS-EN 13445; 2009" in this thesis.

5.1.1 Thin wall construction

5.1.1.1 Design value for action effects

The internal operating pressure (pressure high) is classified as the "Reasonably foreseeable highest pressure" and denoted with the partial safety factor for "action without natural limit".

Hence the design effect for the operating pressure (pressure high) is:

$$\boldsymbol{E_d = P_{sup} \times \gamma_P = 10\ MPa \times 1,2 = 12\ MPa}$$

The "Reasonably foreseeable lowest pressure" is the atmospheric pressure with partial safety factor "action with a natural limit".

$$\boldsymbol{E_d = P_{inf} \times \gamma_P = 0,1\ MPa \times 1,0 = 0,1\ MPa}$$

The following action effects for temperature are identified:

$$\boldsymbol{T_{sup} = 100}℃$$

$$\boldsymbol{T_{inf} = 20}℃$$

The nozzle load is classified as" permanent action" with partial safety factor "action with unfavorable effect".

Hence the design effect for the nozzle load is:

$$\boldsymbol{E_d = G_k \times \gamma_G = 30\ kN \times 1,2 = 36\ kN}$$

The vessel weight is classified as" permanent action" with partial safety factor "action with unfavorable effect".

Hence the design value for earth gravity is:

$$\boldsymbol{E_d = G_k \times \gamma_G = 9,806\ {}^{m}/_{s^2} \times 1,2 = 11,767\ {}^{m}/_{s^2}}$$

The boundary conditions and design values with directions for the imposed actions are shown in Figure 59. Here the frictionless support (C) is fixed in x-direction and accounts for the symmetry conditions. The displacement (D) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (E) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (F) is half the total nozzle load due to symmetry conditions.

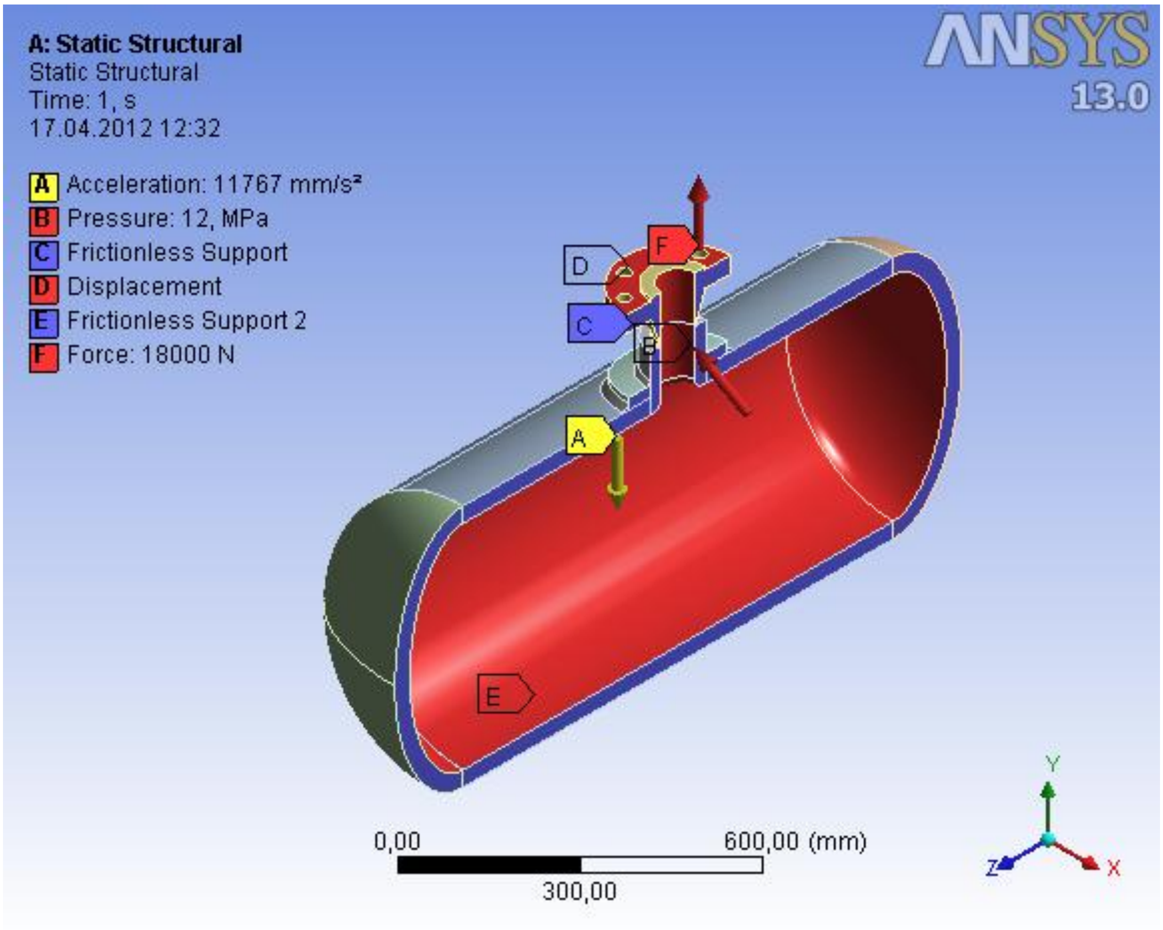


Figure 59: Design values and directions for the action effects.

5.1.1.2 Design resistance for the material

The material strength parameter is:

$$RM = R_{eH} = 260\ MPa$$

Partial safety factor for the material is:

$$\gamma_R = 1{,}25$$

Hence the expression for the design resistance:

$$R_d = \frac{RM}{\gamma_R} = \frac{165\ MPa}{1{,}25} = 208\ MPa$$

5.1.1.3 Gross plastic deformation

The stress intensity (Tresca criterion) is used, and the maximum value of stress is calculated to 258,54 MPa at the point where the nozzle is attached to the main shell as illustrated in Figure 60. The full calculation report from ANSYS is available in APPENDIX D.

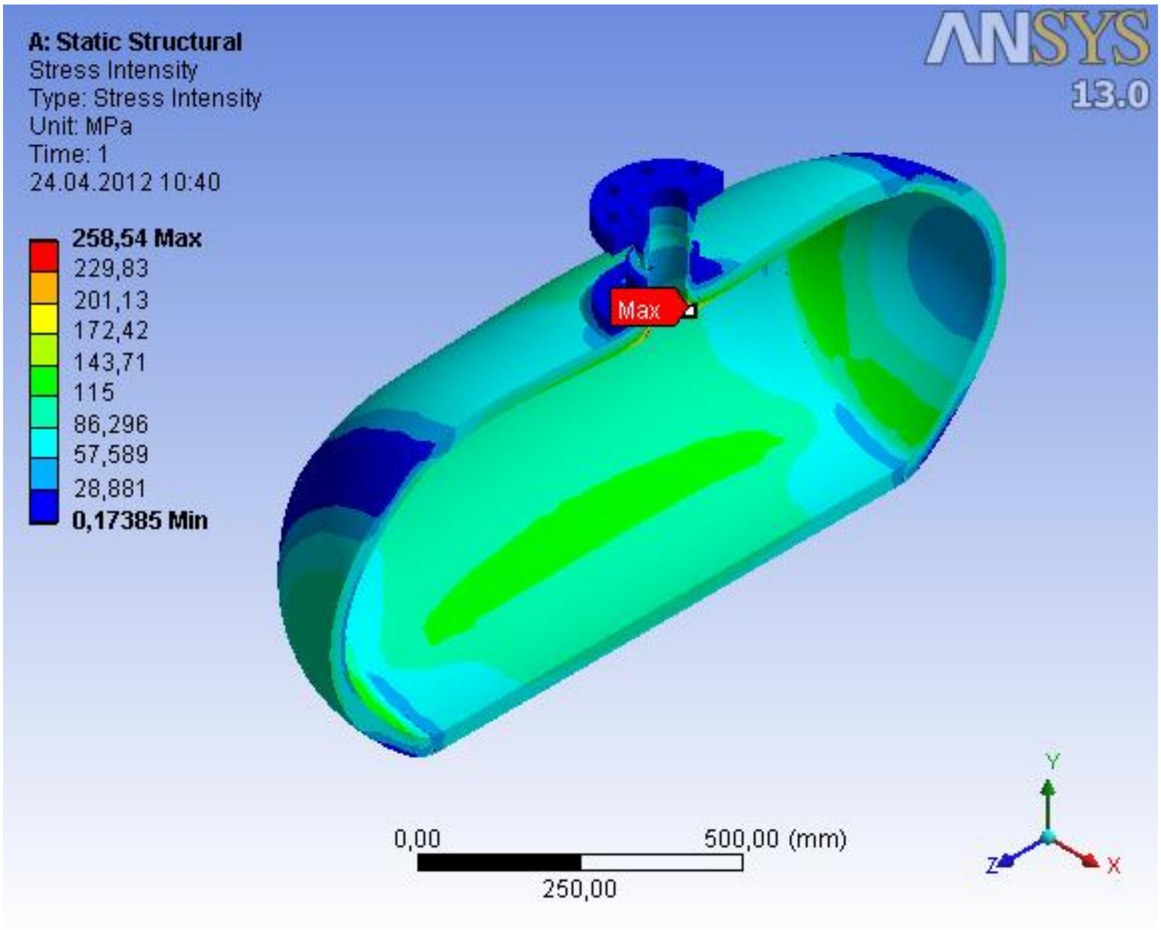


Figure 60: Stress intensity plot.

The value of 258,54 MPa is above the design limit of 208 MPa, indicating that the design might not be acceptable according to the Gross Plastic Deformation design check. Careful engineering judgment is required to evaluate the results.

By investigating Figure 61 and Figure 62 where only areas above the design limit of 208 MPa are shown the conclusion is that the model is fully capable of carrying the design values of the combined action effects. The zones above the design limit are still local, and the model will not sustain gross plastic deformation.

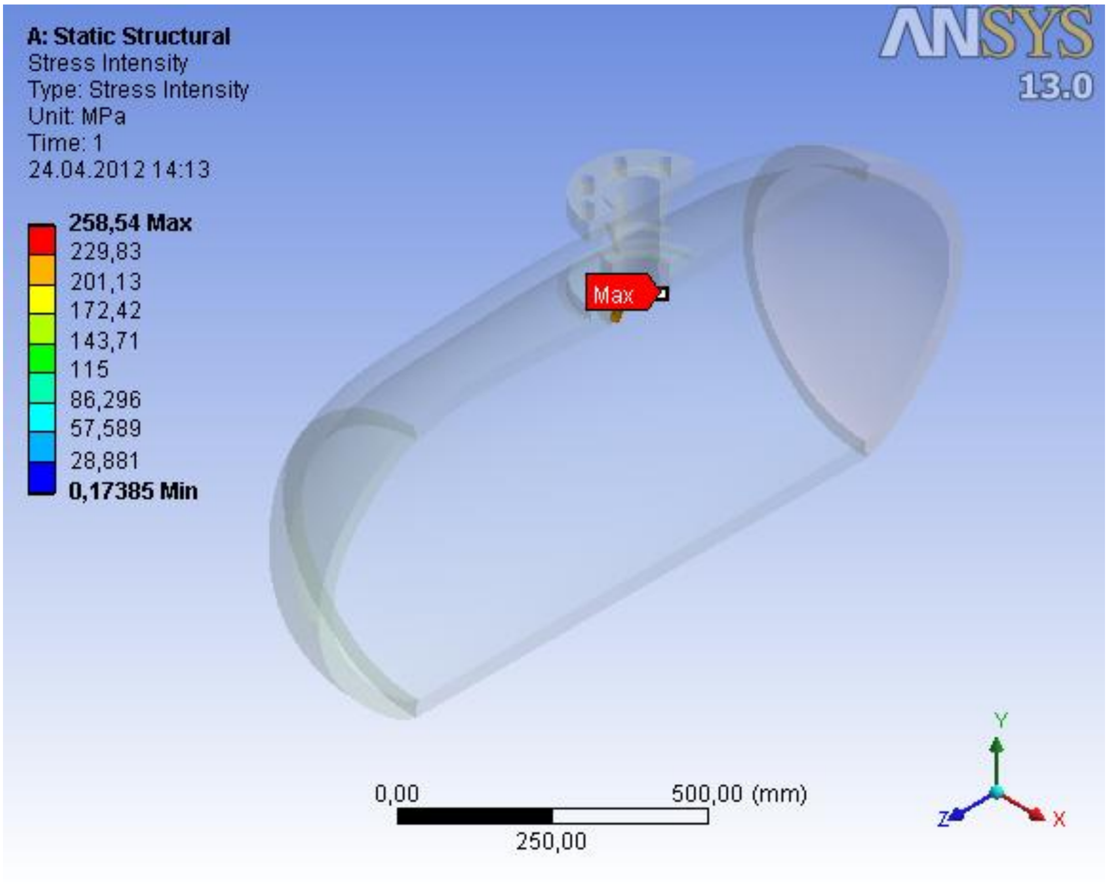


Figure 61: Stress plot of areas above the design limit.

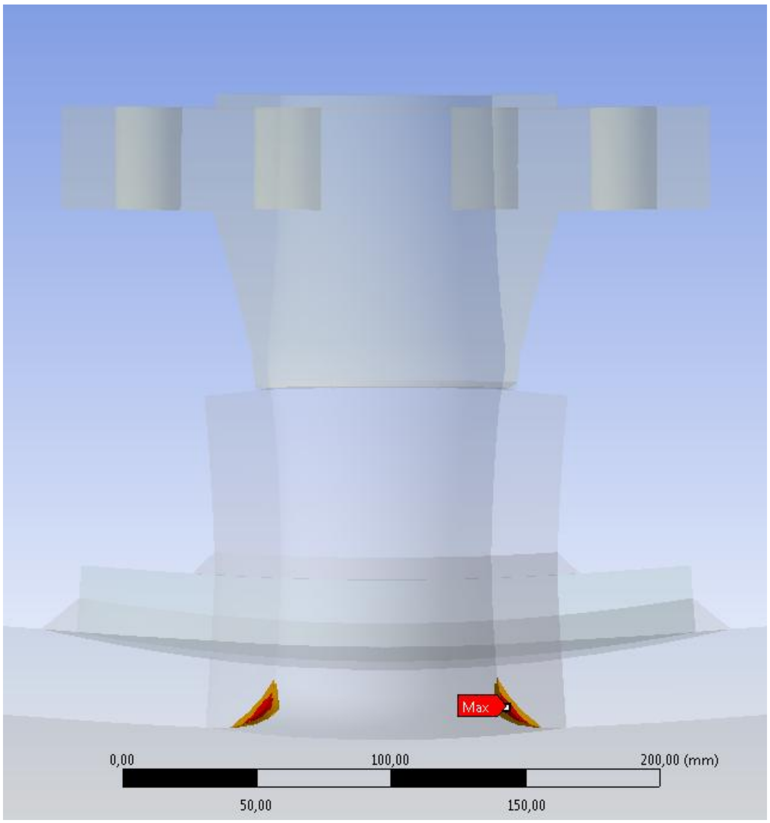
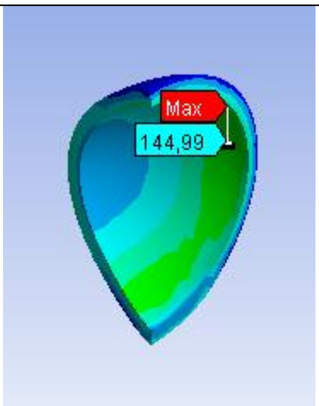


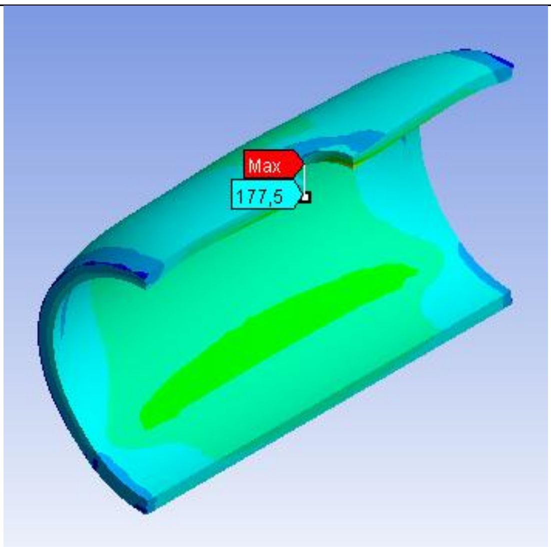
Figure 62: Detail view of areas above the design limit.

The utilization factor:

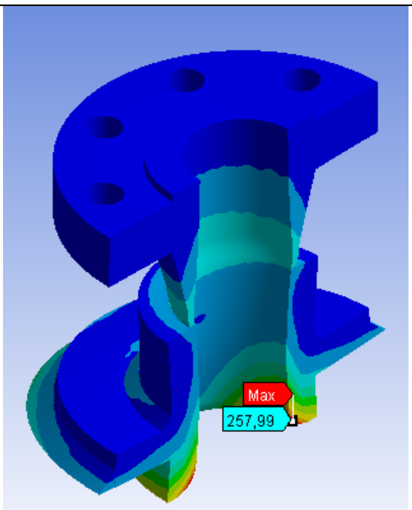
End cap:

	$\frac{\mathbf{\mathit{Analysis\ Result}}}{\mathbf{\mathit{Design\ Limit}}} = \frac{\mathbf{144,99\ \mathit{MPa}}}{\mathbf{208\ \mathit{MPa}}} = \mathbf{0,697}$

Main shell:

	$\frac{\mathbf{\mathit{Analysis\ Result}}}{\mathbf{\mathit{Design\ Limit}}} = \frac{\mathbf{177,5\ \mathit{MPa}}}{\mathbf{208\ \mathit{MPa}}} = \mathbf{0,85}$

Nozzle:

	$\frac{\mathbf{\mathit{Analysis\ Result}}}{\mathbf{\mathit{Design\ Limit}}} = \frac{\mathbf{257,99\ \mathit{MPa}}}{\mathbf{208\ \mathit{MPa}}} = \mathbf{1,24}$

Average utilization factor for the pressure vessel is 0,929.

The maximum value for the principal structural strain is presented in Figure 63.

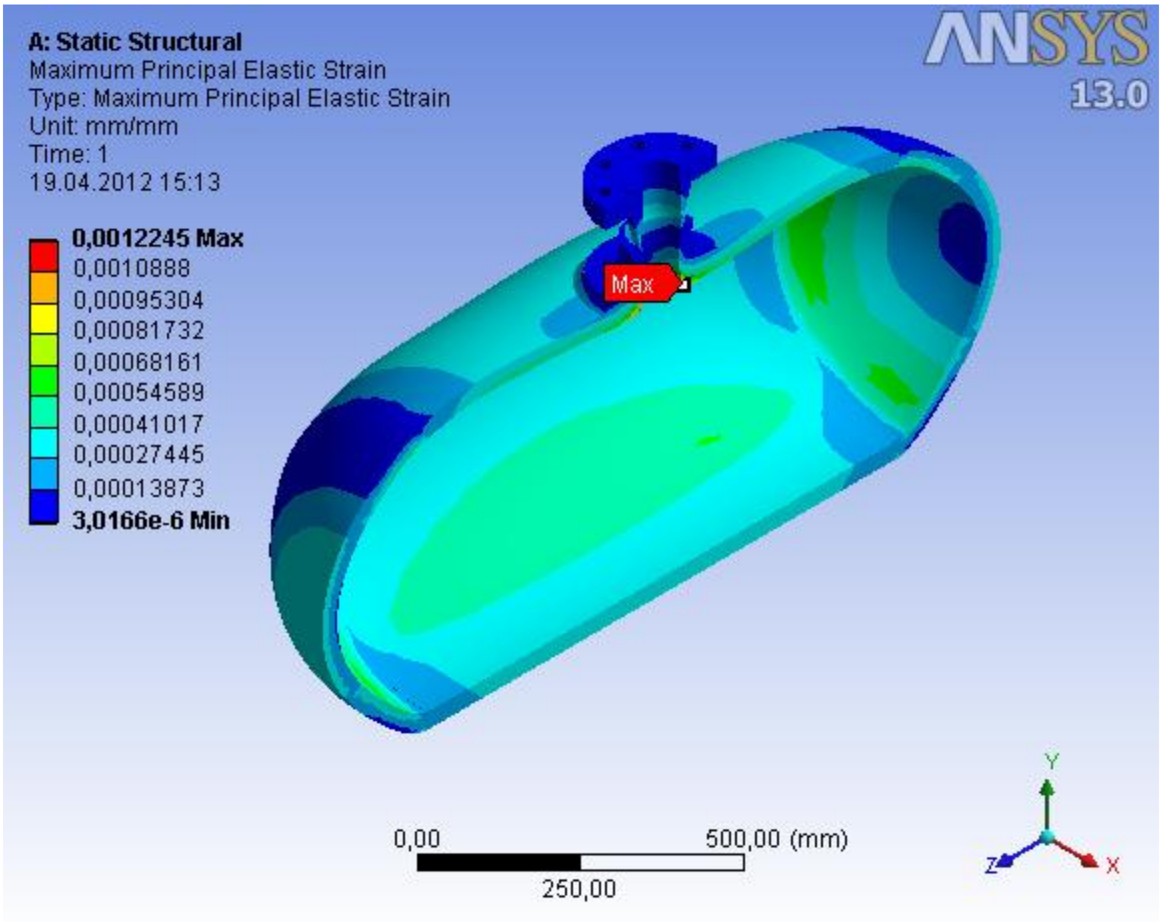


Figure 63: Maximum value for the principal structural strain.

Principal structural strain criterion:

$$\mathbf{0,123\%} < 5\%$$

Hence protection against gross plastic deformation under operating conditions is considered fulfilled according to Direct Route – NS-EN 13445; 2009.

5.1.2 Heavy wall construction

5.1.2.1 Design value for action effects

The internal operating pressure (pressure high) is classified as the “Reasonably foreseeable highest pressure” and denoted with the partial safety factor for “action without natural limit”.

Hence the design effect for the operating pressure (pressure high) is:

$$\boldsymbol{E_d = P_{sup} \times \gamma_P = 20\ MPa \times 1,2 = 24\ MPa}$$

The “Reasonably foreseeable lowest pressure” is the atmospheric pressure with partial safety factor “action with a natural limit”.

$$\boldsymbol{E_d = P_{inf} \times \gamma_P = 0,1\ MPa \times 1,0 = 0,1\ MPa}$$

For the temperature following action effects are identified:

$$\boldsymbol{T_{sup} = 100}℃$$

$$\boldsymbol{T_{inf} = 20}℃$$

The nozzle load is classified as” permanent action” with partial safety factor “action with unfavorable effect”.

Hence the design effect for the nozzle load is:

$$\boldsymbol{E_d = G_k \times \gamma_G = 30\ kN \times 1,2 = 36\ kN}$$

The vessel weight is classified as” permanent action” with partial safety factor “action with unfavorable effect”.

Hence the design value for earth gravity is:

$$\boldsymbol{E_d = G_k \times \gamma_G = 9,806\ {}^{m}/_{s^2} \times 1,2 = 11,767\ {}^{m}/_{s^2}}$$

The boundary conditions and design values with directions for the imposed actions are shown in Figure 64. Here the frictionless support (C) is fixed in x-direction and accounts for the symmetry conditions. The displacement (D) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (E) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (F) is half the total nozzle load due to symmetry conditions.

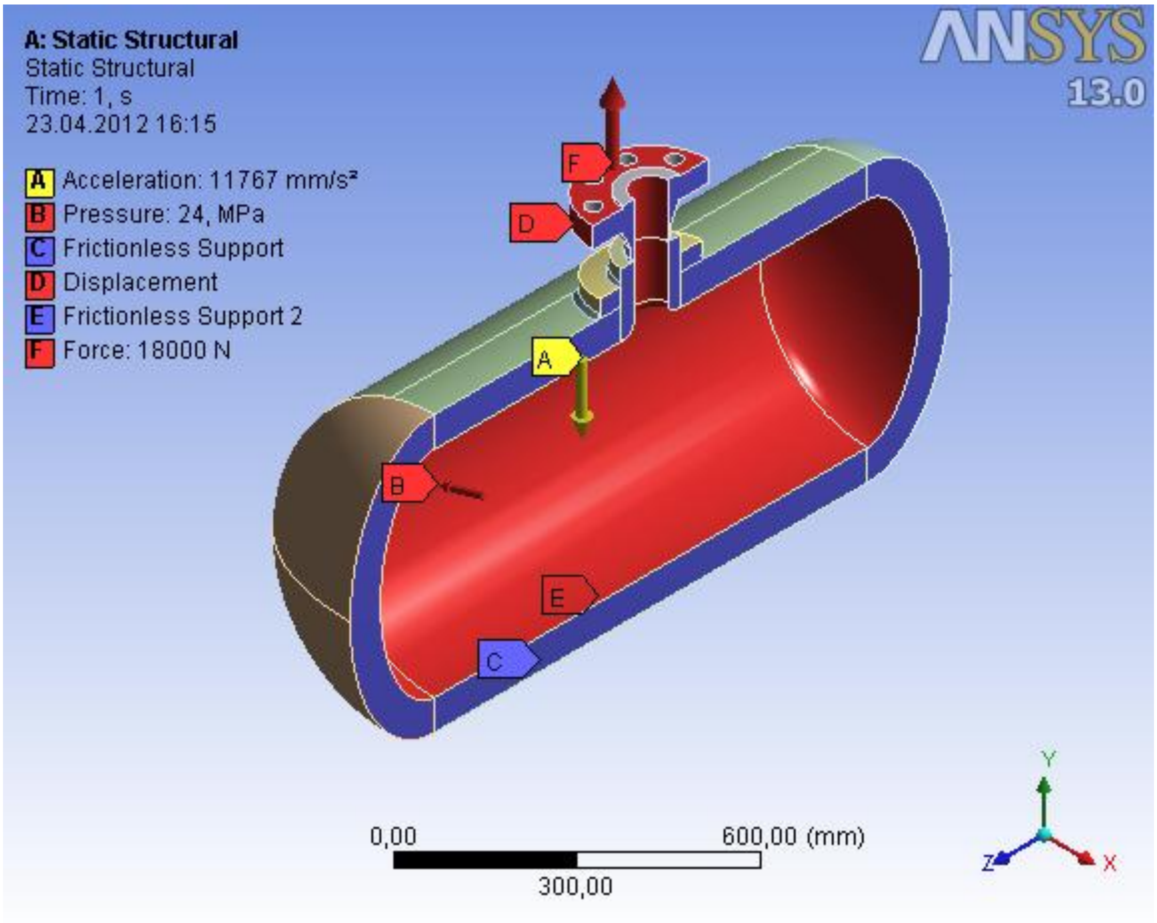


Figure 64: Design values for the action effects.

5.1.2.2 Design resistance for the material

The material strength parameter is:

$$RM = R_{eH} = 260\ MPa$$

Partial safety factor for the material is:

$$\gamma_R = 1{,}25$$

Hence the expression for the design resistance:

$$R_d = \frac{RM}{\gamma_R} = \frac{165\ MPa}{1{,}25} = 208\ MPa$$

5.1.2.3 Gross plastic deformation

The stress intensity (Tresca criterion) is used, and the maximum value of stress is calculated to 326,02 MPa at the point where the nozzle is attached to the main shell as illustrated in Figure 65.The full calculation report from ANSYS is available in APPENDIX E.

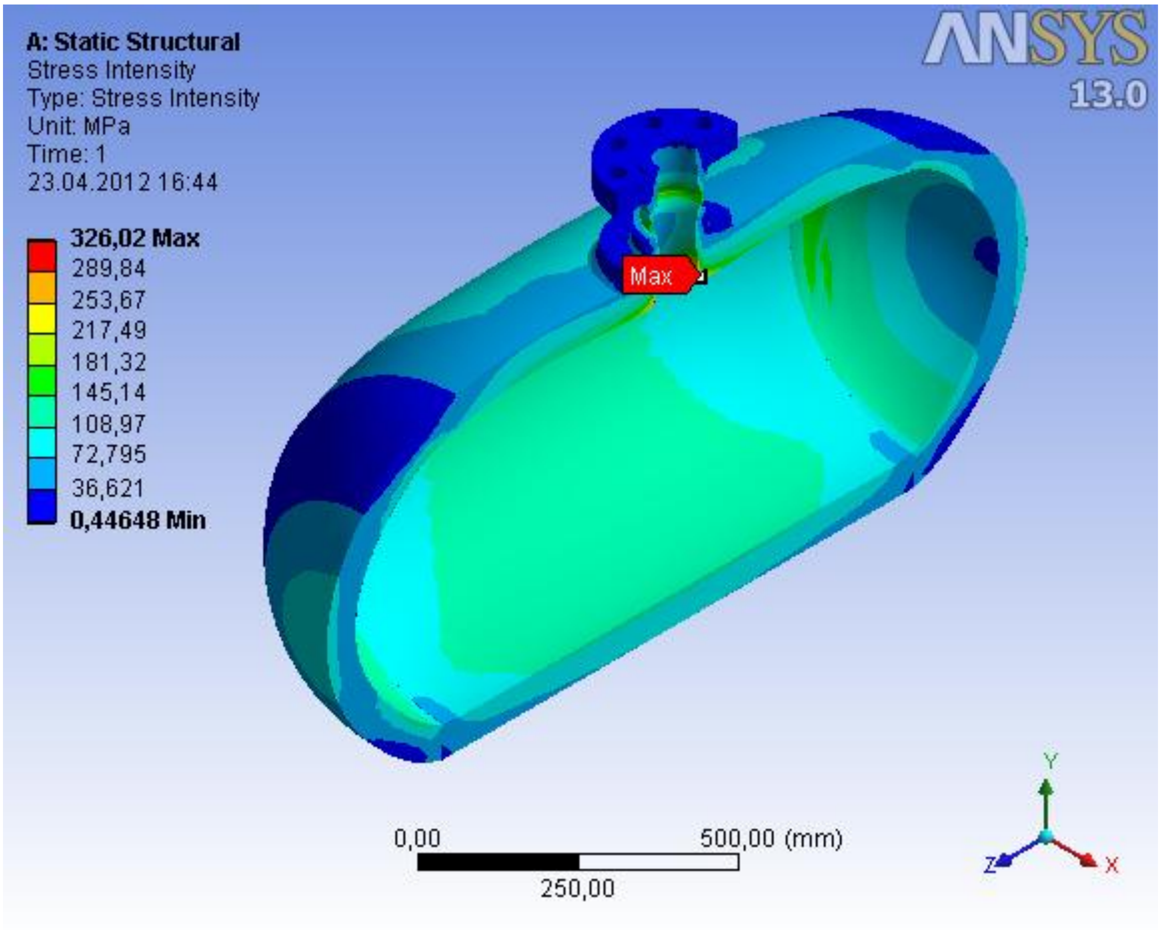


Figure 65: Stress intensity plot.

The value of 326,02 MPa is above the design limit of 208 MPa, in fact it is above the material yield strength of 260 MPa, indicating that the design might not be acceptable according to the Gross Plastic Deformation design check. Careful engineering judgment is required to evaluate the results.

By investigating Figure 66 and Figure 67 where only areas above the design limit of 208 MPa are shown the conclusion is that the model is fully capable of carrying the design values of the combined action effects. The zones above the design limit are still local, and the model will not sustain gross plastic deformation.

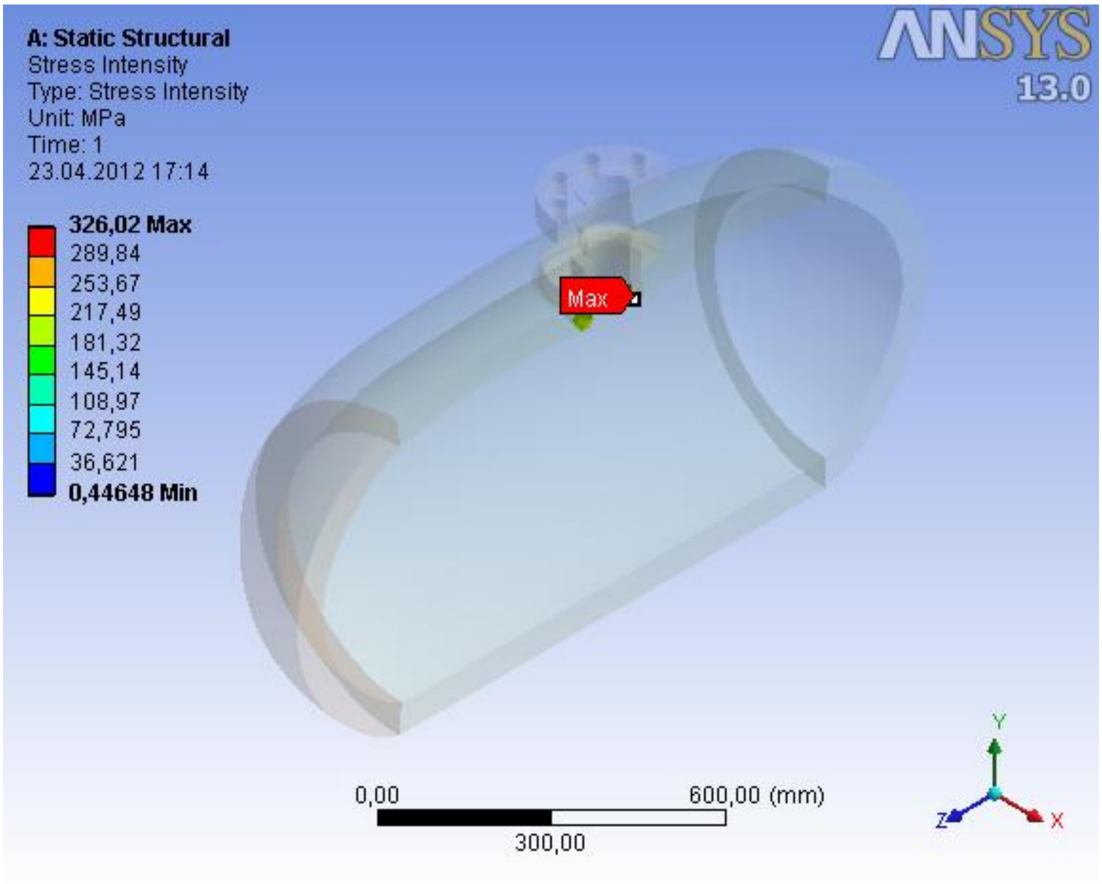


Figure 66: Stress plot of areas above the design limit.

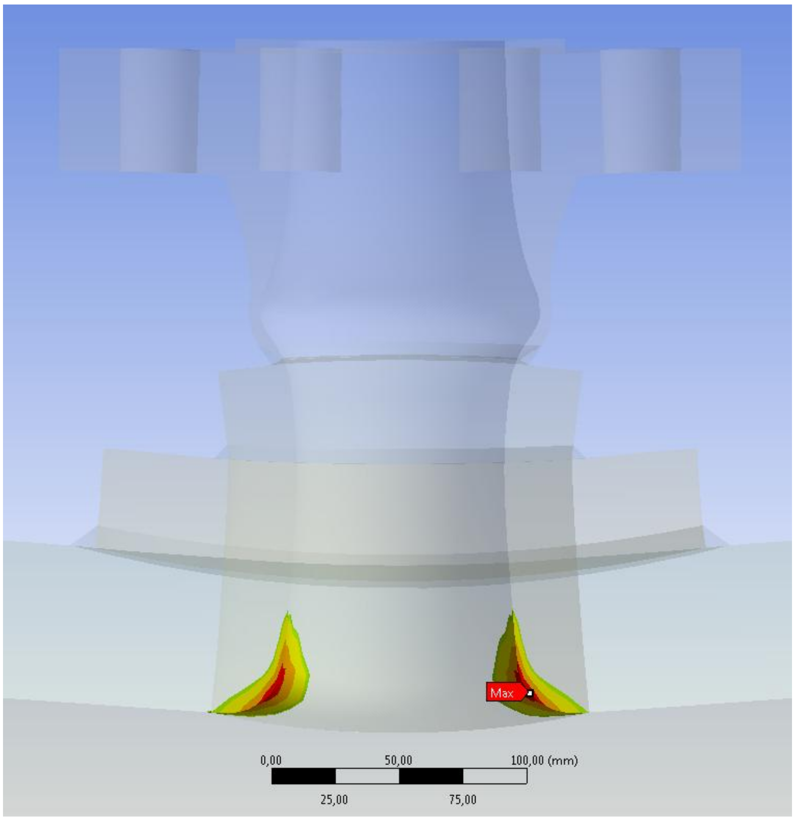
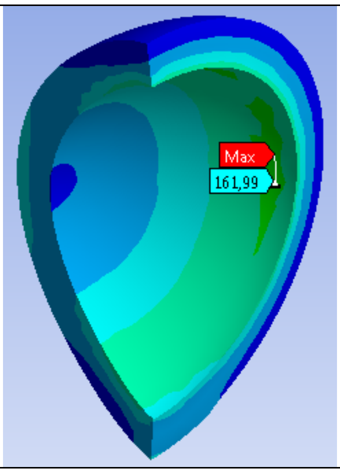


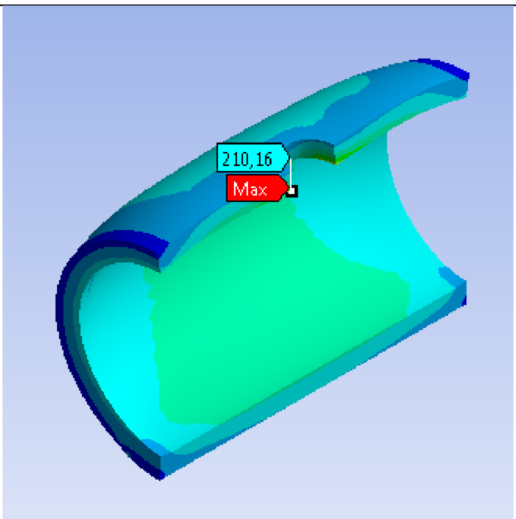
Figure 67: Detail view of areas above the design limit.

The utilization factor:

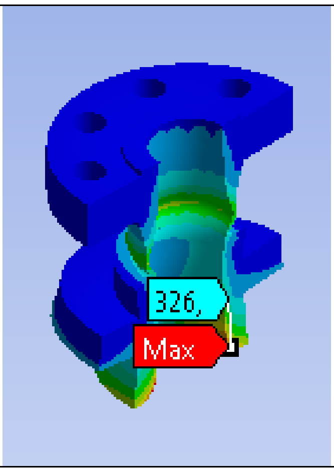
End cap:

	$\frac{Analysis\ Result}{Design\ Limit} = \frac{161{,}99\ MPa}{208\ MPa} = 0{,}779$

Main shell:

	$\frac{Analysis\ Result}{Design\ Limit} = \frac{210{,}16\ MPa}{208\ MPa} = 1{,}01$

Nozzle:

	$\frac{Analysis\ Result}{Design\ Limit} = \frac{326\ MPa}{208\ MPa} = 1{,}57$

Average utilization factor for the pressure vessel is 1,12.

The maximum value for the principal structural strain is presented in Figure 68.

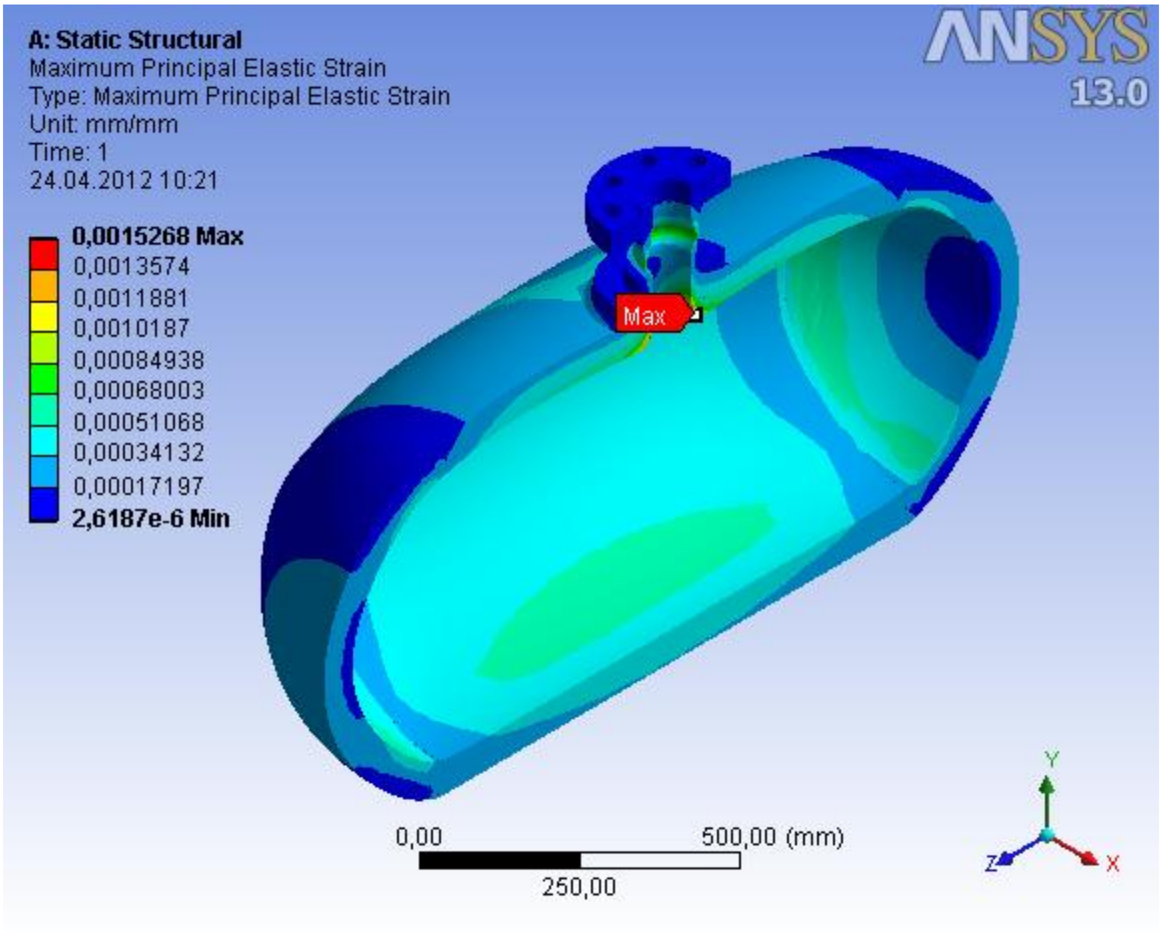


Figure 68: Maximum value for the principal structural strain.

Principal structural strain criterion:

$$\mathbf{0,153}\% < 5\%$$

Hence protection against gross plastic deformation under operating conditions is considered fulfilled according to Direct Route – NS-EN 13445; 2009.

5.2 Elastic Stress Analysis – ASME VIII div.2; 2010

The calculations are carried out according to section "2.2 Elastic Stress Analysis – ASME VIII div. 2; 2010" in this thesis.

5.2.1 Thin wall construction

5.2.1.1 Loading conditions

The loading condition to consider in this case is:

$$\boldsymbol{P + P_s + D + L}$$

Where:

P is the operating pressure of 100 bar.

P$_s$ is the static head from liquid or bulk materials, in this case zero due to the content being gas.

D is the weight of the vessel accounted for by standard earth gravity of 9,8066 m/s^2.

L is the live loading applied to the nozzle of 30000 N.

The loading condition and boundary conditions are illustrated in Figure 69. Here the frictionless support (B) is fixed in x-direction and accounts for the symmetry conditions. The displacement (C) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (D) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (E) is half the total nozzle load due to symmetry conditions.

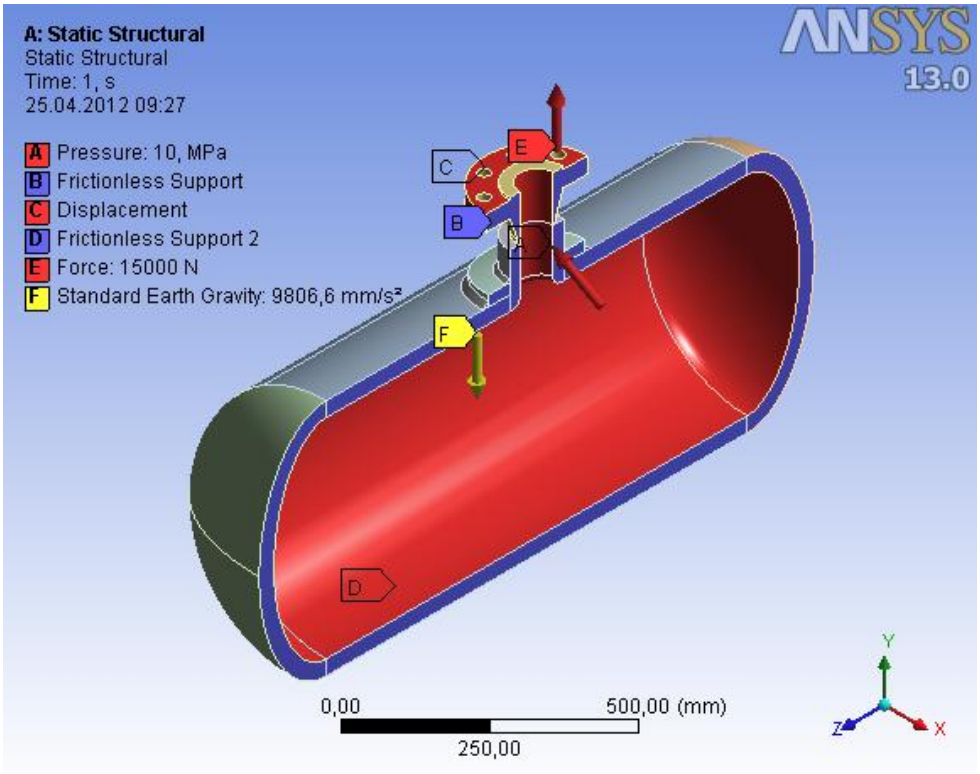


Figure 69: Loading conditions and boundary conditions.

5.2.1.2 Material properties

The maximum allowable stress at maximum anticipated operation temperature for the SA-516 Grade 70, UNS K02700 steel is obtained from table 5A, Ref /13/.

Thus for 100°C maximum allowable stress is:

$$S = 159\ MPa$$

5.2.1.3 Protection against plastic collapse

The stresses are evaluated along the paths illustrated in Figure 70 using the procedure presented in section 2.2.4 Protection against plastic collapse in this thesis. The full calculation report is available in APPENDIX F.

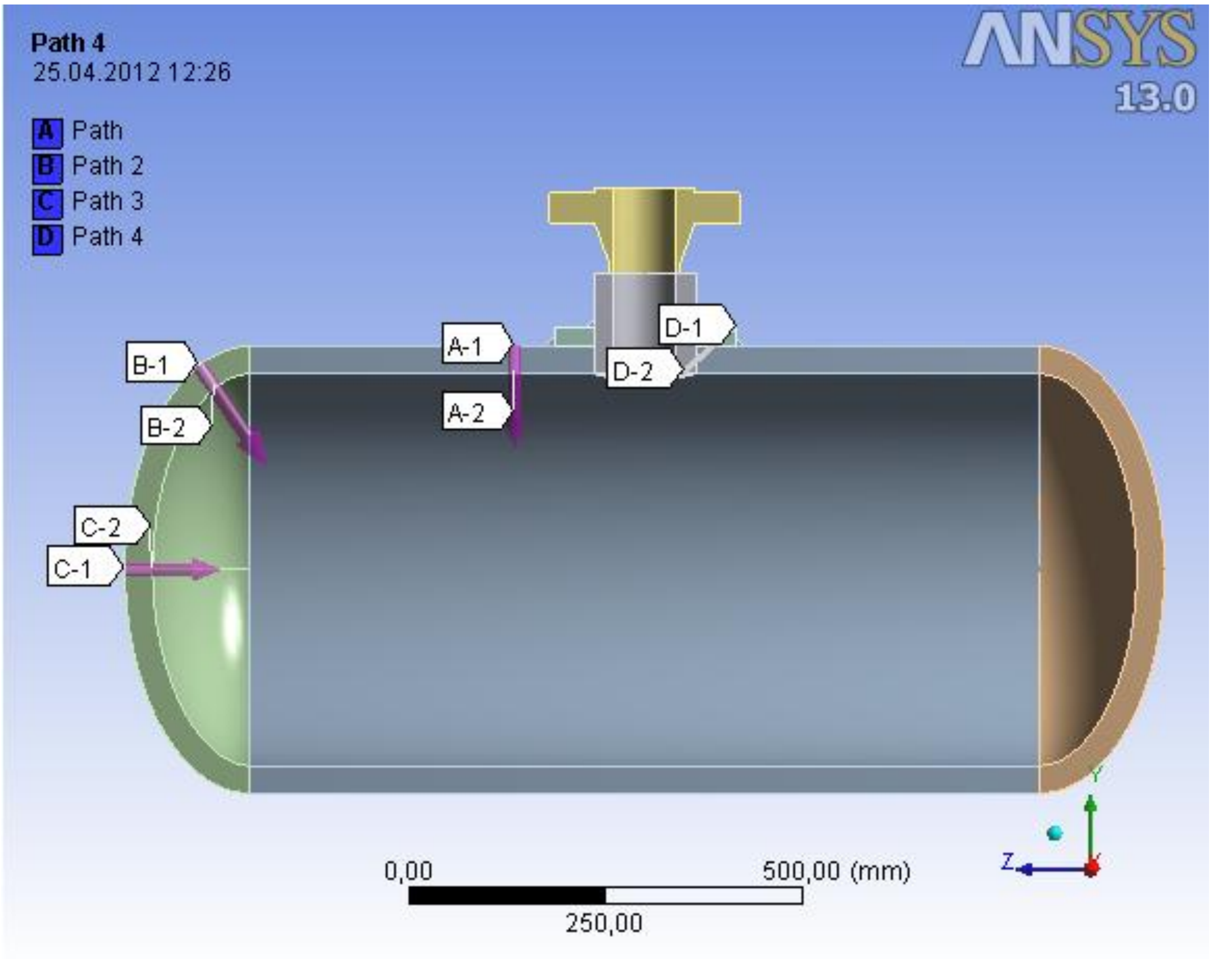


Figure 70: Stress linearization paths.

Here the following paths are used:

- A-1 to A-2 is used for the main shell.
- The most unfavorable of B-1 to B-2 or C-1 to C-2 is used for the end cap.
- D-1 to D-2 is used for the nozzle.

Main shell:

Detailed plot of the path A-1 to A-2 is shown in Figure 71 and the numerical values for the stresses are shown in the chart presented in Figure 72. (Membrane + Bending is almost identical to Total)

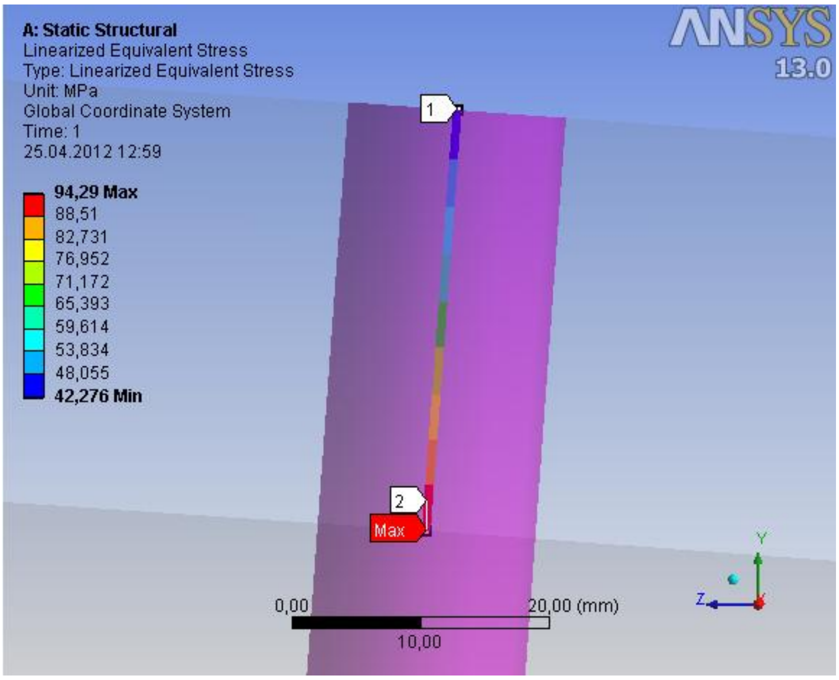


Figure 71: Path A-1 to A-2 for the main shell.

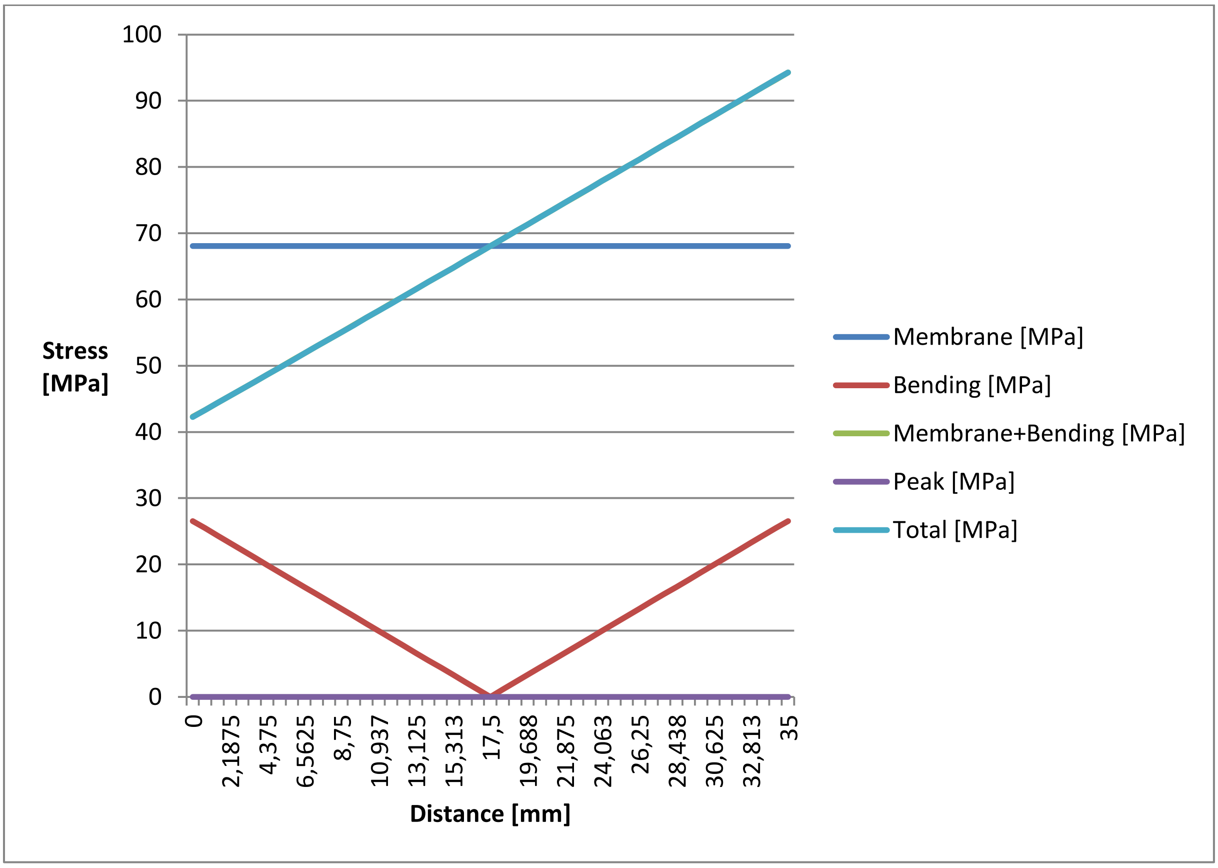


Figure 72: Stress versus distance for path A-1 to A-2.

From this data set the following results are obtained:

$$\boldsymbol{P_m = 68,062\ MPa}$$

$$\boldsymbol{P_L + P_B = 94,278\ MPa}$$

$$\boldsymbol{Max\ total\ stress = 94,29\ MPa}$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$\boldsymbol{P_m \leq S}$$

$$\boldsymbol{P_L \leq 1,5 \times S}$$

$$\boldsymbol{(P_L + P_b) \leq 1,5 \times S}$$

Hence:

$$\boldsymbol{68,062\ MPa \leq 159\ MPa}$$

$$\boldsymbol{94,278\ MPa\ \ \leq 1,5 \times 159\ MPa}$$

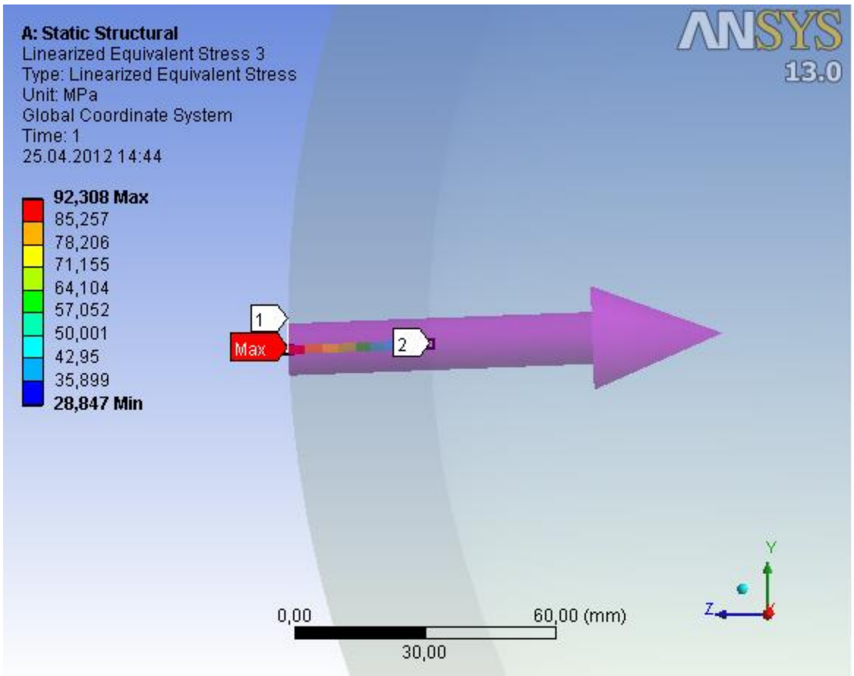
The main shell protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the main shell is:

$$\boldsymbol{\frac{Analysis\ Result}{Design\ Limit} = \frac{94,29\ MPa}{159\ MPa} = 0,593}$$

End cap:

Detailed plot of the path C-1 to C-2 is shown in Figure 73 and the numerical values for the stresses are shown in the chart presented in Figure 72



A: Static Structural
Linearized Equivalent Stress 3
Type: Linearized Equivalent Stress
Unit: MPa
Global Coordinate System
Time: 1
25.04.2012 14:44

Figure 73: Path C-1 to C-2 for the end cap.

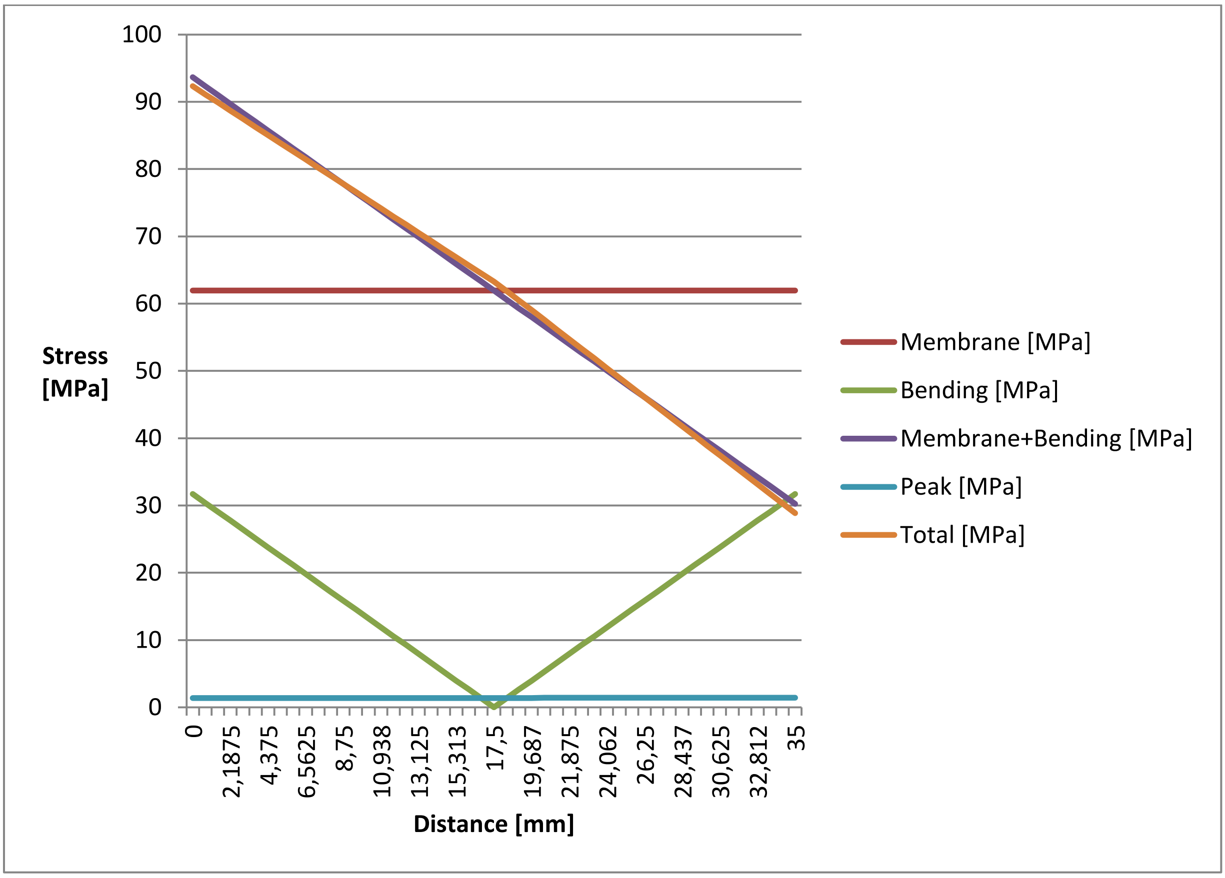


Figure 74: Stress versus distance for path C-1 to C-2.

From this data set the following results are obtained:

$$\boldsymbol{P_m = 61{,}936\ MPa}$$

$$\boldsymbol{P_L + P_B = 93{,}633\ MPa}$$

$$\boldsymbol{Max\ total\ stress = 92{,}308\ MPa}$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$\boldsymbol{P_m \leq S}$$

$$\boldsymbol{P_L \leq 1{,}5 \times S}$$

$$\boldsymbol{(P_L + P_b) \leq 1{,}5 \times S}$$

Hence:

$$\boldsymbol{61{,}936\ MPa \leq 159\ MPa}$$

$$\boldsymbol{93{,}633\ MPa \leq 1{,}5 \times 159\ MPa}$$

The end cap protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the end cap is:

$$\boldsymbol{\frac{Analysis\ Result}{Design\ Limit} = \frac{92{,}308\ MPa}{159\ MPa} = 0{,}581}$$

Nozzle:

Detailed plot of the path D-1 to D-2 is shown in Figure 75 and the numerical values for the stresses are shown in the chart presented in Figure 76.

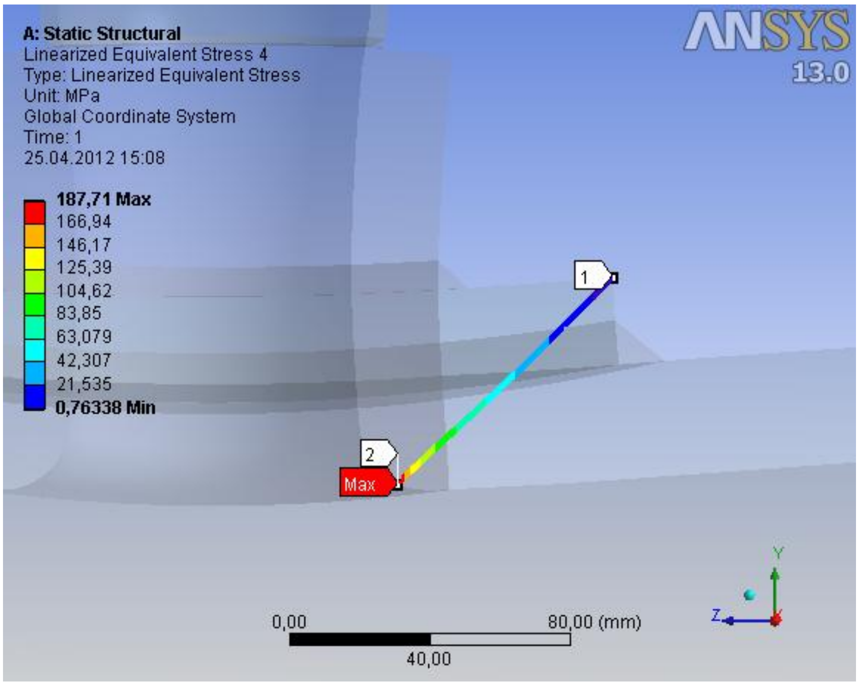


Figure 75: Path D-1 to D-2 for the nozzle.

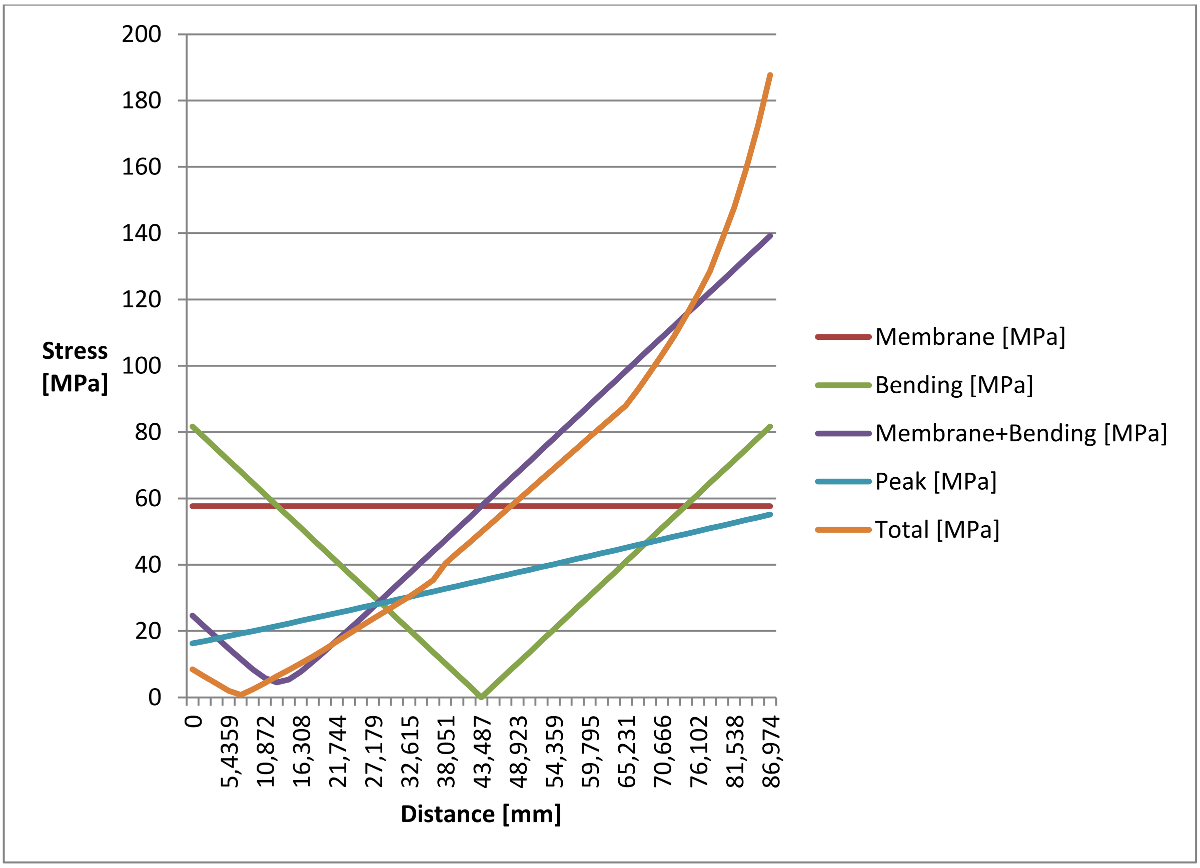


Figure 76: Stress versus distance for path D-1 to D-2.

From this data set the following results are obtained:

$$P_m = 57,628\ MPa$$

$$P_L + P_B = 139,21\ MPa$$

$$Max\ total\ stress = 187,71\ MPa$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$P_m \leq S$$

$$P_L \leq 1,5 \times S$$

$$(P_L + P_b) \leq 1,5 \times S$$

Hence:

$$57,628\ MPa \leq 159\ MPa$$

$$139,21\ MPa \leq 1,5 \times 159\ MPa$$

The nozzle protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the nozzle is:

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{187,71\ MPa}{159\ MPa} = 1,181$$

The nozzle design is considered acceptable even with a utilization factor of above (1) and the reason for this is careful engineering judgment when reading and evaluating the calculation results. By consider a point 5 mm from the end, the total stress is calculated to 147,79 MPa which is acceptable. The conclusion is that the stress concentration is only local and the vessel will not sustain any total plastic deformation.

However the nozzle should be evaluated according to the protection against local failure procedure.

5.2.1.4 Protection against local failure

The criterion to satisfy at the point of interest, (at X = 86,974 mm) for the path D-1 to D-2:

$$(\sigma_1 + \sigma_2 + \sigma_3) \leq 4 \times S$$

The principal stresses are obtained from the charts presented in Figure 77, Figure 78 and Figure 79.

$$\sigma_1 = 147,33\ MPa$$

$$\sigma_2 = 28,258\ MPa$$

$$\sigma_3 = -5,7269\ MPa$$

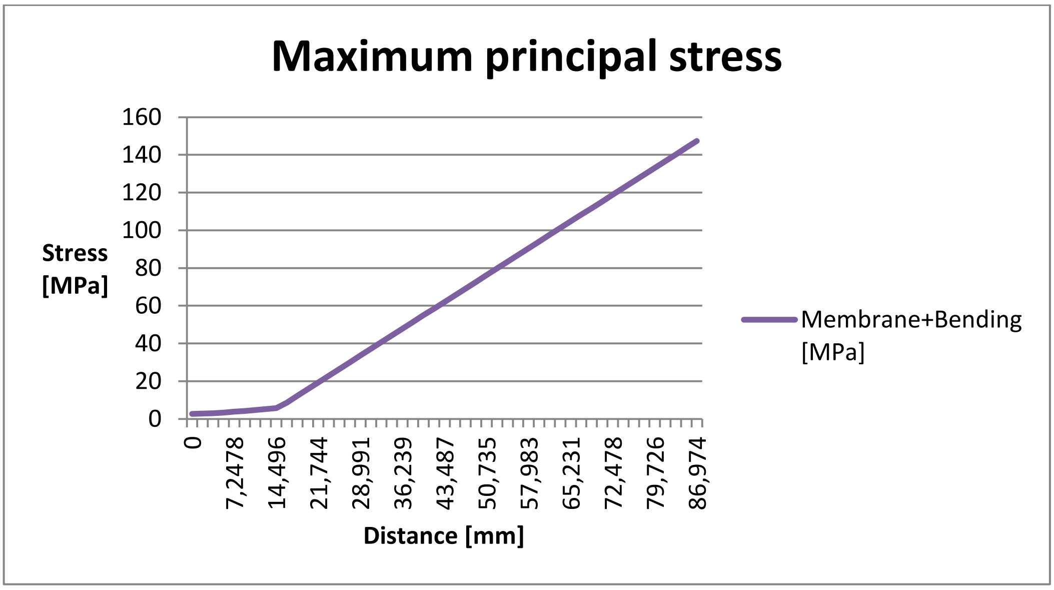


Figure 77: Maximum principal stress.

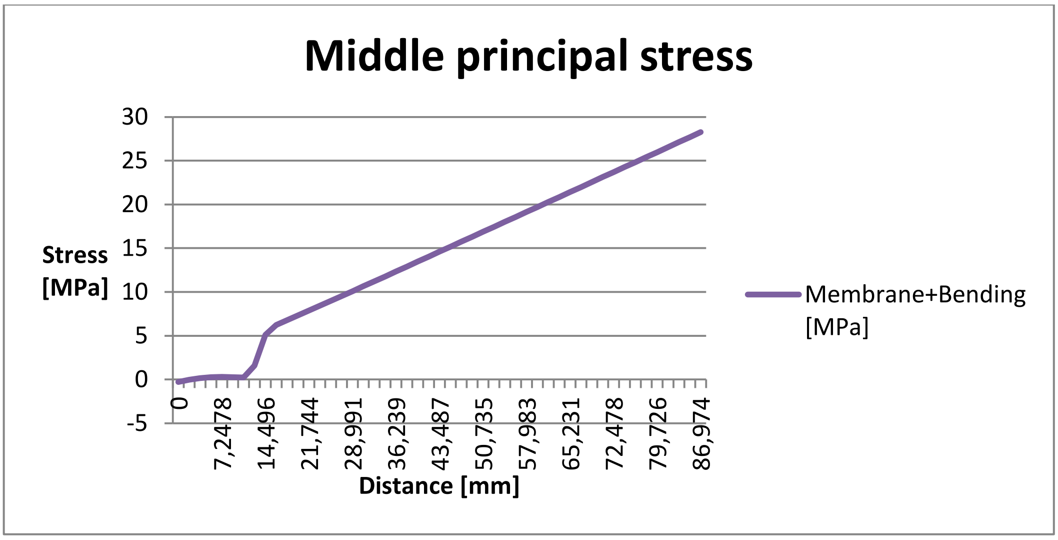


Figure 78: Middle principal stress.

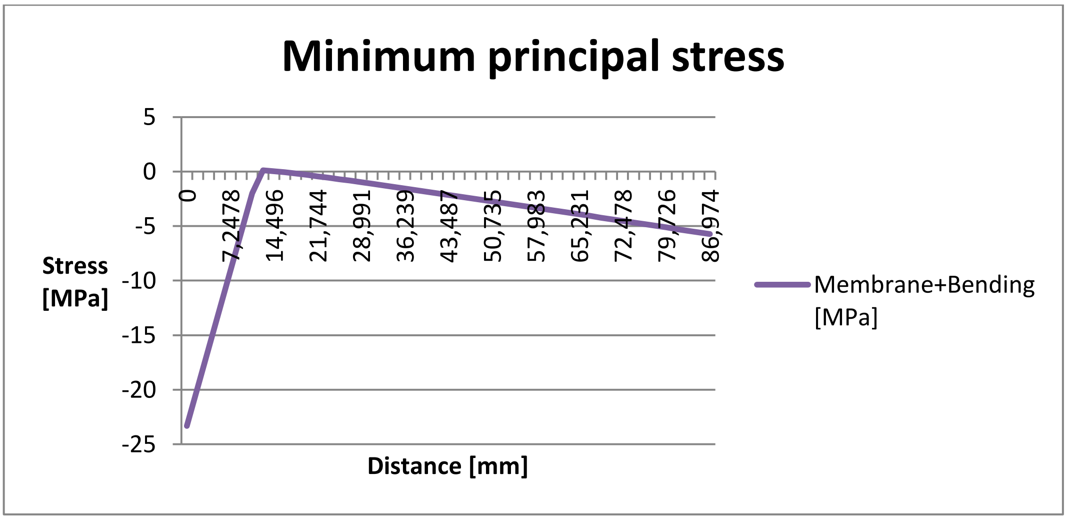


Figure 79: Minimum principal stress.

$$\mathbf{(147,33 + 28,258 - 5,7269)}\boldsymbol{MPa} \leq \mathbf{4 \times 159}\ \boldsymbol{MPa}$$

$$\mathbf{170}\ \boldsymbol{MPa} \leq \mathbf{636}\ \boldsymbol{MPa}$$

The utilization factor for the nozzle according to the local failure criteria is:

$$\frac{\boldsymbol{Analysis\ Result}}{\boldsymbol{Design\ Limit}} = \frac{\mathbf{170}\ \boldsymbol{MPa}}{\mathbf{636}\ \boldsymbol{MPa}} = \mathbf{0,267}$$

Hence the nozzle will not fail and protection against local failure is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

5.2.2 Heavy wall construction

5.2.2.1 Loading conditions

The loading condition to consider in this case is:

$$P + P_s + D + L$$

Where:

P is the operating pressure of 200 bar.

P$_s$ is the static head from liquid or bulk materials, in this case zero due to the content being gas.

D is the weight of the vessel accounted for by standard earth gravity of 9,8066 m/s^2.

L is the live loading applied to the nozzle of 30000 N.

The loading condition and boundary conditions are illustrated in Figure 80. Here the frictionless support (B) is fixed in x-direction and accounts for the symmetry conditions. The displacement (C) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (D) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (E) is half the total nozzle load due to symmetry conditions.

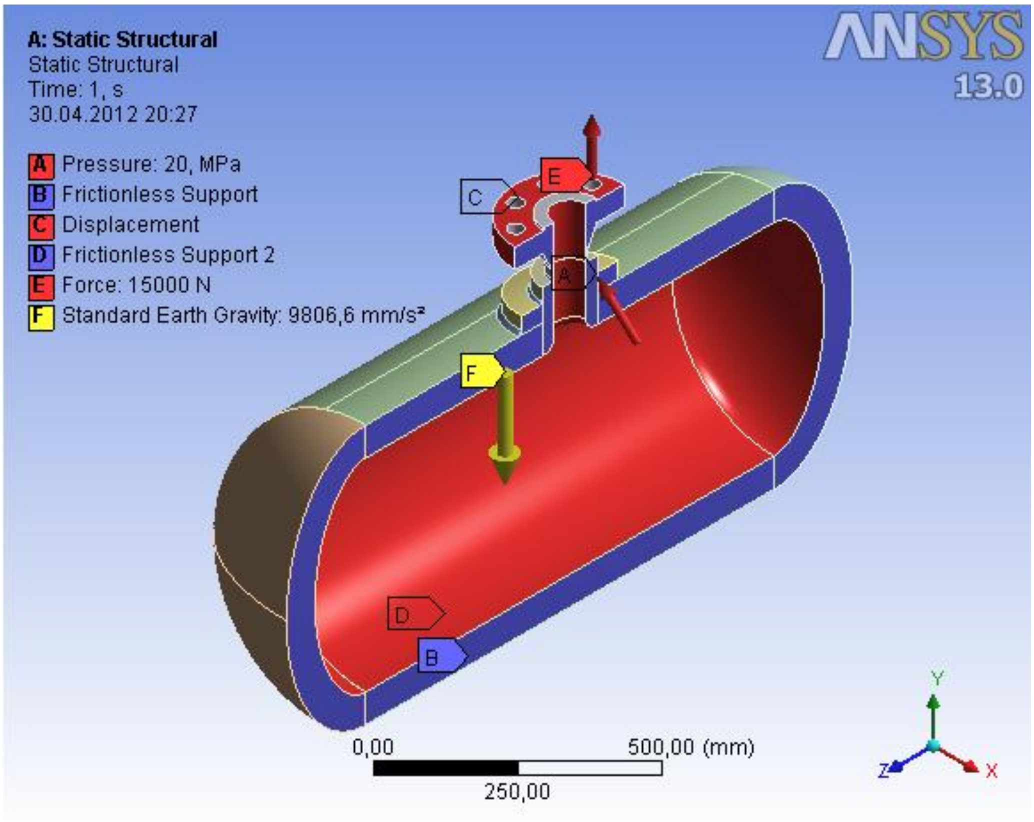


Figure 80: Loading conditions and boundary conditions.

5.2.2.2 Material properties

The maximum allowable stress at maximum anticipated operation temperature for the SA-516 Grade 70, UNS K02700 steel is obtained from table 5A, Ref /13/.

Thus for 100°C maximum allowable stress is:

$$S = 159\ MPa$$

5.2.2.3 Protection against plastic collapse

The stresses are evaluated along the paths illustrated in Figure 81 using the procedure presented in section 2.2.4 Protection against plastic collapse in this thesis. The full calculation report is available in APPENDIX G.

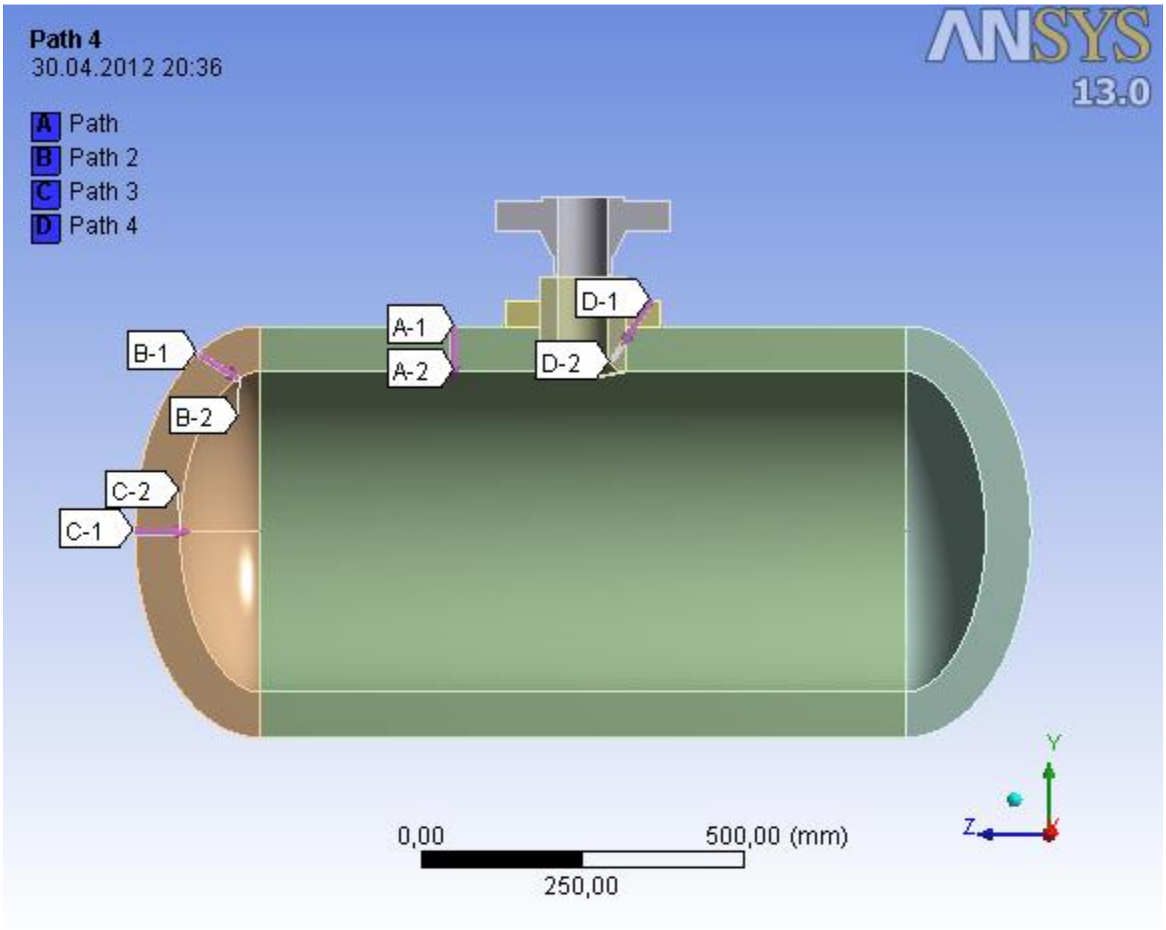


Figure 81: Stress linearization paths.

Here the following paths are used:

- A-1 to A-2 is used for the main shell.
- The most unfavorable of B-1 to B-2 or C-1 to C-2 is used for the end cap.
- D-1 to D-2 is used for the nozzle.

Main shell:

Detailed plot of the path A-1 to A-2 is shown in Figure 82 and the numerical values for the stresses are shown in the chart presented in Figure 83.

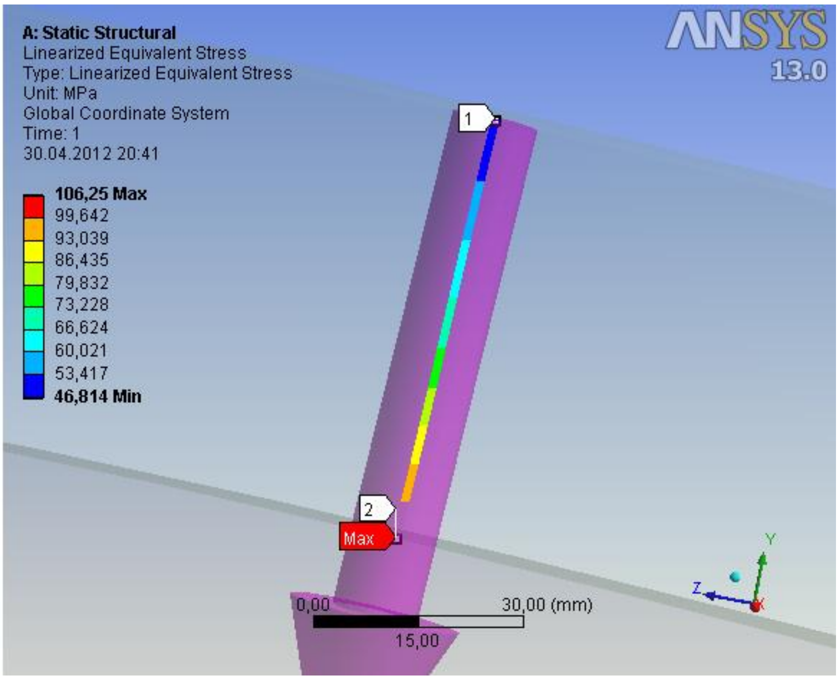


Figure 82: Path A-1 to A-2 for the main shell.

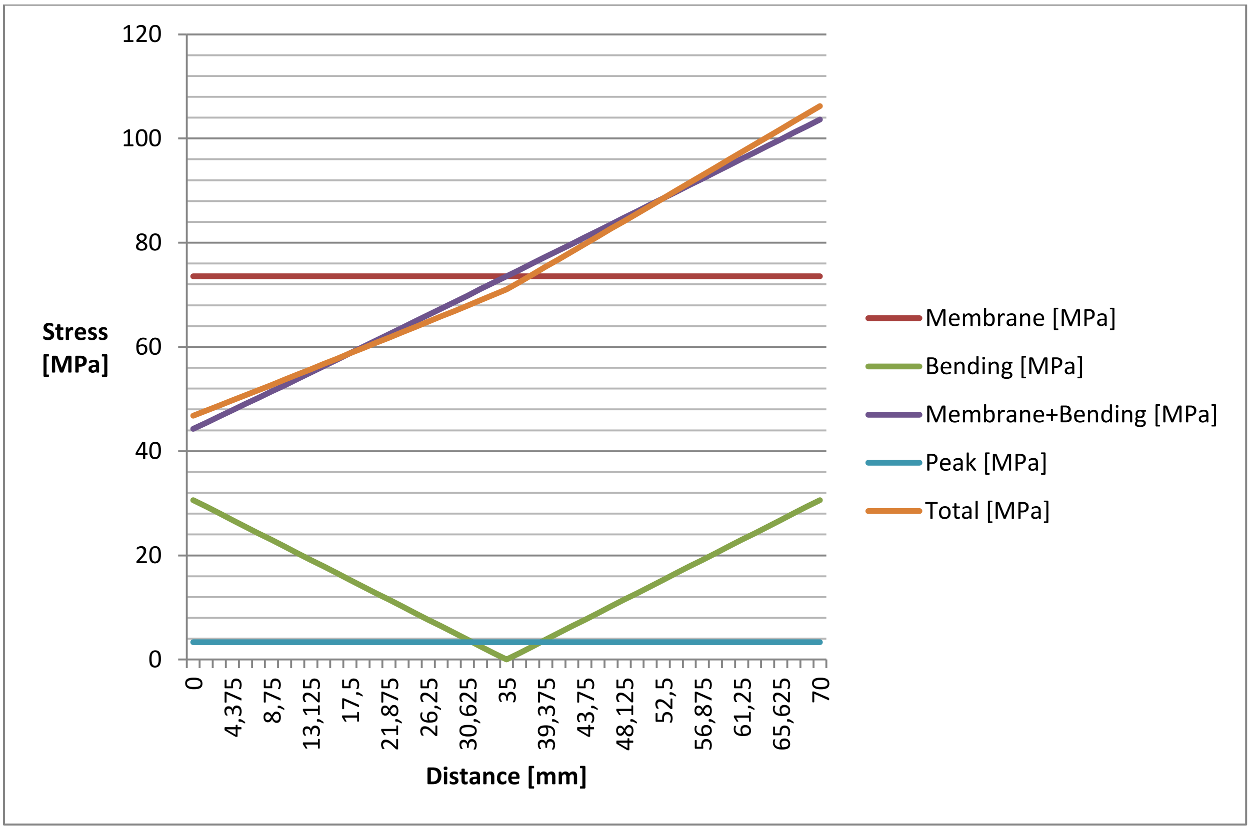


Figure 83: Stress versus distance for path A-1 to A-2.

From this data set the following results are obtained:

$$\boldsymbol{P_m = 73,563\ MPa}$$

$$\boldsymbol{P_L + P_B = 103,62\ MPa}$$

$$\boldsymbol{Max\ total\ stress = 106,25\ MPa}$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$\boldsymbol{P_m \leq S}$$

$$\boldsymbol{P_L \leq 1,5 \times S}$$

$$\boldsymbol{(P_L + P_b) \leq 1,5 \times S}$$

Hence:

$$\boldsymbol{73,563\ MPa \leq 159\ MPa}$$

$$\boldsymbol{103,62\ MPa \leq 1,5 \times 159\ MPa}$$

The main shell protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the main shell is:

$$\boldsymbol{\frac{Analysis\ Result}{Design\ Limit} = \frac{106,25\ MPa}{159\ MPa} = 0,668}$$

End cap:

Detailed plot of the path C-1 to C-2 is shown in Figure 84 and the numerical values for the stresses are shown in the chart presented in Figure 85.

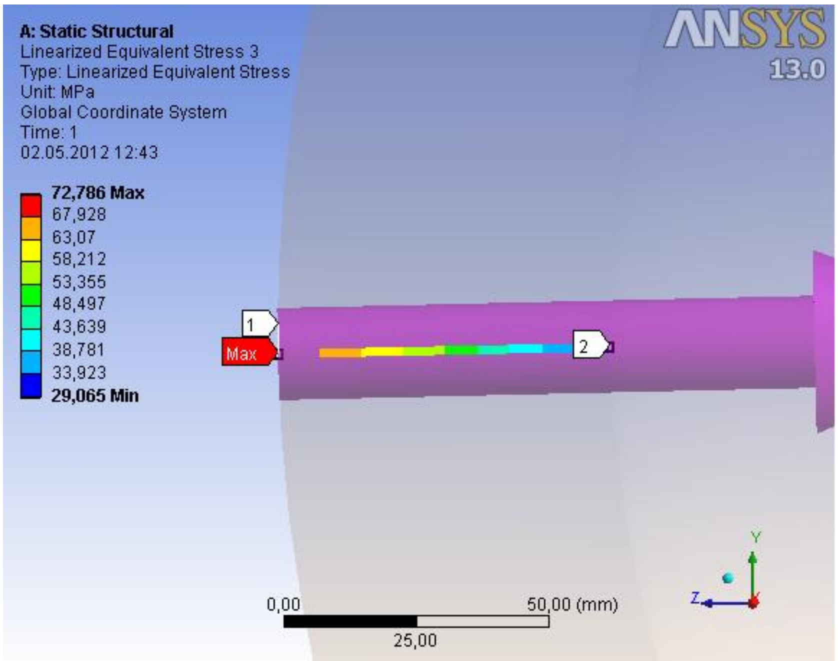


Figure 84: Path C-1 to C-2 for the end cap.

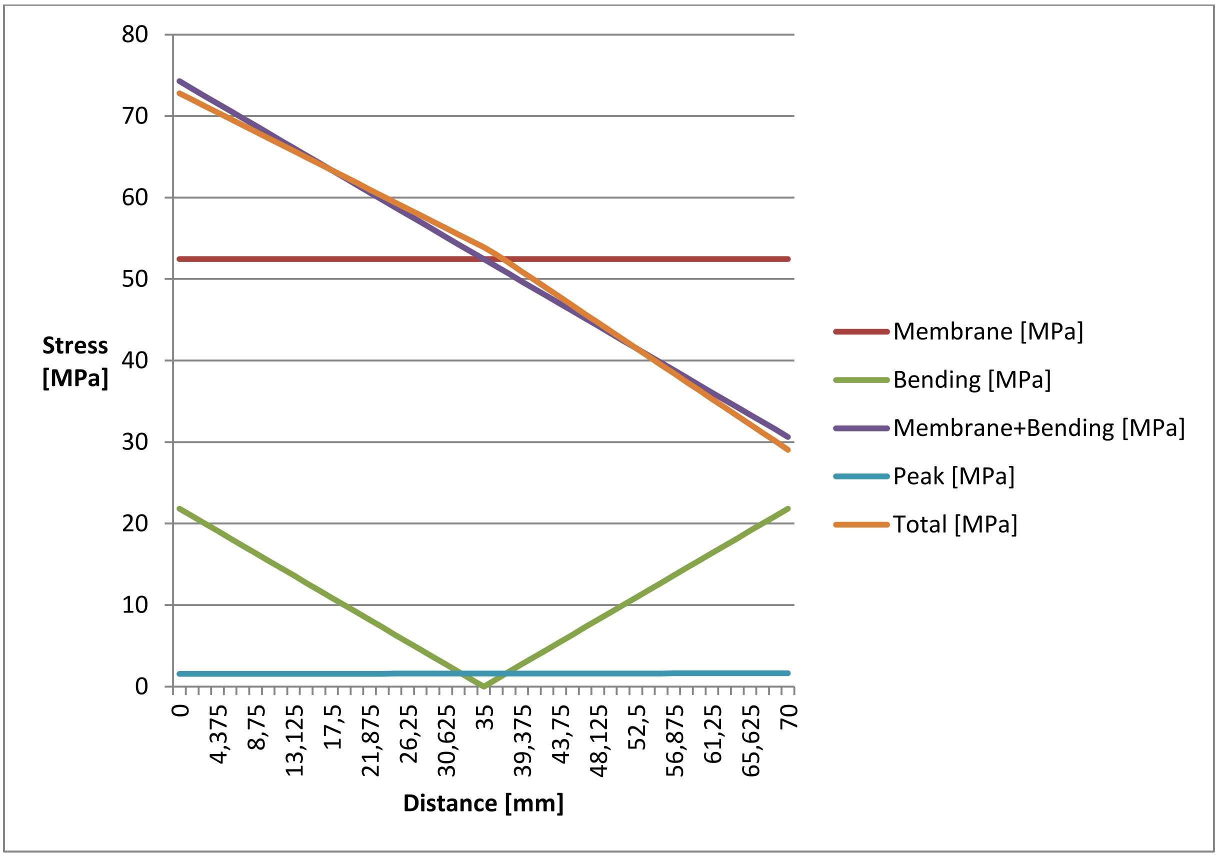


Figure 85: Stress versus distance for path C-1 to C-2.

From this data set the following results are obtained:

$$\boldsymbol{P_m = 52,451\ MPa}$$

$$\boldsymbol{P_L + P_B = 74,273\ MPa}$$

$$\boldsymbol{Max\ total\ stress = 72,786\ MPa}$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$\boldsymbol{P_m \leq S}$$

$$\boldsymbol{P_L \leq 1,5 \times S}$$

$$\boldsymbol{(P_L + P_b) \leq 1,5 \times S}$$

Hence:

$$\boldsymbol{52,451\ MPa \leq 159\ MPa}$$

$$\boldsymbol{74,273\ MPa \leq 1,5 \times 159\ MPa}$$

The end cap protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the end cap is:

$$\boldsymbol{\frac{Analysis\ Result}{Design\ Limit} = \frac{72,786\ MPa}{159\ MPa} = 0,458}$$

Nozzle:

Detailed plot of the path D-1 to D-2 is shown in Figure 86 and the numerical values for the stresses are shown in the chart presented in Figure 87.

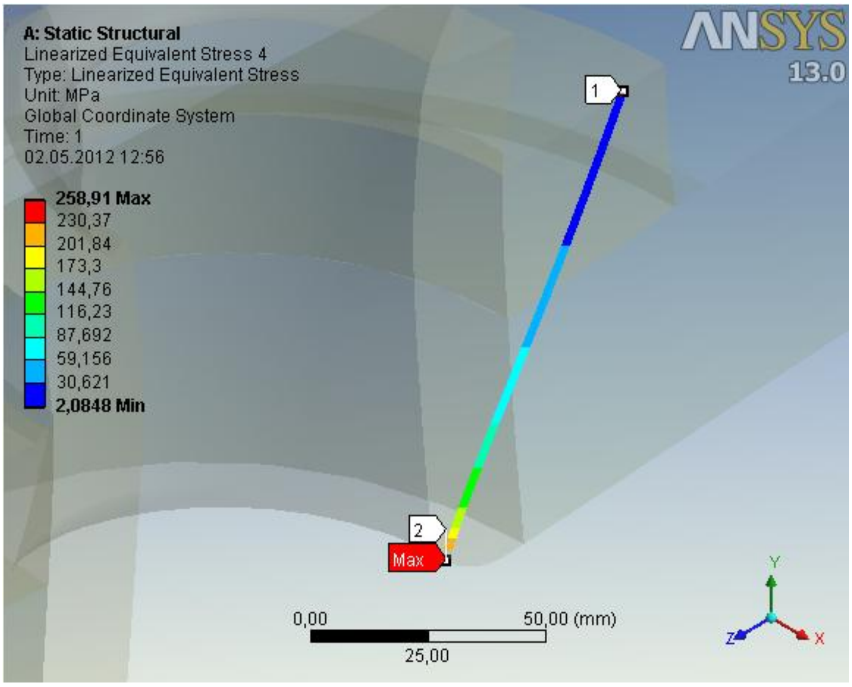


Figure 86: Path D-1 to D-2 for the nozzle.

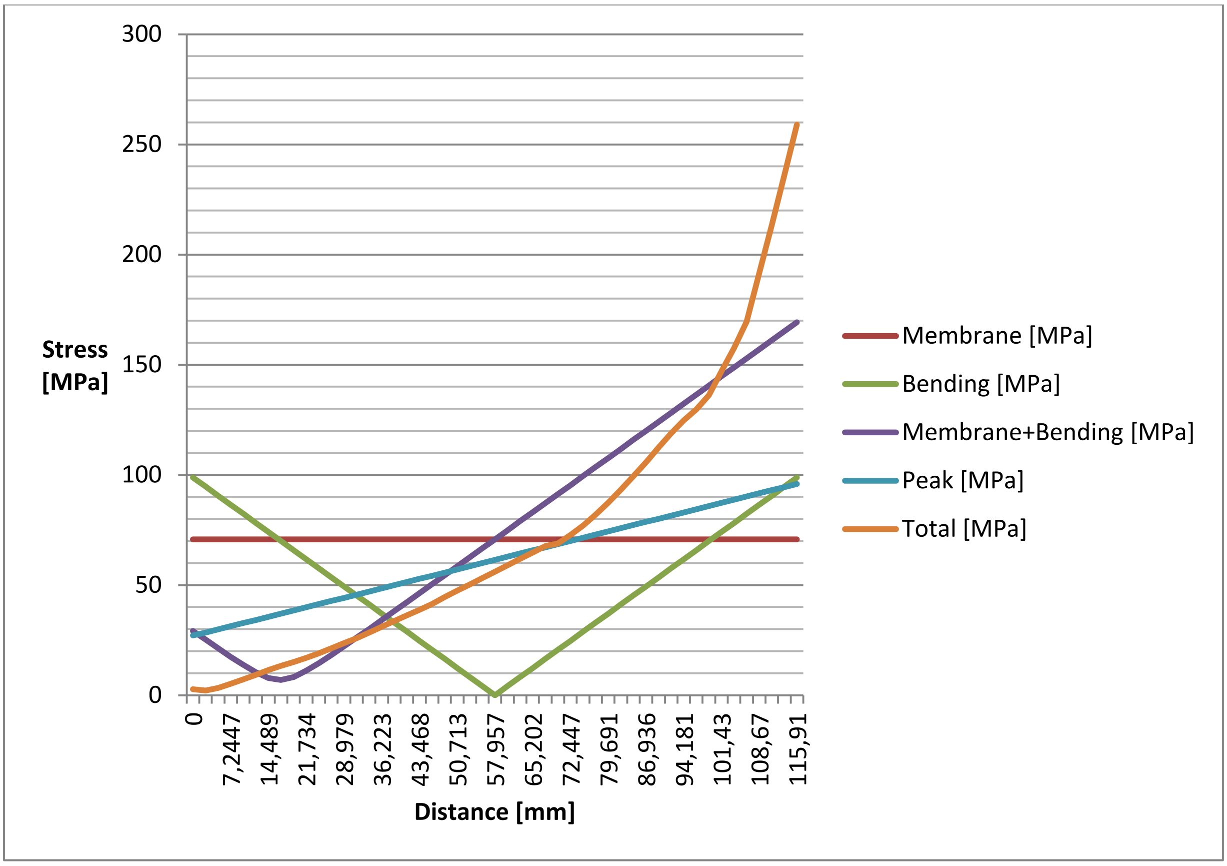


Figure 87: Stress versus distance for path D-1 to D-2.

From this data set the following results are obtained:

$$\boldsymbol{P_m = 70{,}739\ MPa}$$

$$\boldsymbol{P_L + P_B = 169{,}300\ MPa}$$

$$\boldsymbol{Max\ total\ stress = 258{,}910\ MPa}$$

To evaluate protection against plastic collapse, the computed equivalent stress shall be compared to:

$$\boldsymbol{P_m \leq S}$$

$$\boldsymbol{P_L \leq 1{,}5 \times S}$$

$$\boldsymbol{(P_L + P_b) \leq 1{,}5 \times S}$$

Hence:

$$\boldsymbol{70{,}739\ MPa \leq 159\ MPa}$$

$$\boldsymbol{169{,}300\ MPa \leq 1{,}5 \times 159\ MPa}$$

The nozzle protection against plastic collapse is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

The utilization factor for the nozzle is:

$$\boldsymbol{\frac{Analysis\ Result}{Design\ Limit} = \frac{258{,}910\ MPa}{159\ MPa} = 1{,}628}$$

The nozzle design is considered acceptable even with a utilization factor of above (1) and the reason for this is careful engineering judgment when reading and evaluating the calculation results. By consider a point 12 mm from the end, the total stress is calculated to 157,57 MPa which is acceptable. The conclusion is that the stress concentration is only local and the vessel will not sustain any total plastic deformation.

However the nozzle should be evaluated according to the protection against local failure procedure.

5.2.2.4 Protection against local failure

The criterion to satisfy at the point of interest, (at X = 115,91 mm) for the path D-1 to D-2:

$$\boldsymbol{(\sigma_1 + \sigma_2 + \sigma_3) \leq 4 \times S}$$

The principal stresses are obtained from the charts presented in Figure 88, Figure 89 and Figure 90.

$$\boldsymbol{\sigma_1 = 176{,}79\ MPa}$$

$$\boldsymbol{\sigma_2 = 17{,}867\ MPa}$$

$$\boldsymbol{\sigma_3 = -1{,}256\ MPa}$$

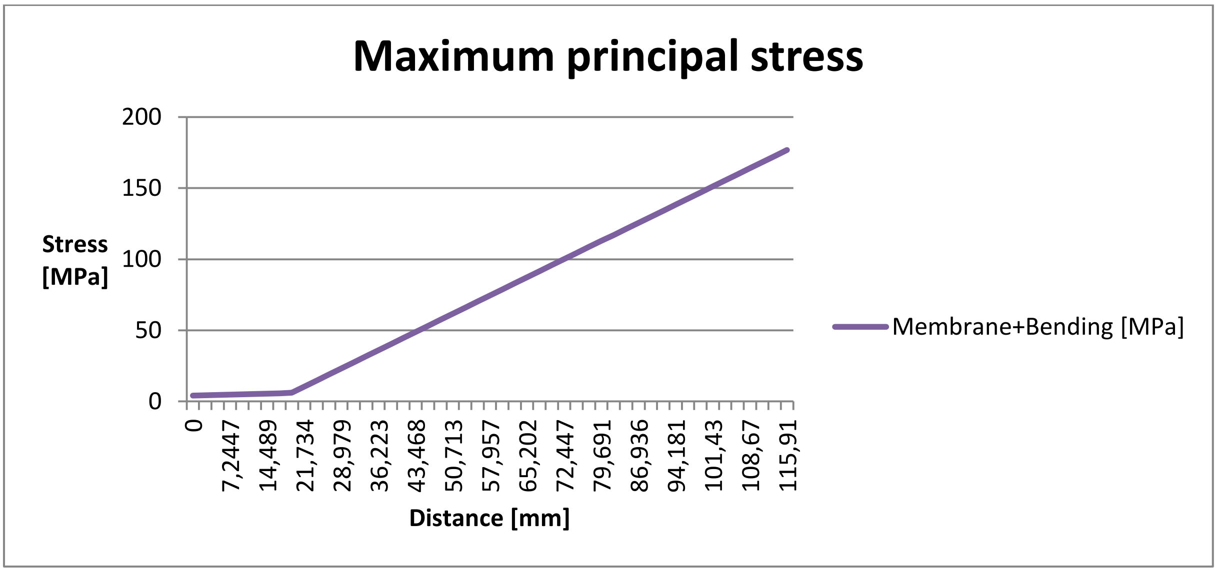


Figure 88: Maximum principal stress.

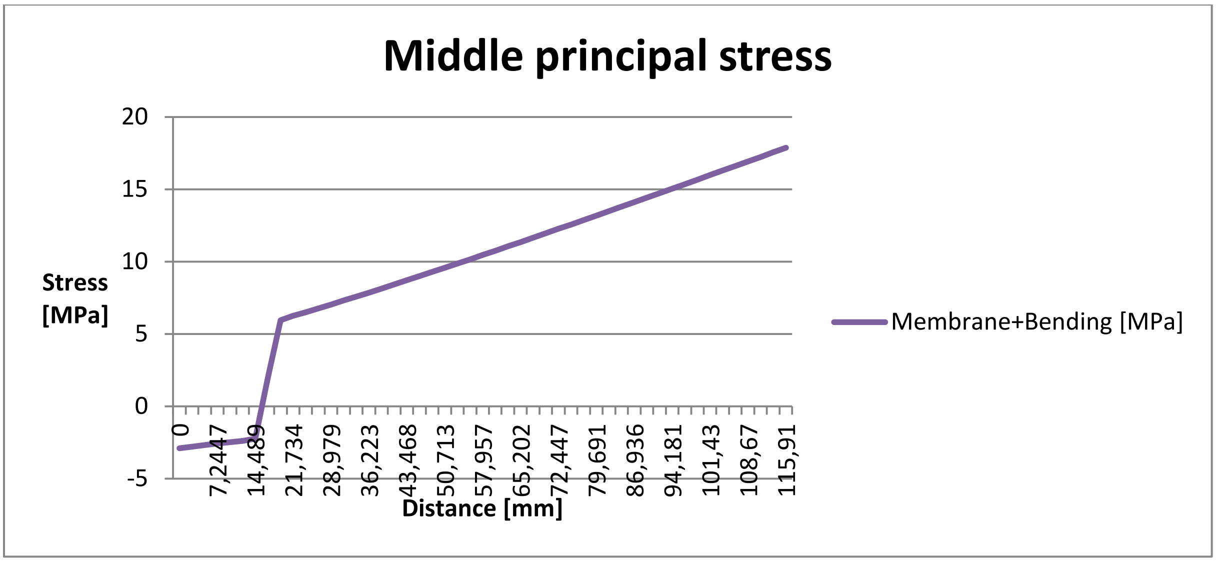


Figure 89: Middle principal stress.

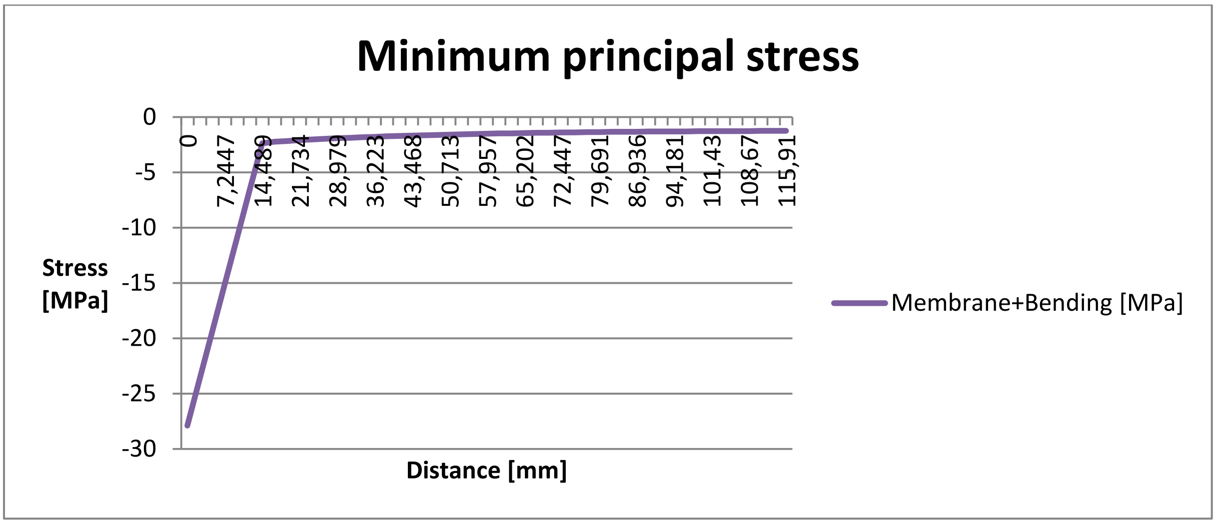


Figure 90: Minimum principal stress.

$$\mathbf{(176,79+17,867-1,256)\mathit{MPa} \leq 4 \times 159\ \mathit{MPa}}$$

$$\mathbf{193,4\ \mathit{MPa} \leq 636\ \mathit{MPa}}$$

The utilization factor for the nozzle according to the local criteria is:

$$\frac{\boldsymbol{Analysis\ Result}}{\boldsymbol{Design\ Limit}} = \frac{\mathbf{193,4\ \mathit{MPa}}}{\mathbf{636\ \mathit{MPa}}} = \mathbf{0,304}$$

Hence the nozzle will not fail and protection against local failure is considered fulfilled according to the Elastic Stress Analysis – ASME VIII div.2; 2010.

5.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010

The calculations are carried out according to section “2.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010” in this thesis.

To use this method an accurate numerical model containing the true stress-strain curve for the material must be constructed. Appendix 3.D, Ref /12/ contains the equations required to construct the material curve. The material curve constructed in Mathcad is shown in Figure 91, the Excel graph in Figure 92 and the calculation sheet for the material curve from Mathcad is presented in Figure 93.

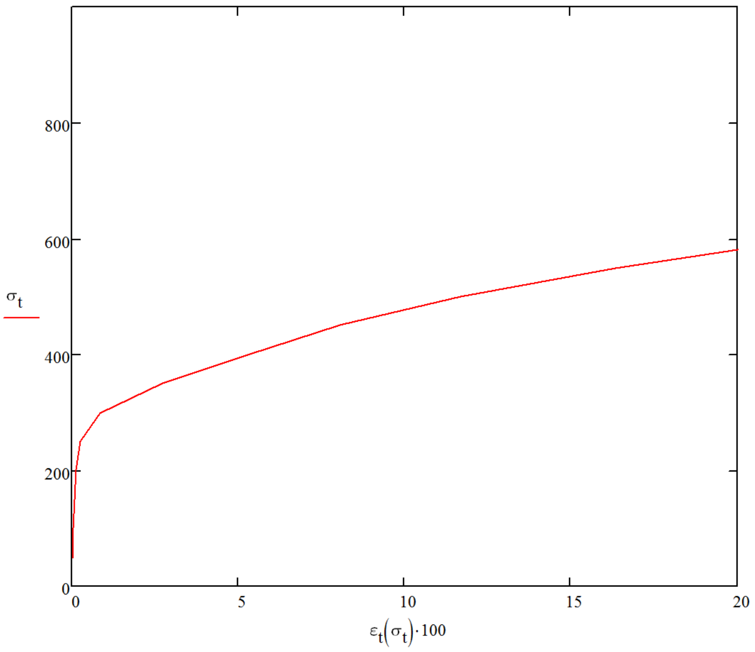


Figure 91: Material curve for SA-516 Grade 70 pressure vessel steel from Mathcad.

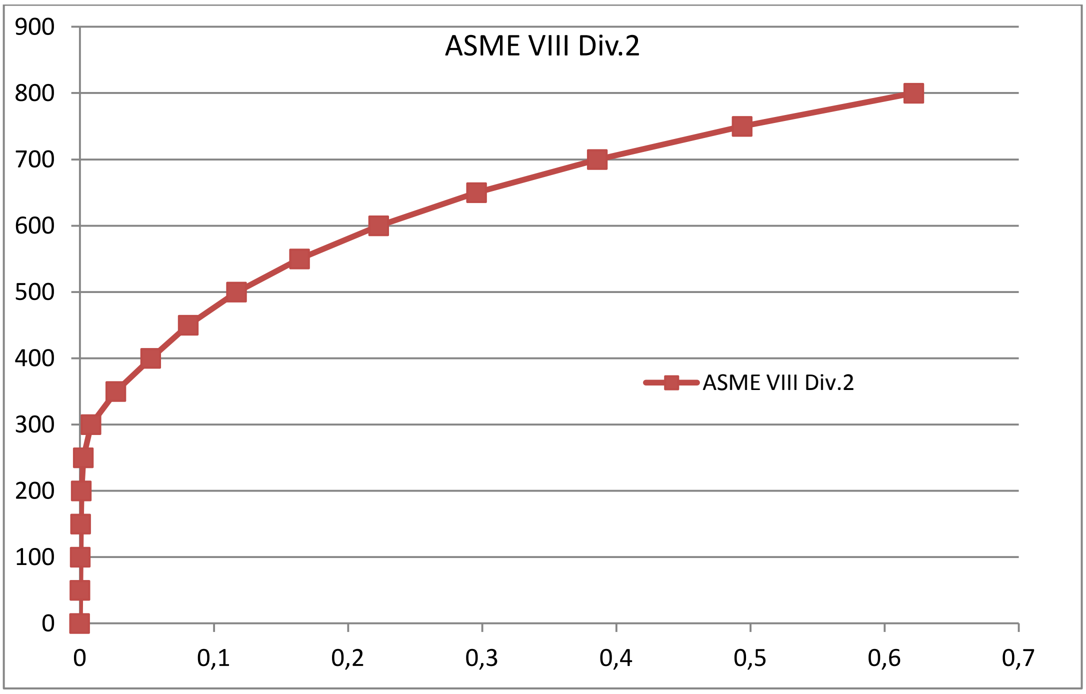


Figure 92: Material curve for SA-516 Grade 70 pressure vessel steel from Excel.

Stress strain Curve in according to AMSE VIII div. 2

Annex 3D Strength parameters

Material SA - 516 Grade 70

Material yield strength $\sigma_{ys} := 260$

Material ultimate strength $\sigma_{uts} := 485$

Ratio $R_1 := \dfrac{\sigma_{ys}}{\sigma_{uts}}$

Material parameter $K_1 := 1.5 \cdot R_1^{1.5} - 0.5 \cdot R_1^{2.5} - R_1^{3.5} = 0.371$

Engineering offset strain $\varepsilon_{ys} := 0.002$

Curve fitting exponent $m_2 := 0.60 \cdot (1 - R_1)$

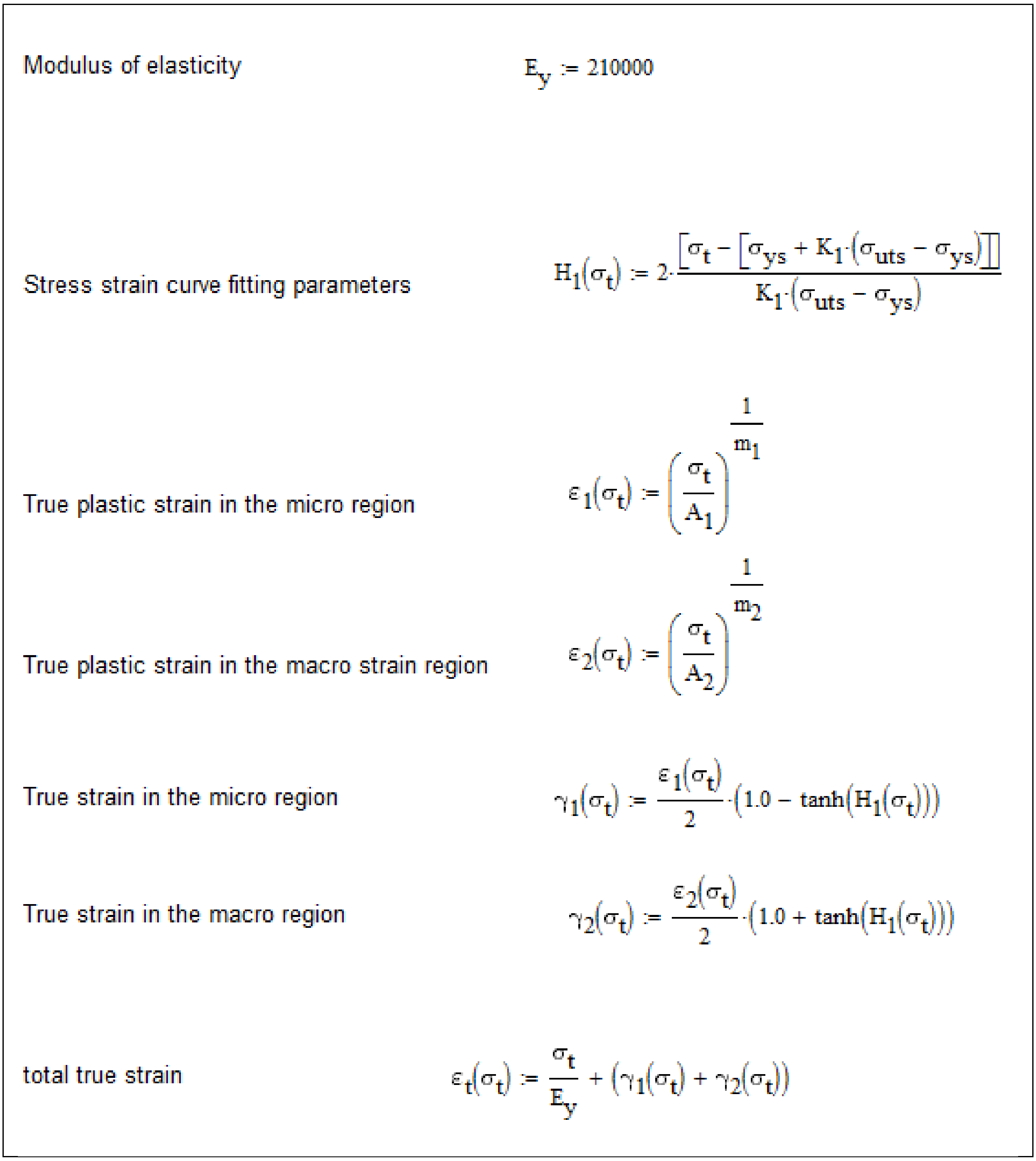
Stress strain curve fitting parameter $\varepsilon_p := 2.0 \cdot 10^{-5}$

Curve fitting constant plastic region $A_2 := \dfrac{\sigma_{uts} \cdot e^{m_2}}{m_2^{m_2}}$

Curve fitting exponent $m_1 := \dfrac{\ln(R_1) + (\varepsilon_p - \varepsilon_{ys})}{\ln\left(\dfrac{\ln(1 + \varepsilon_p)}{\ln(1 + \varepsilon_{ys})}\right)} = 0.136$

Curve fitting constant $A_1 := \dfrac{\sigma_{ys} \cdot (1 + \varepsilon_{ys})}{\left(\ln(1 + \varepsilon_{ys})\right)^{m_1}}$

True stress $\sigma_t := 50, 100 .. 600$



Modulus of elasticity $E_y := 210000$

Stress strain curve fitting parameters $H_1(\sigma_t) := 2 \cdot \frac{\left[\sigma_t - \left[\sigma_{ys} + K_1 \cdot (\sigma_{uts} - \sigma_{ys})\right]\right]}{K_1 \cdot (\sigma_{uts} - \sigma_{ys})}$

True plastic strain in the micro region $\varepsilon_1(\sigma_t) := \left(\frac{\sigma_t}{A_1}\right)^{\frac{1}{m_1}}$

True plastic strain in the macro strain region $\varepsilon_2(\sigma_t) := \left(\frac{\sigma_t}{A_2}\right)^{\frac{1}{m_2}}$

True strain in the micro region $\gamma_1(\sigma_t) := \frac{\varepsilon_1(\sigma_t)}{2} \cdot (1.0 - \tanh(H_1(\sigma_t)))$

True strain in the macro region $\gamma_2(\sigma_t) := \frac{\varepsilon_2(\sigma_t)}{2} \cdot (1.0 + \tanh(H_1(\sigma_t)))$

total true strain $\varepsilon_t(\sigma_t) := \frac{\sigma_t}{E_y} + (\gamma_1(\sigma_t) + \gamma_2(\sigma_t))$

Figure 93: Calculation sheet from Mathcad, true stress-strain curve.

The material curve constructed in Excel is used in the input file for further calculations using the Elastic-plastic method in ANSYS workbench.

5.3.1 Thin wall construction

5.3.1.1 Factored loading conditions

The factored loading condition to consider in this case is:

$$\mathbf{2,1(P + P_s + D + L + T) + 2,7L + 0,86S_s}$$

Where:

P is the operating pressure of 100 bar.

P$_s$ is the static head from liquid or bulk materials, in this case zero due to the content being gas.

D is the weight of the vessel accounted for by standard earth gravity of 9,8066 m/s^2.

L is the live loading applied to the nozzle of 30000 N.

T and **S$_S$** are not applicable and therefore set to zero.

The factored loading condition and boundary conditions are illustrated in Figure 94. Here the frictionless support (B) is fixed in x-direction and accounts for the symmetry conditions. The displacement (C) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (D) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (E) is half the total nozzle load due to symmetry conditions. The acceleration (F) accounts for the factored weight of the vessel.

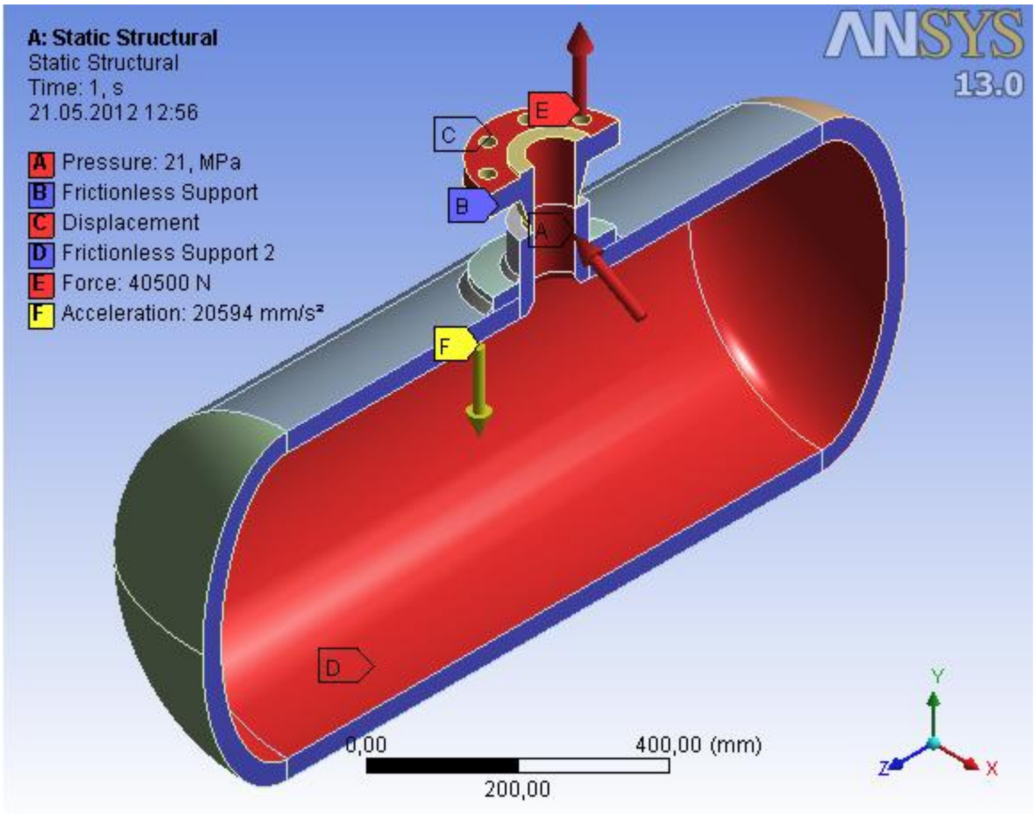


Figure 94: Factored loading condition and boundary conditions.

5.3.1.2 Protection against plastic collapse

The solution converges for the factored loading condition considered, hence the pressure vessel will not sustain plastic collapse, and protection against plastic collapse is considered fulfilled according to 2.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010. The full calculation report is available in APPENDIX H.

5.3.1.3 Protection against local failure

If the following equation is satisfied at the location being investigated the component is considered protected against local failure.

$$\boldsymbol{\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L}$$

Heat treatment is assumed performed therefore $\boldsymbol{\varepsilon_{cf} = 0}$.

The limiting tri-axial strain must be determined for each location being investigated.

The limiting tri-axial strain is determined by the following equation:

$$\boldsymbol{\varepsilon_L = \varepsilon_{Lu} \times exp\left[-\left(\frac{\alpha_{sl}}{1+m_2}\right)\left(\left\{\frac{\sigma_1+\sigma_2+\sigma_3}{3\sigma_e}\right\}-\frac{1}{3}\right)\right]}$$

The quantities **(** $\boldsymbol{\varepsilon_{Lu}}$**,** $\boldsymbol{m_2}$ **and** $\boldsymbol{\alpha_{sl}}$**)** are determined from (table 5.7) Figure 43 , Ref /12/.

$$\boldsymbol{\varepsilon_{Lu} = 0,60\left(1-\frac{260\ MPa}{485\ MPa}\right) = 0,2784}$$

$$\boldsymbol{m_2 = 0,2784}$$

$$\boldsymbol{\alpha_{sl} = 2,2}$$

End cap calculations:

The quantities needed for the end cap calculations are presented in Figure 95, Figure 96, Figure 97, Figure 98 and Figure 99.

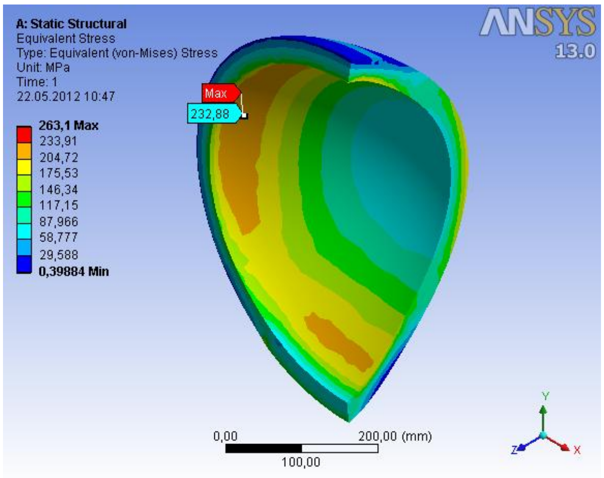


Figure 95: Equivalent stress for the end cap.

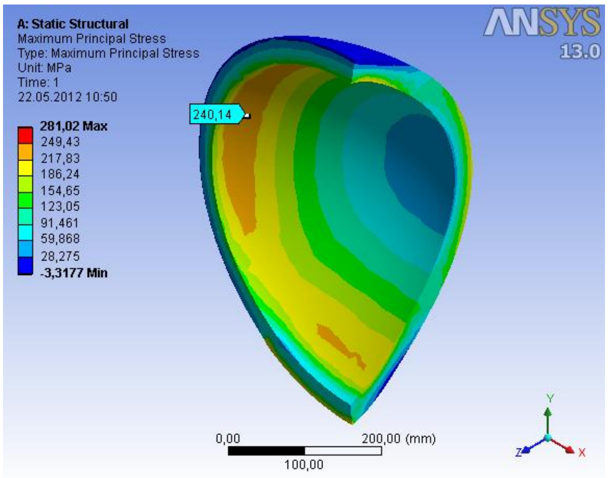


Figure 96: Maximum principal stress for the end cap.

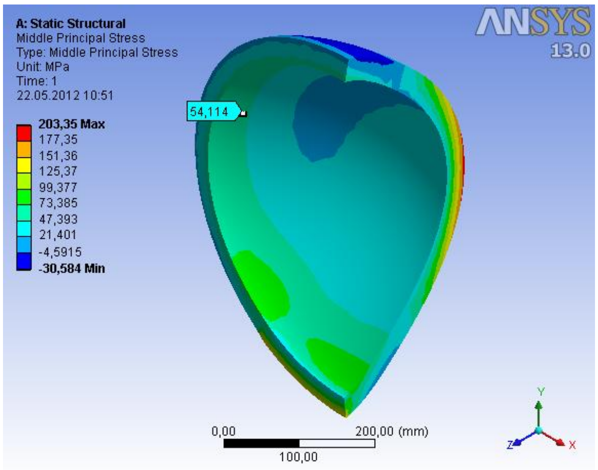


Figure 97: Middle principal stress for the end cap.

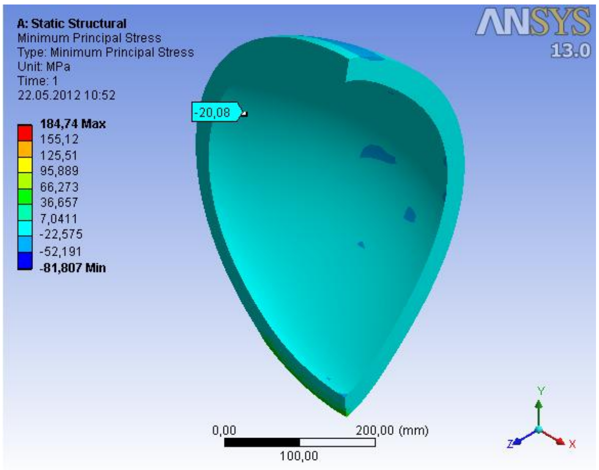


Figure 98: Minimum principal stress for the end cap.

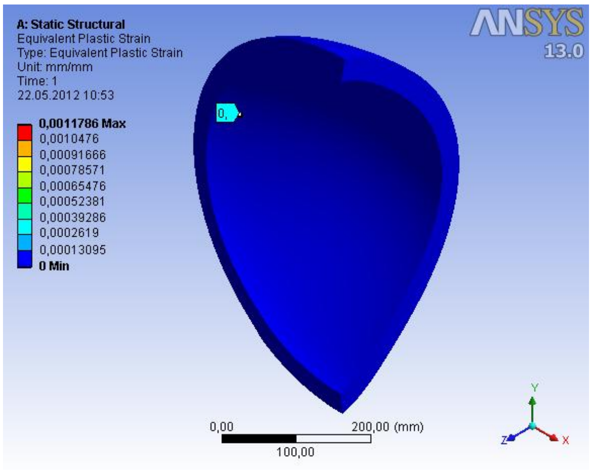


Figure 99: Equivalent plastic strain for the end cap.

Quantities needed for the end cap calculations:

$$\sigma_e = 233\ MPa$$

$$\sigma_1 = 240\ MPa$$

$$\sigma_2 = 54\ MPa$$

$$\sigma_3 = -20\ MPa$$

$$\varepsilon_{peq} = 0$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{240+54-20}{3 \times 233}\right\} - \frac{1}{3}\right)\right] = 0,25$$

There is no plastic strain present in the pressure vessel end cap, hence the following strain criteria are satisfied and the protection against local failure is fulfilled for the end cap.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

$$0 \leq 0,25$$

The utilization factor for the end cap is obtained using the limiting tri-axial strain in the material curve as shown in Figure 100 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{233\ MPa}{600\ MPa} = 0,388$$

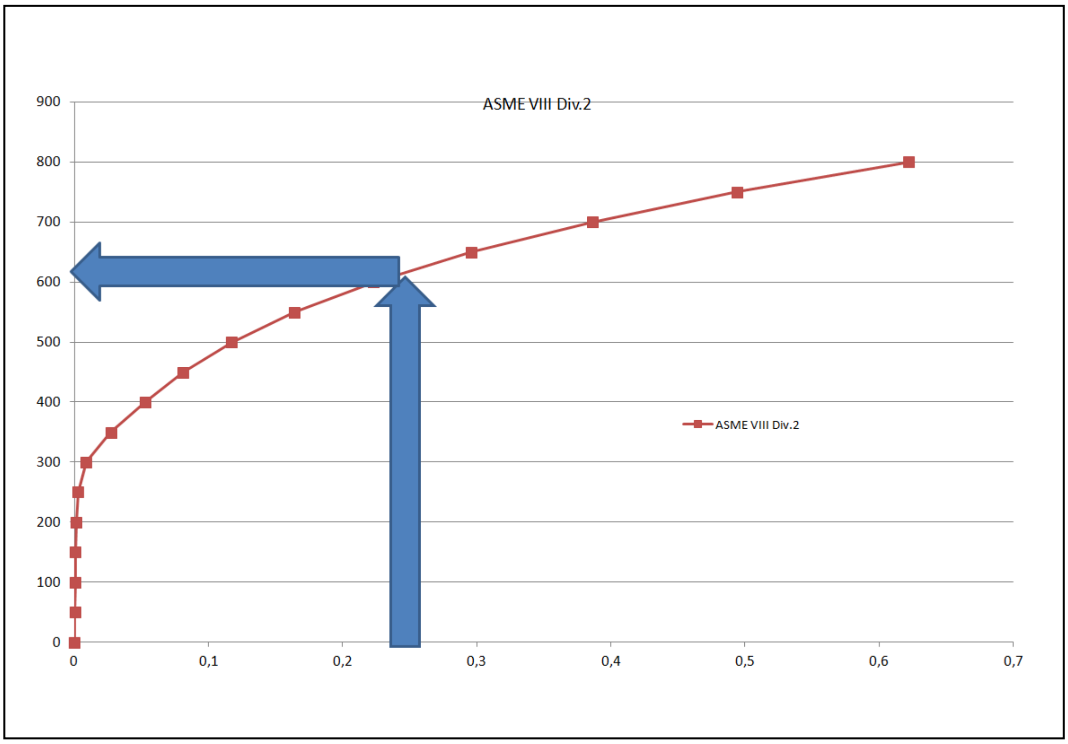


Figure 100: Design limit for the end cap.

Main shell calculations:

The quantities needed for the main shell calculations are presented in Figure 101, Figure 102, Figure 103, Figure 104 and Figure 105.

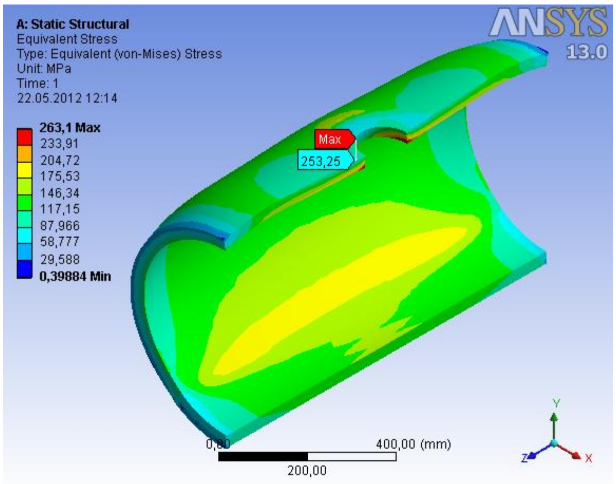


Figure 101: Equivalent stress for the main shell.

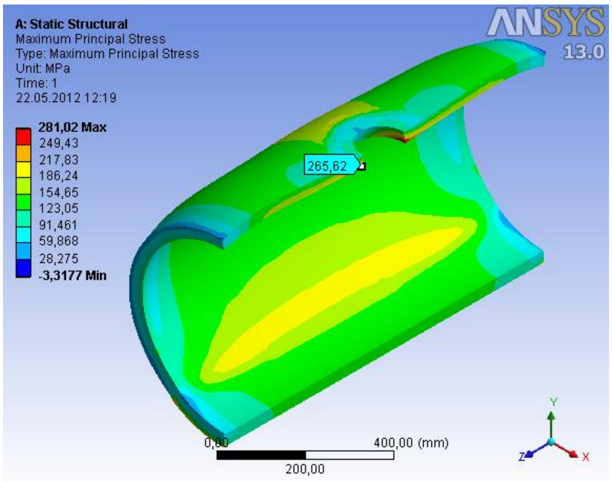


Figure 102: Maximum principal stress for the main shell.

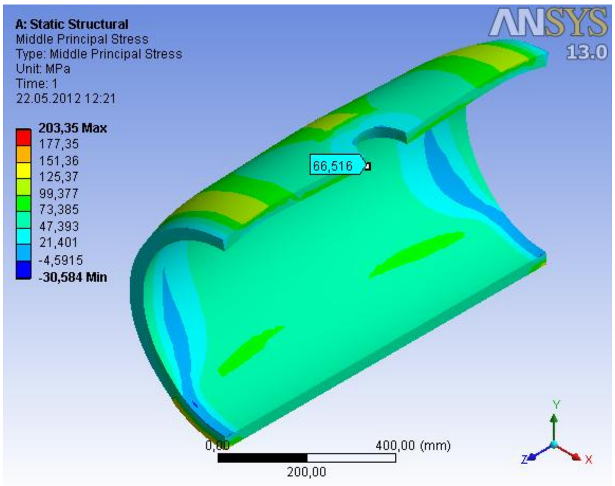


Figure 103: Middle principal stress for the main shell.

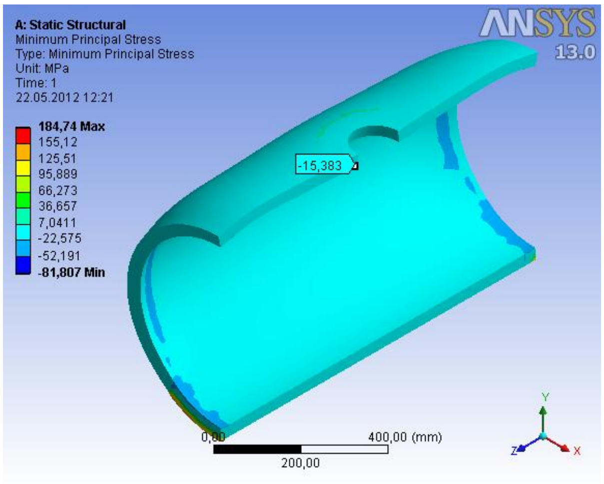


Figure 104: Minimum principal stress for the main shell.

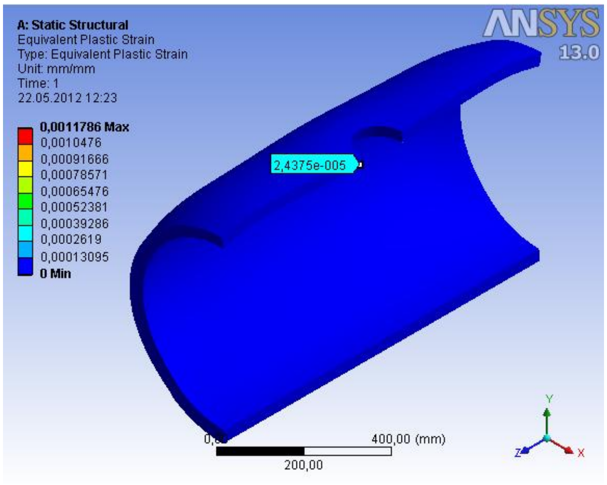


Figure 105: Equivalent plastic strain for the main shell.

Quantities needed for the main shell calculations:

$$\sigma_e = 253\ MPa$$

$$\sigma_1 = 266\ MPa$$

$$\sigma_2 = 67\ MPa$$

$$\sigma_3 = -15\ MPa$$

$$\varepsilon_{peq} = 2,4375E-5$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{266+67-15}{3\times 253}\right\}-\frac{1}{3}\right)\right] = 0,240$$

The plastic strain present in the main shell is **(2,4375E-5),** hence the following strain criteria are satisfied and the protection against local failure is considered fulfilled for the main shell.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

$$2,4375E-5 \leq 0,240$$

The utilization factor for the main shell is obtained using the limiting tri-axial strain in the material curve as shown in Figure 106 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{253\ MPa}{600\ MPa} = 0,422$$

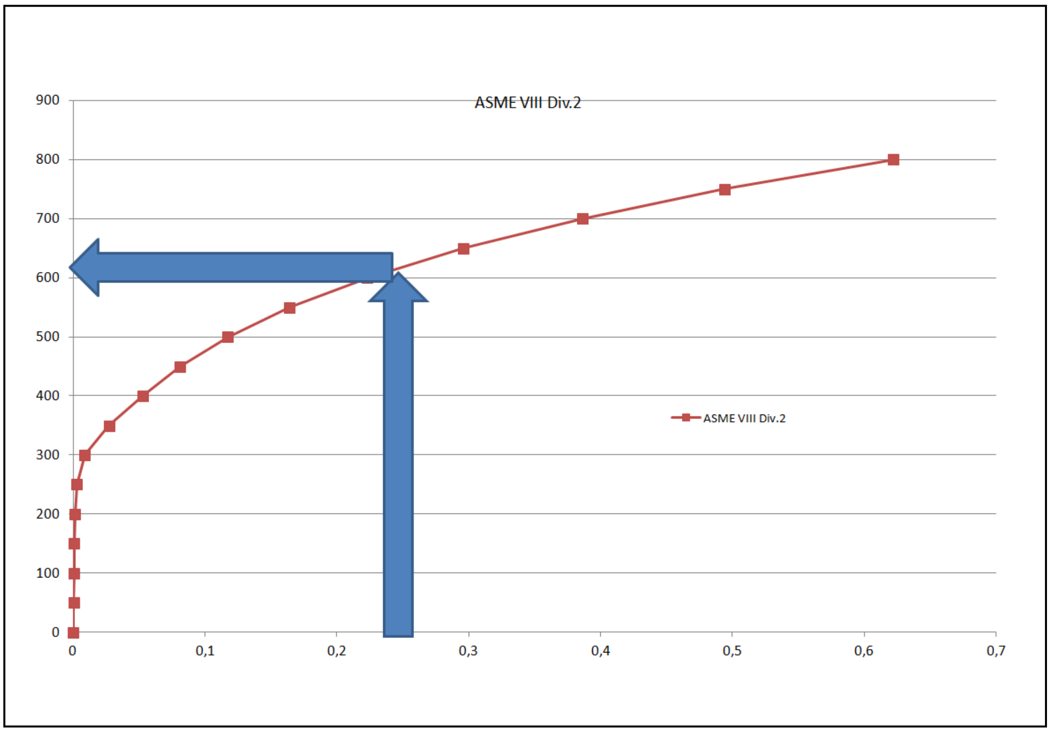


Figure 106: Design limit for the main shell.

Nozzle calculations:

The quantities needed for the nozzle calculations are presented in Figure 107, Figure 108, Figure 109, Figure 110 and Figure 111.

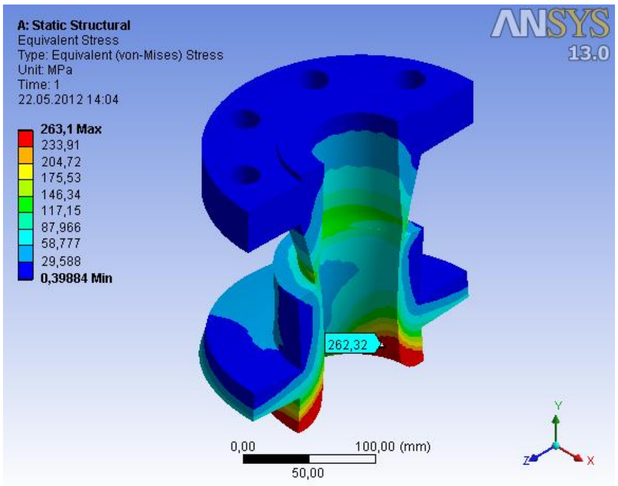


Figure 107: Equivalent stress for the nozzle.

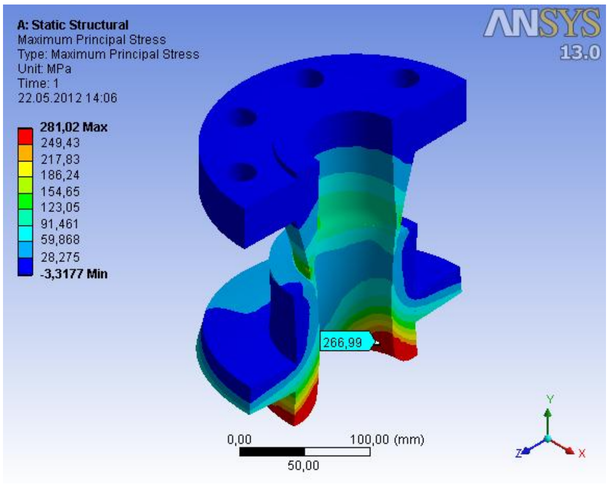


Figure 108: Maximum principal stress for the nozzle.

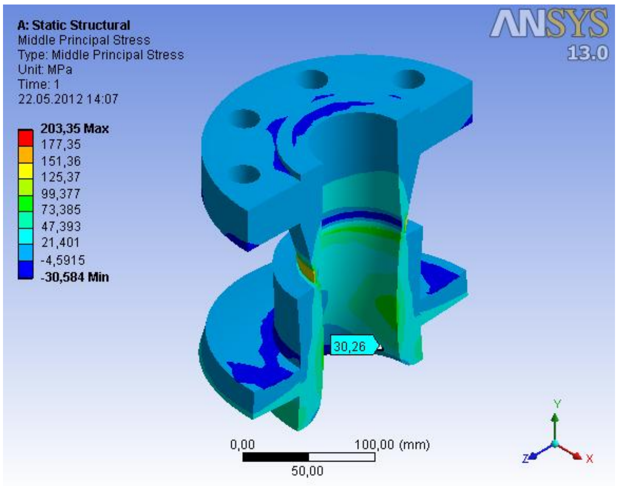


Figure 109: Middle principal stress for the nozzle.

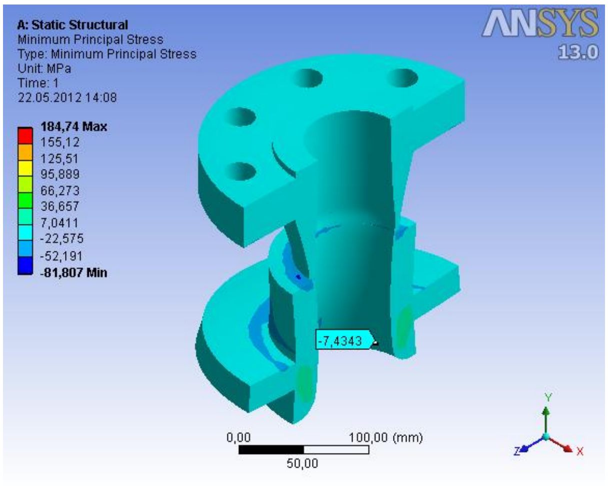


Figure 110: Minimum principal stress for the nozzle.

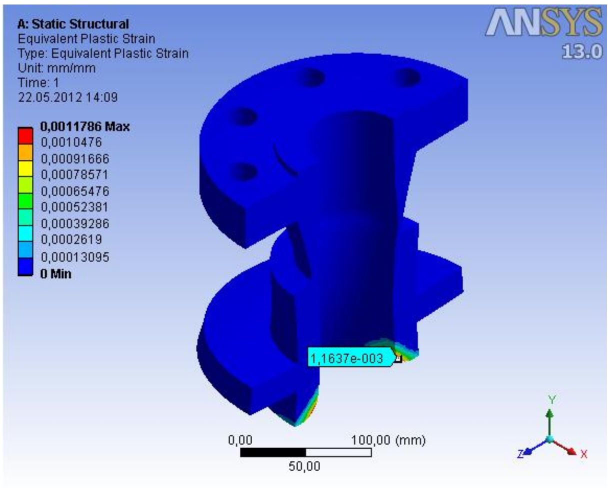


Figure 111: Equivalent plastic strain for the nozzle.

Quantities needed for the nozzle calculations:

$$\sigma_e = 262\ MPa$$

$$\sigma_1 = 267\ MPa$$

$$\sigma_2 = 30\ MPa$$

$$\sigma_3 = \ -7,5\ MPa$$

$$\varepsilon_{peq} = 1,1637E-3$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{267+30-7,5}{3\times 262}\right\}-\frac{1}{3}\right)\right] = 0,262$$

The plastic strain present in the nozzle is **(1,1637E-3),** hence the following strain criteria are satisfied and the protection against local failure is considered fulfilled for the nozzle.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

$$1,1637E-3 \leq 0,262$$

The utilization factor for the nozzle is obtained using the limiting tri-axial strain in the material curve as shown in Figure 112 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{262\ MPa}{600\ MPa} = 0,437$$

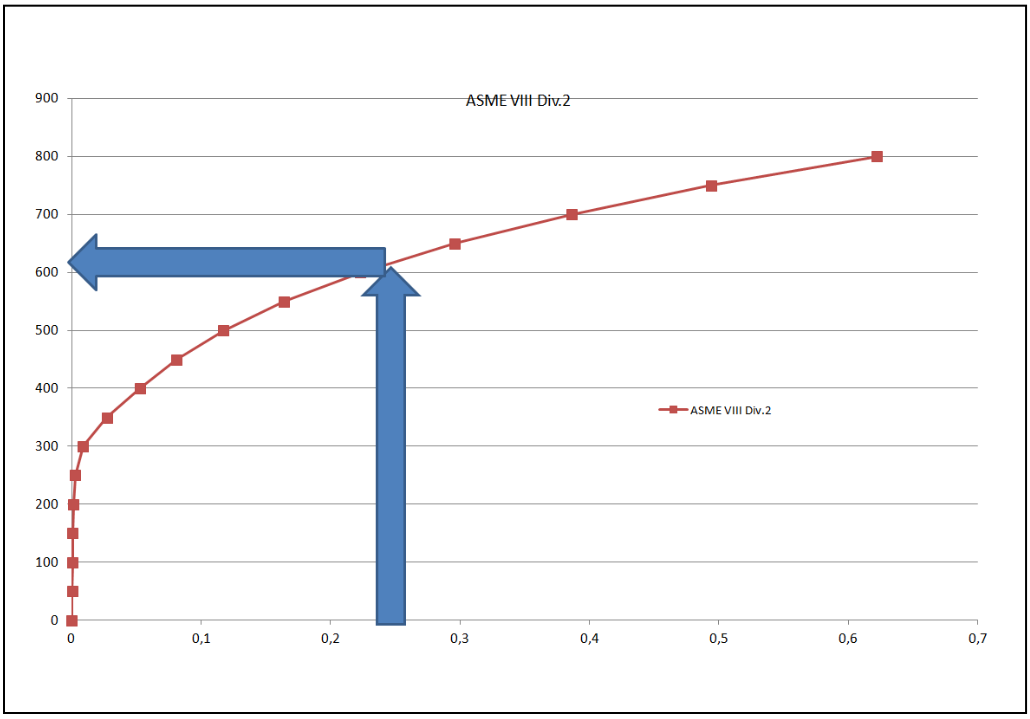


Figure 112: Design limit for the nozzle.

5.3.2 Heavy wall construction

5.3.2.1 Factored loading conditions

The factored loading condition to consider in this case is:

$$\mathbf{2,1(\boldsymbol{P}+\boldsymbol{P_s}+\boldsymbol{D}+\boldsymbol{L}+\boldsymbol{T})+2,7\boldsymbol{L}+0,86\boldsymbol{S_s}}$$

Where:

P is the operating pressure of 200 bar.

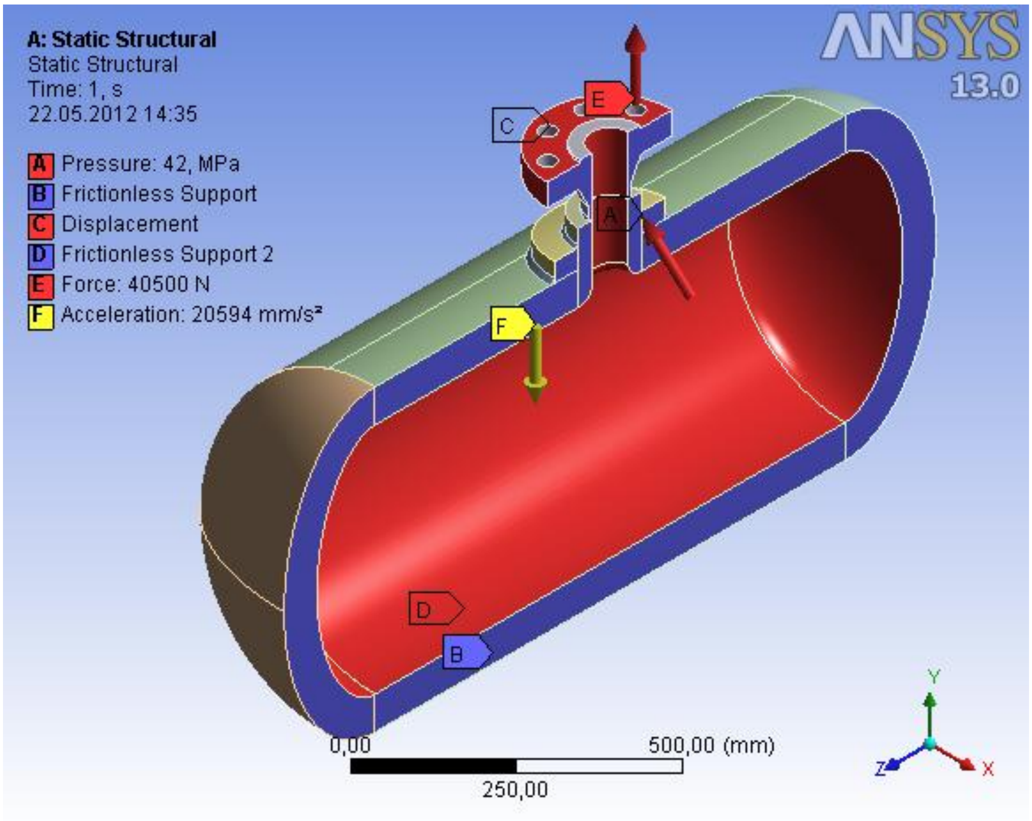
P$_s$ is the static head from liquid or bulk materials, in this case zero due to the content being gas.

D is the weight of the vessel accounted for by standard earth gravity of 9,8066 m/s^2.

L is the live loading applied to the nozzle of 30000 N.

T and **S$_S$** are not applicable and therefore set to zero.

The factored loading condition and boundary conditions are illustrated in Figure 113. Here the frictionless support (B) is fixed in x-direction and accounts for the symmetry conditions. The displacement (C) is set to zero for the x and z component to prevent the vessel from rotating. The frictionless support (D) is fixed in y-direction and accounts for the vessel standing on the floor. The nozzle load (E) is half the total nozzle load due to symmetry conditions. The acceleration (F) accounts for the factored weight of the vessel.



A: Static Structural
Static Structural
Time: 1, s
22.05.2012 14:35

A Pressure: 42, MPa
B Frictionless Support
C Displacement
D Frictionless Support 2
E Force: 40500 N
F Acceleration: 20594 mm/s²

Figure 113: Factored loading condition and boundary conditions.

5.3.2.2 Protection against plastic collapse

The solution converges for the factored loading condition considered, hence the pressure vessel will not sustain plastic collapse, and protection against plastic collapse is considered fulfilled according to 2.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010. The full calculation report is available in APPENDIX I.

5.3.2.3 Protection against local failure

If the following equation is satisfied at the location being investigated the component is considered protected against local failure.

$$\boldsymbol{\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L}$$

Heat treatment is assumed performed therefore $\boldsymbol{\varepsilon_{cf} = 0}$.

The limiting tri-axial strain must be determined for each location being investigated.

The limiting tri-axial strain is determined by the following equation:

$$\boldsymbol{\varepsilon_L = \varepsilon_{Lu} \times exp\left[-\left(\frac{\alpha_{sl}}{1+m_2}\right)\left(\left\{\frac{\sigma_1+\sigma_2+\sigma_3}{3\sigma_e}\right\}-\frac{1}{3}\right)\right]}$$

The quantities **(** $\boldsymbol{\varepsilon_{Lu}}$**,** $\boldsymbol{m_2}$ **and** $\boldsymbol{\alpha_{sl}}$**)** are determined from (table 5.7) Figure 43 , Ref /12/.

$$\boldsymbol{\varepsilon_{Lu} = 0,60\left(1-\frac{260\ MPa}{485\ MPa}\right) = 0,2784}$$

$$\boldsymbol{m_2 = 0,2784}$$

$$\boldsymbol{\alpha_{sl} = 2,2}$$

End cap calculations:

The quantities needed for the end cap calculations are presented in Figure 114, Figure 115, Figure 116, Figure 117 and Figure 118.

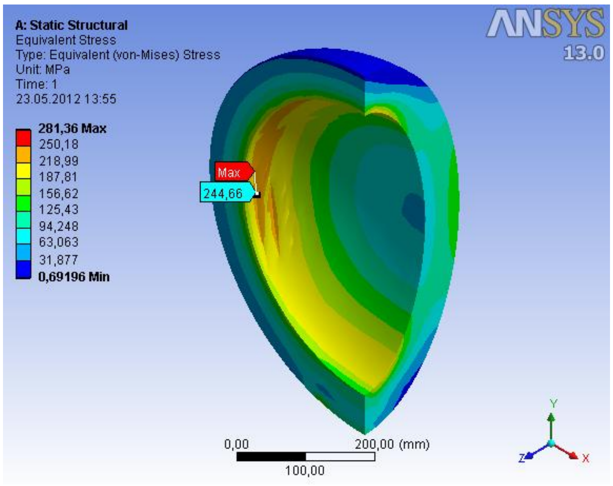


Figure 114: Equivalent stress for the end cap.

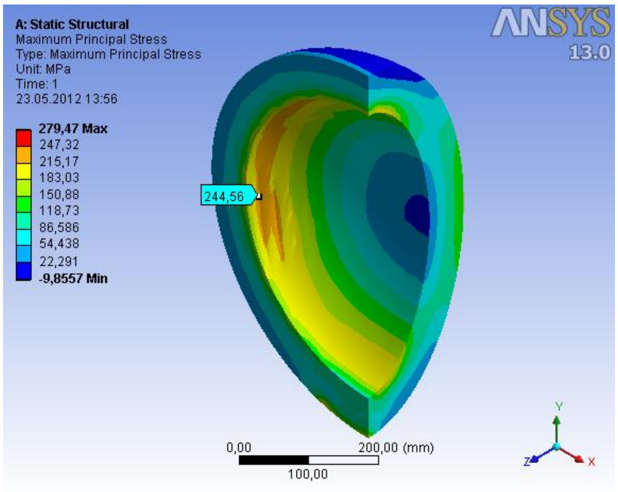


Figure 115: Maximum principal stress for the end cap.

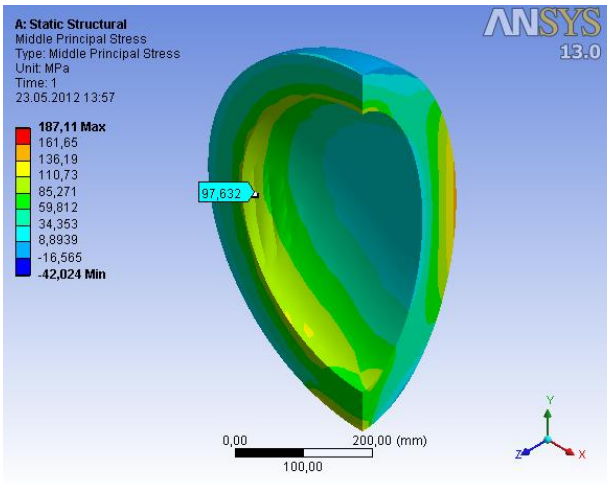


Figure 116: Middle principal stress for the end cap.

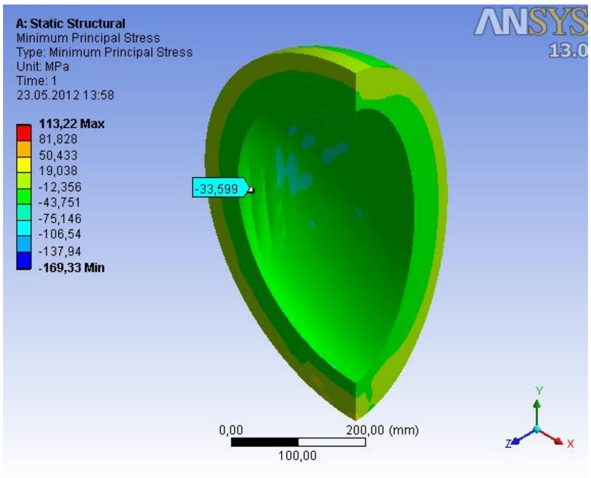


Figure 117: Minimum principal stress for the end cap.

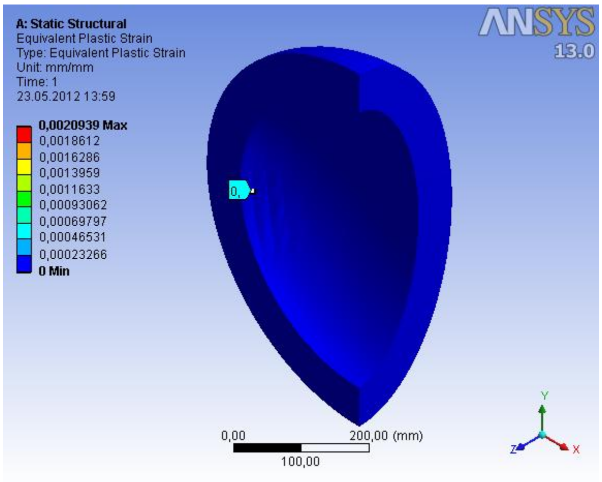


Figure 118: Equivalent plastic strain for the end cap.

Quantities needed for the end cap calculations:

$$\sigma_e = 245\ MPa$$

$$\sigma_1 = 245\ MPa$$

$$\sigma_2 = 98\ MPa$$

$$\sigma_3 = -34\ MPa$$

$$\varepsilon_{peq} = 0$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{245+98-34}{3\times 245}\right\}-\frac{1}{3}\right)\right] = 0,24$$

There is no plastic strain present in the pressure vessel end cap, hence the following strain criteria are satisfied and the protection against local failure is fulfilled for the end cap.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

$$0 \leq 0,24$$

The utilization factor for the end cap is obtained using the limiting tri-axial strain in the material curve as shown in Figure 119 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{245\ MPa}{600\ MPa} = 0,408$$

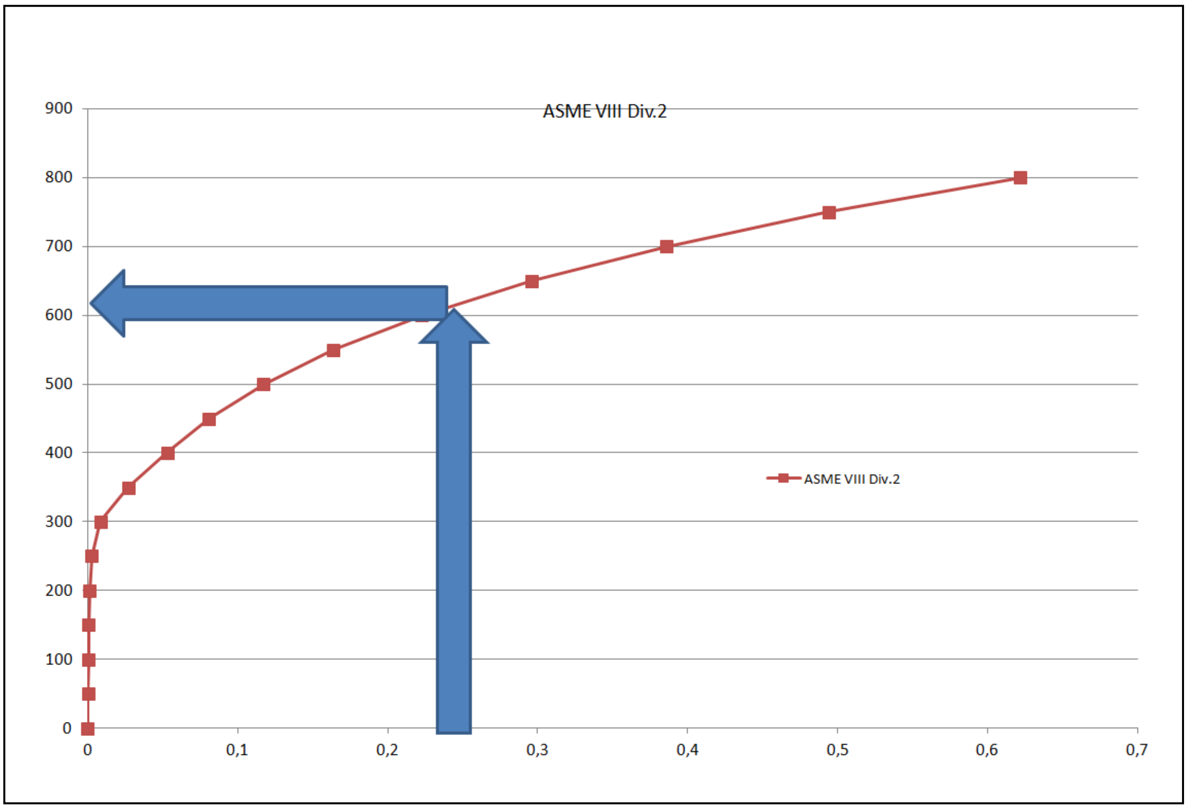


Figure 119: Design limit for the end cap.

Main shell calculations:

The quantities needed for the main shell calculations are presented in Figure 120, Figure 121, Figure 122, Figure 123 and Figure 124.

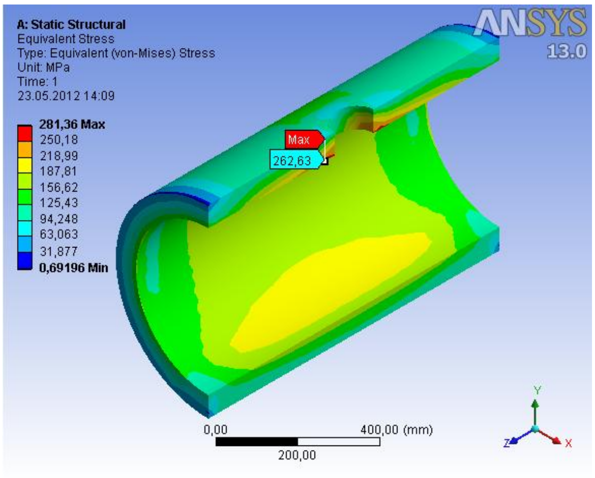


Figure 120: Equivalent stress for the main shell.

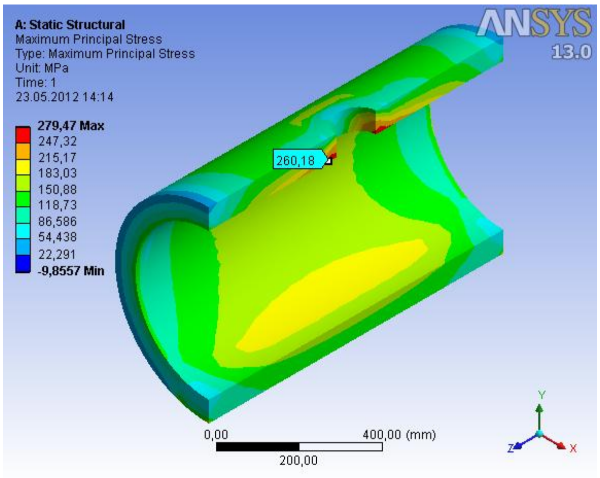


Figure 121: Maximum principal stress for the main shell.

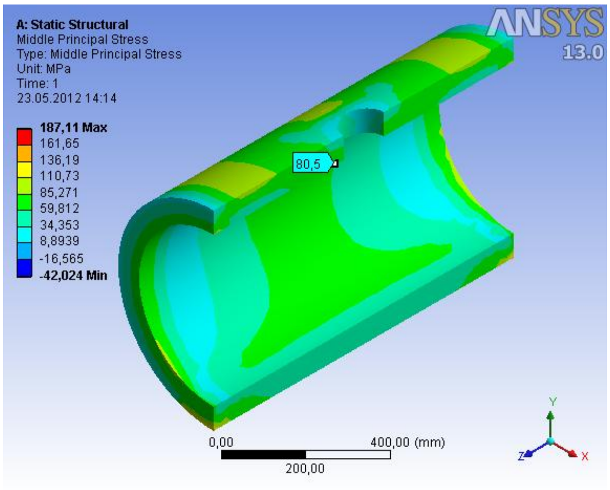


Figure 122: Middle principal stress for the main shell.

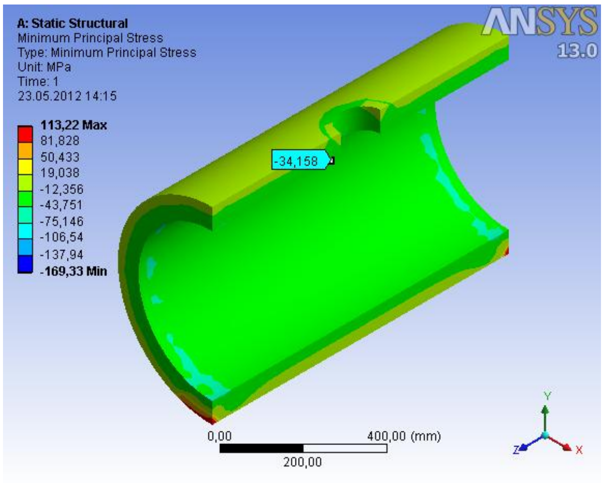


Figure 123: Minimum principal stress for the main shell.

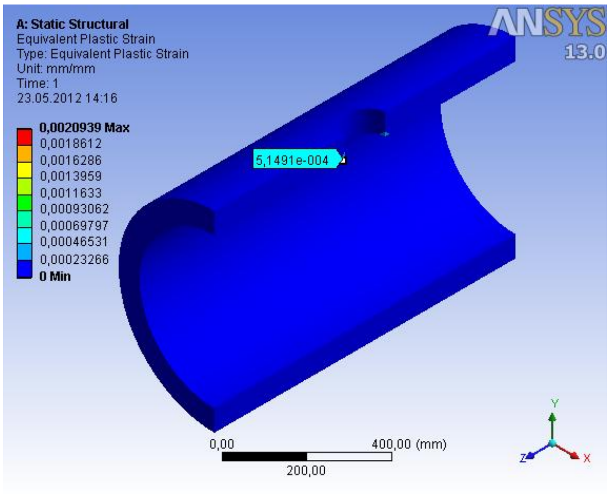


Figure 124: Equivalent plastic strain for the main shell.

Quantities needed for the main shell calculations:

$$\sigma_e = 263\ MPa$$

$$\sigma_1 = 260\ MPa$$

$$\sigma_2 = 81\ MPa$$

$$\sigma_3 = -34\ MPa$$

$$\varepsilon_{peq} = 5,15E-4$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{260+81-34}{3\times 263}\right\}-\frac{1}{3}\right)\right] = 0,253$$

The plastic strain present in the main shell is **(2,15E-4),** hence the following strain criteria are satisfied and the protection against local failure is considered fulfilled for the main shell.

$$\varepsilon_{peq} + \varepsilon_{cf} \le \varepsilon_L$$

$$5,15E-4 \le 0,253$$

The utilization factor for the main shell is obtained using the limiting tri-axial strain in the material curve as shown in Figure 125 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{263\ MPa}{600\ MPa} = 0,438$$

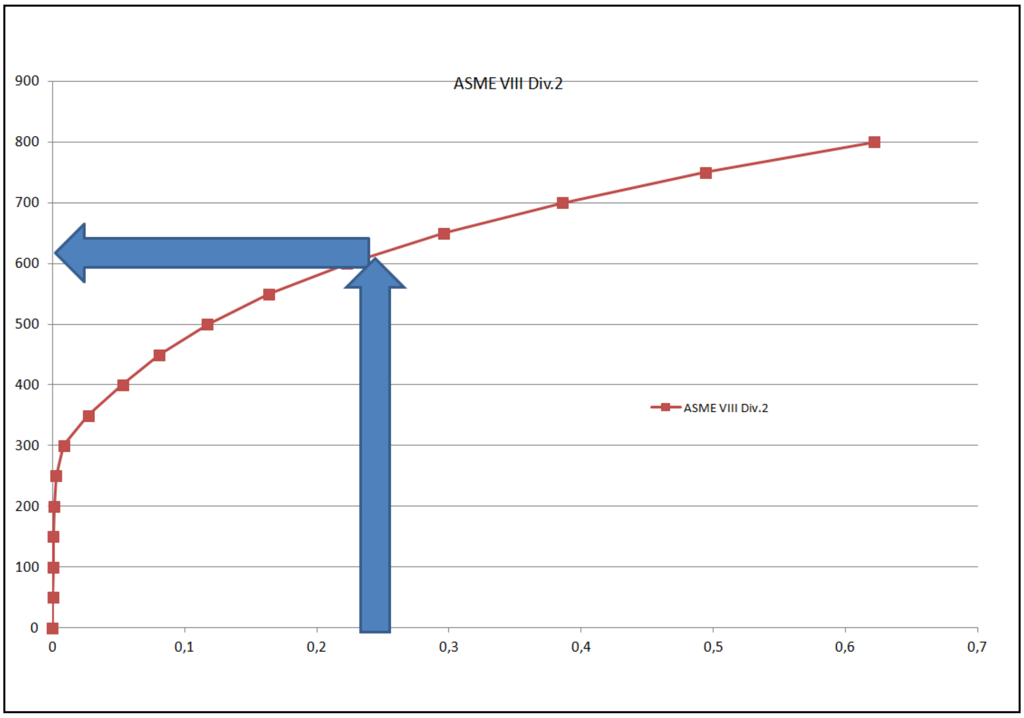


Figure 125: Design limit for the main shell.

Nozzle calculations:

The quantities needed for the nozzle calculations are presented in Figure 126, Figure 127, Figure 128, Figure 129 and Figure 130.

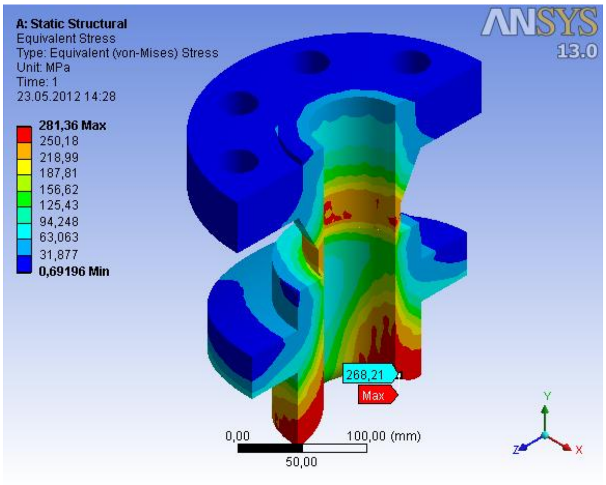


Figure 126: Equivalent stress for the nozzle.

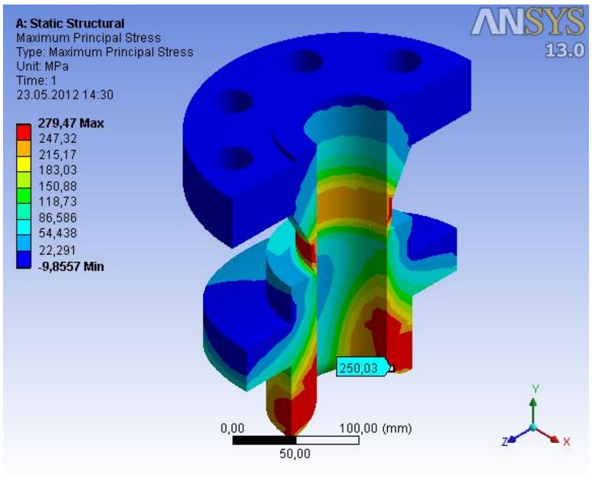


Figure 127: Maximum principal stress for the nozzle.

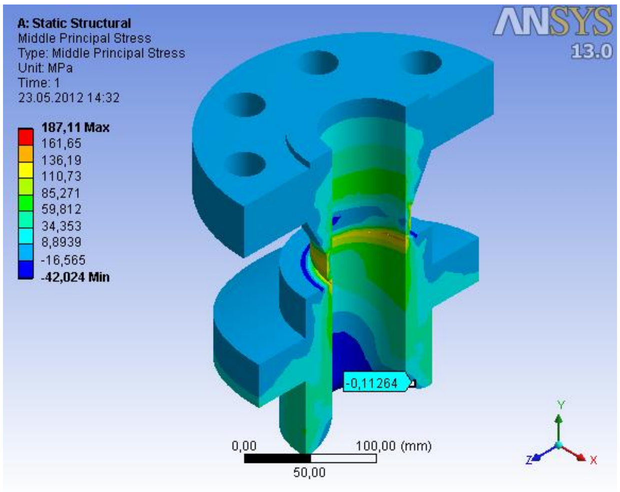


Figure 128: Middle principal stress for the nozzle.

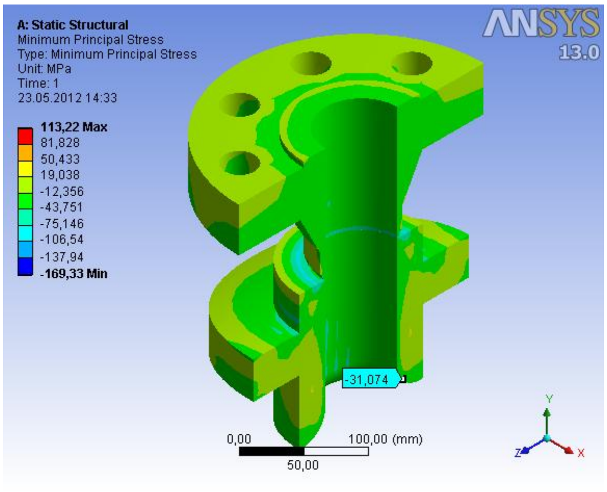


Figure 129: Minimum principal stress for the nozzle.

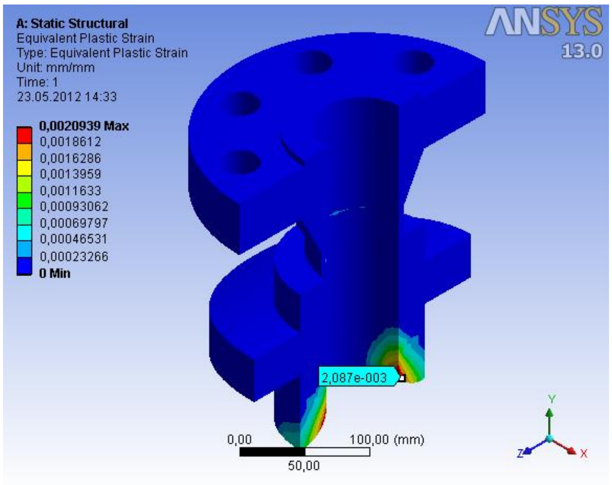


Figure 130: Equivalent plastic strain for the nozzle.

Quantities needed for the nozzle calculations:

$$\sigma_e = 268\ MPa$$

$$\sigma_1 = 250\ MPa$$

$$\sigma_2 = -0,1\ MPa$$

$$\sigma_3 = -31\ MPa$$

$$\varepsilon_{peq} = 2,087E-3$$

$$\varepsilon_L = 0,2784 \times exp\left[-\left(\frac{2,2}{1+0,2784}\right)\left(\left\{\frac{250-0,1-31}{3\times 268}\right\}-\frac{1}{3}\right)\right] = 0,309$$

The plastic strain present in the nozzle is **(2,087E-3)**, hence the following strain criteria are satisfied and the protection against local failure is considered fulfilled for the nozzle.

$$\varepsilon_{peq} + \varepsilon_{cf} \leq \varepsilon_L$$

$$2,087E-3 \leq 0,309$$

The utilization factor for the nozzle is obtained using the limiting tri-axial strain in the material curve as shown in Figure 131 and reading out the corresponding stress.

$$\frac{Analysis\ Result}{Design\ Limit} = \frac{268\ MPa}{650\ MPa} = 0,412$$

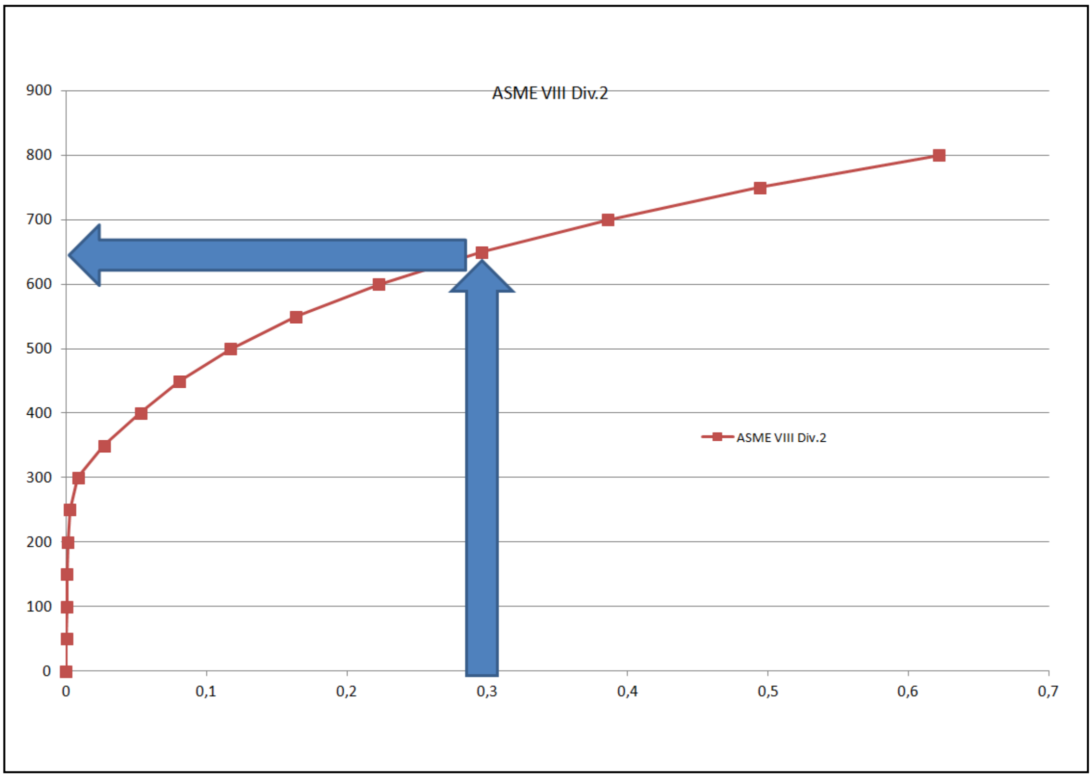


Figure 131: Analysis limit for the nozzle.

5.4 Summary

The best afford is made to obtain the calculation results as accurate as possible using the respective standard. All results have been evaluated as specified in the code and interpreted with the best possible engineering judgments.

The Direct Route, NS-EN 13445 clearly gives more conservative results than the Elastic-plastic stress analysis, ASME VIII div. 2.

The method requiring the absolute shortest set-up and calculation time is the Elastic stress analysis, ASME VIII div. 2. Using this method also makes it very easy to interpret the results as the code offers clear comparison limits.

The Elastic-plastic stress analysis, ASME VIII div. 2 is not affected by local stress concentrations, and offer the great advantage of utilizing the structure`s plastic capacity resulting in a more optimized design. However the time required for set-up and calculations is significant.

The results obtained are presented in table format in the following section of the thesis.

5.5 Comparison of results

Utilization factor of the different components against various design limits				
Thin wall pressure vessel (35 mm @ 100 bar)	Main shell	End cap	Nozzle	Average
Analysis type and method				
DBF, VVD	0,595	0,574	0,597	0,585
DBA, Direct Route, NS-EN 13445	0,850	0,697	1,240[5]	0,929
DBA, Elastic stress analysis, ASME VIII div. 2	0,593	0,581	1,181[6] (0,267)	0,785
DBA, Elastic-plastic stress analysis, ASME VIII div. 2[7]	0,422	0,388	0,437	0,416

Utilization factor of the different components against various design limits				
Heavy wall pressure vessel (70 mm @ 200 bar)	Main shell	End cap	Nozzle	Average
Analysis type and method				
DBF, VVD	0,595	0,551	0,481	0,544
DBA, Direct Route, NS-EN 13445	1,010	0,779	1,570[5]	1,120[8]
DBA, Elastic stress analysis, ASME VIII div. 2	0,668	0,458	1,628[6] (0,304)	0,918
DBA, Elastic-plastic stress analysis, ASME VIII div. 2[7]	0,438	0,408	0,412	0,419

Notes:

---

[5] The complete pressure vessel is considered acceptable even though the nozzle itself fails the test.

[6] The nozzle is considered acceptable according to the local failure criteria. The utilization factor for the nozzle according to the local failure criteria is presented in parentheses.

[7] The pressure vessel design is acceptable according to the protection against plastic collapse due to a converging solution. The utilization factors are obtained from the protection against local failure criterion.

[8] The heavy wall pressure vessel utilization factor indicates that the design should be re-evaluated according to this method.

CHAPTER 6 COMMENTS

6.1 Pressure vessel failure

One should always remember that the way pressure vessels will fail has never changed even if the methods for calculations, codes or standards sometimes do change. The most common pressure vessel failure modes are illustrated in Figure 132[9]. When evaluating calculation or analysis results good engineering judgment should be used to conclude if the results make any sense or that they are reasonable compared to real anticipate physical behavior.

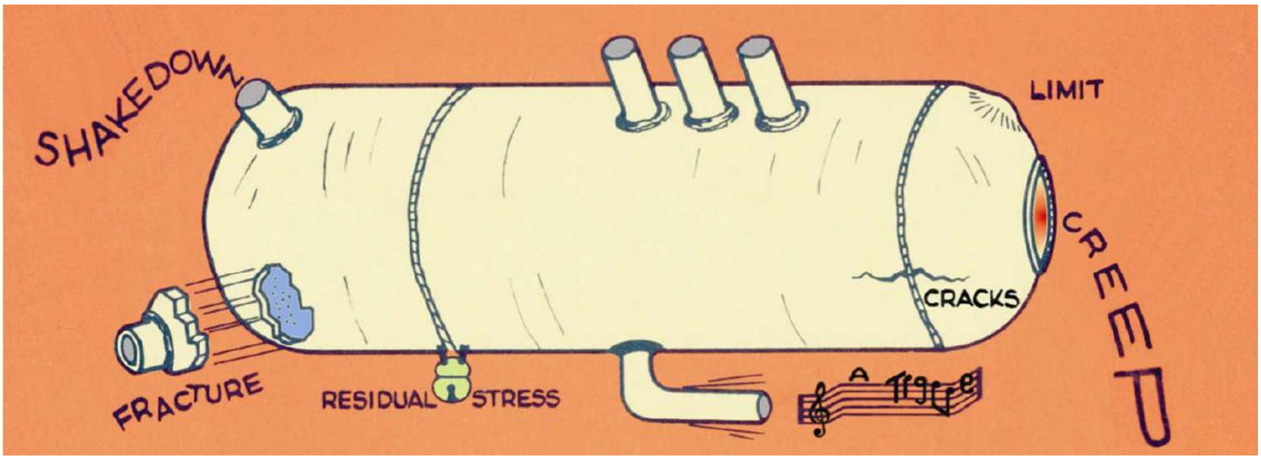


Figure 132: Pressure vessel failure modes.

6.2 Comments

Normally there is no reason to assume that the design resulting from one method is better than the design resulting from another method, especially if the method has been widely used and the designs have performed satisfactorily.

Sometimes there might be reasons to prefer or reject a method on technical background. For example, if large stress concentrations are expected in a structure, an elastic-plastic analysis method is recommended. For a total new and unproven design for which no previous experience exists, it would be wise to use different methods and compare the results.

Remember DO NOT mix the codes, this could cause SERIOUS design errors.

It will probably not be possible to resolve all conflicts between the different acceptable design approaches, but an attempt should be made to ensure consistency as far as possible. If several examples from different sources suggest that a method is too conservative or that it generates non-conservative results, a proper adjustment should be made.

Possible adjustment could be a change in the methodology, or a change in factors of safety.

---

[9] The picture is taken from the front of a Strathclyde training course leaflet from 1973.

6.2.1 Direct Route – NS-EN 13445; 2009

The Direct Route method assesses the entire structure and therefore the calculation time might be considerable. Interpreting the results require a great amount of experience especially for geometrical complex models where stress concentrations might occur.

Incorrect interpretation of the analysis results might cause an over conservative design due to the local stress concentrations being incorporated in the utilization factor calculations.

The use of partial safety factors might give an incorrect (to conservative) picture of the combined load cases for the pressure vessel, especially if the load picture is complex.

6.2.2 Elastic Stress Analysis – ASME VIII div. 2; 2010

The Elastic Stress analysis method is very quick both in terms of set up and calculation time. It is very easy to use, and the code offers clear guidelines for comparison limits and acceptance criterions.

The main problem with this method is that the calculations are ONLY done along the stress classification lines (or paths). Therefore the analysis result is strongly dependent on the placement of these lines. Failing to create a path through a "weak" zone in the pressure vessel might cause serious design errors. Identifying the "correct" zones for the stress classification lines requires great experience of structural analysis.

The linearization procedure might fail for structures with significant non-linear variation in the through thickness stress gradient. (E.g. heavy wall pressure vessels or structures with complex geometry).

6.2.3 Elastic-plastic Analysis – ASME VIII div. 2; 2010

The main drawback with this method is the time required to perform an analysis of the model. The *"true stress-true strain"* material curve has to be constructed for each material type present in the structure.

For the protection against plastic collapse no utilization factor is obtained, either the solution converges and the design is acceptable or the solution does not converge and the design is not acceptable and modifications are required. However the utilization factor can be obtained from the protection against local failure but the calculations take considerable time.

A great advantage with this method is that the plastic capacity of the material is utilized which could lead to a more optimized design in terms of material costs.

The large deformation consideration in combination with constructed material curves presents a model which is closer to the structures real behavior.

CHAPTER 7 CONCLUSIONS AND FURTHER STUDIES

7.1 Conclusions

As mentioned in the thesis there is no reason to assume that the design resulting from one method is better than the design resulting from another method, especially if the method has been widely used and the designs have performed satisfactorily.

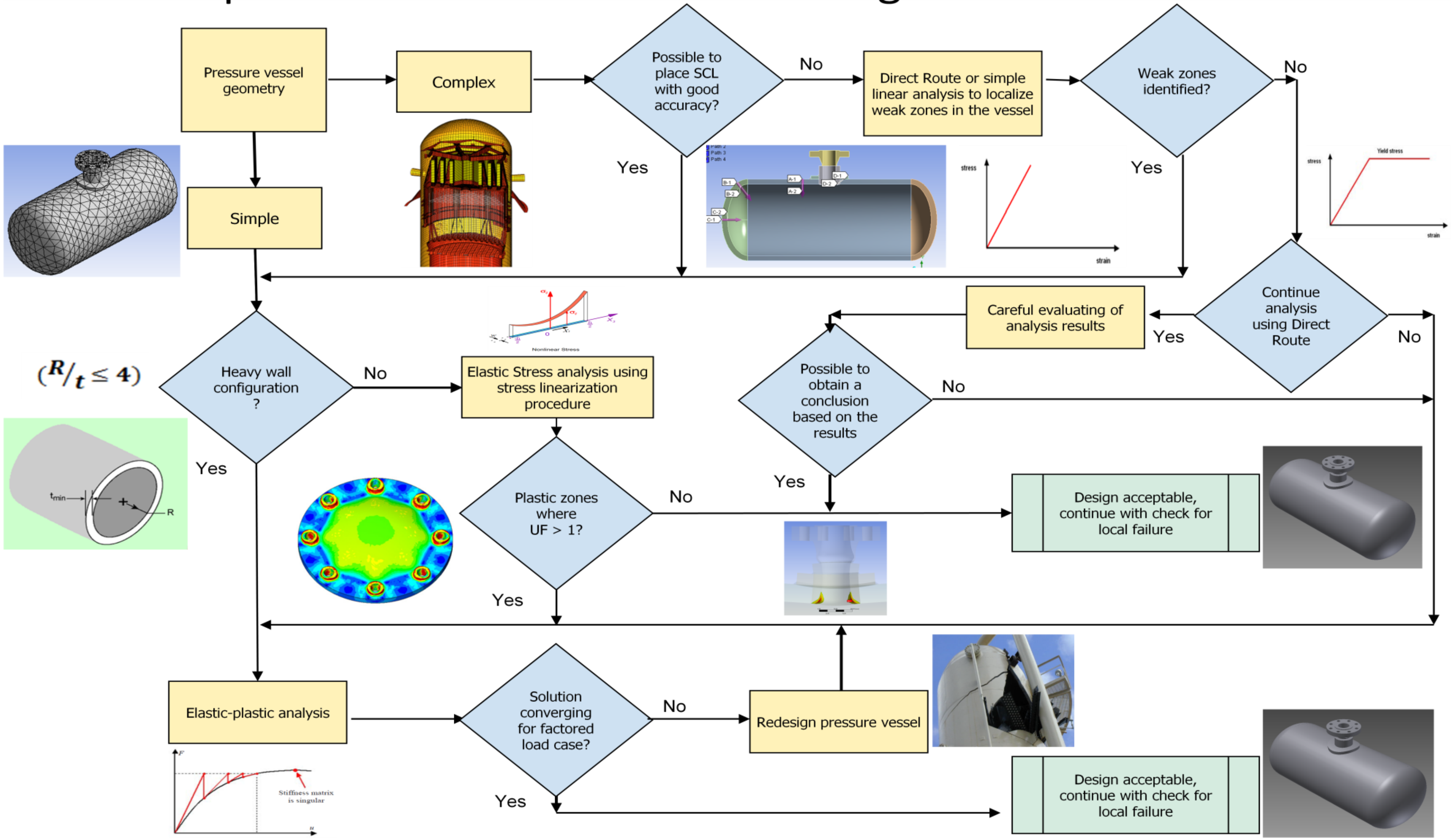
However there are significant differences and limitations to be aware of.

- **Direct Route – NS-EN 13445; 2009**
    - Offers limited descriptions and require a great amount of experience when interpreting the results.
    - Incorrect interpreting of the results might cause an over conservative design due to local stress concentrations being incorporated in the utilization factor calculations.
    - For cases with complex load picture the design might be too conservative.
    - In this comparison investigation this method represents the most conservative results.
- **Elastic Stress Analysis – ASME VIII div. 2; 2010**
    - Evaluates stresses only along the" stress classification line" and therefore the results are greatly dependent on the correct placement of these lines. Incorrect placement might cause the acceptance of non-conservative results.
    - The modeling time and calculation time is by far the shortest of the methods considered, thus for simple structures significant savings in terms of cost related to time usage might be achieved.
    - The linearization procedure might fail for structures with a significant non-linear variation in the through thickness stress gradient.
- **Elastic-plastic Analysis – ASME VIII div. 2; 2010**
    - Requires a great deal of time for modeling and the calculation time might be considerable for geometric complex structures. The calculations require high performance computing power, thus expensive computer equipment are needed.
    - A great advantage with this method is that the plastic capacity of the material is utilized which could lead to a more optimized design in terms of material costs.
    - The constructed material curves represent a model which is closer to the structures` real behavior.
    - In this comparison investigation this method represents the least conservative results.

7.1.1 Flow chart for method selection

The experience gained during the work with this thesis along with the calculation results obtained were used to construct a flow chart for simplified method selection. The intention is to use this flow chart in combination with the theoretical information, and examples given in this thesis, as a "simplified guide to pressure vessel design". Pictures used in the flow chart are previously used in this thesis and therefore no references are made. The complex pressure vessel picture in the flow chart is the internal of a nuclear reactor, Ref /18/.

Simplified flow chart for initial design method selection



Pressure vessel geometry

Complex

Possible to place SCL with good accuracy?

No

Direct Route or simple linear analysis to localize weak zones in the vessel

Weak zones identified?

No

Yes

Yes

Simple

Continue analysis using Direct Route

Careful evaluating of analysis results

Yes

No

$(R/t \leq 4)$

Heavy wall configuration ?

No

Elastic Stress analysis using stress linearization procedure

Possible to obtain a conclusion based on the results

No

Yes

Plastic zones where UF > 1?

No

Design acceptable, continue with check for local failure

Yes

Yes

Elastic-plastic analysis

Solution converging for factored load case?

No

Redesign pressure vessel

Yes

Design acceptable, continue with check for local failure

7.2 Further Studies

Due to time and size constrains for this master thesis only one design check for each method were investigated and the results compared, namely the check for total plastic collapse called:

- Gross Plastic Deformation design check by NS-EN 13445; 2009.
- Protection against plastic collapse by ASME VIII div. 2; 2010.
    - Protection against local failure by ASME VIII div. 2; 2010.

It is suggested **(strongly recommended)** to repeat the same procedure as presented in this thesis for the other design checks mentioned in the different standards, comparing the results and evaluating and commenting upon any differences discovered.

Thus comparing:

- Progressive Plastic Deformation design check by NS-EN 13445; 2009.
    - Fatigue failure design check by NS-EN 13445; 2009.
- Protection against failure from cyclic loading by ASME VIII div. 2; 2010.
    - Ratcheting assessment by ASME VIII div. 2; 2010.

And:

- Instability design check by NS-EN 13445; 2009.
- Protection against collapse from buckling by ASME VIII div. 2; 2010.

REFERENCES

/1/ Boresi, A. P. and Schmidt, R. J.; Advanced Mechanics of Materials, sixth edition, John Wiley and Sons, 2003.

/2/ Dassault Systèmes SolidWorks Corp.; Understanding Non-linear Analysis, 2008.

/3/ Institution of mechanical engineers; Advanced design and life assessment methods for pressure plants, London, 2010.

/4/ Porter M. A., Reinhardt W. D., Martens D. H.; Comparison of limit load, linear and non-linear FE analysis of a typical vessel nozzle, 2001.

/5/ Peng-fei LIU, Jin-yang ZHENG, Li MA, Cun-jian MIAO, Lin-lin WU; Calculations of plastic collapse load of pressure vessel using FEA, Zhejiang University, Hangzhou, China.

/6/ European Committee for Standardization; NS-EN 13445; 2009, Unfired Pressure vessels, Brussels.

/7/ Union de Normalisation de la Mècanique, Background to the rules in Part 3 of EN 13445, 2004.

/8/ Det Norske Veritas; DNV-OS-F101, Offshore pressure containing components, Oslo, 2010.

/9/ Zeman, J. L., The European Approach to Design by Analysis, Institute for Pressure Vessel and Plant Technology, Proceedings of the 2002 ASME PVP Conference Vancouver, Canada.

/10/ Zeman, J.L., Preiss, R., The Deviatoric Map – a simple tool to design by analysis, International Journal of Pressure Vessels and piping, 1999.

/11/ Weichert, D, A historical view on the development of shakedown theory, Presentation, Institute of General Mechanics RWTH Aachen University, Graz, 2011.

/12/ ASME (American Society of Mechanical Engineers) VIII div. 2; 2007/2010, Boiler and Pressure Vessel Code, New York.

/13/ ASME (American Society of Mechanical Engineers) II Part D; 2007, Boiler and Pressure Vessel Code, Material properties, New York.

/14/ Mackenzie, D., The Finite Element Method in Pressure Vessel Design by Analysis, University of Strathclyde, Glasgow, Scotland, presented at FENet conference in Budapest, 2005.

/15/ Bushnell, D., Computational Buckling Analysis of Shells, Martinus Nijhof Publishers, 1985.

/16/ OhmTech AS, Visual Vessel Design, user manual, 2005, http://www.ohmtech.no/

/17/ Learning Autodesk Inventor, Official training guide, Vol. 1 and vol. 2, 2010.

/18/ Scanscot Technology – Nuclear Engineering, http://www.scanscot.com/services/nuclear-engineering/ .

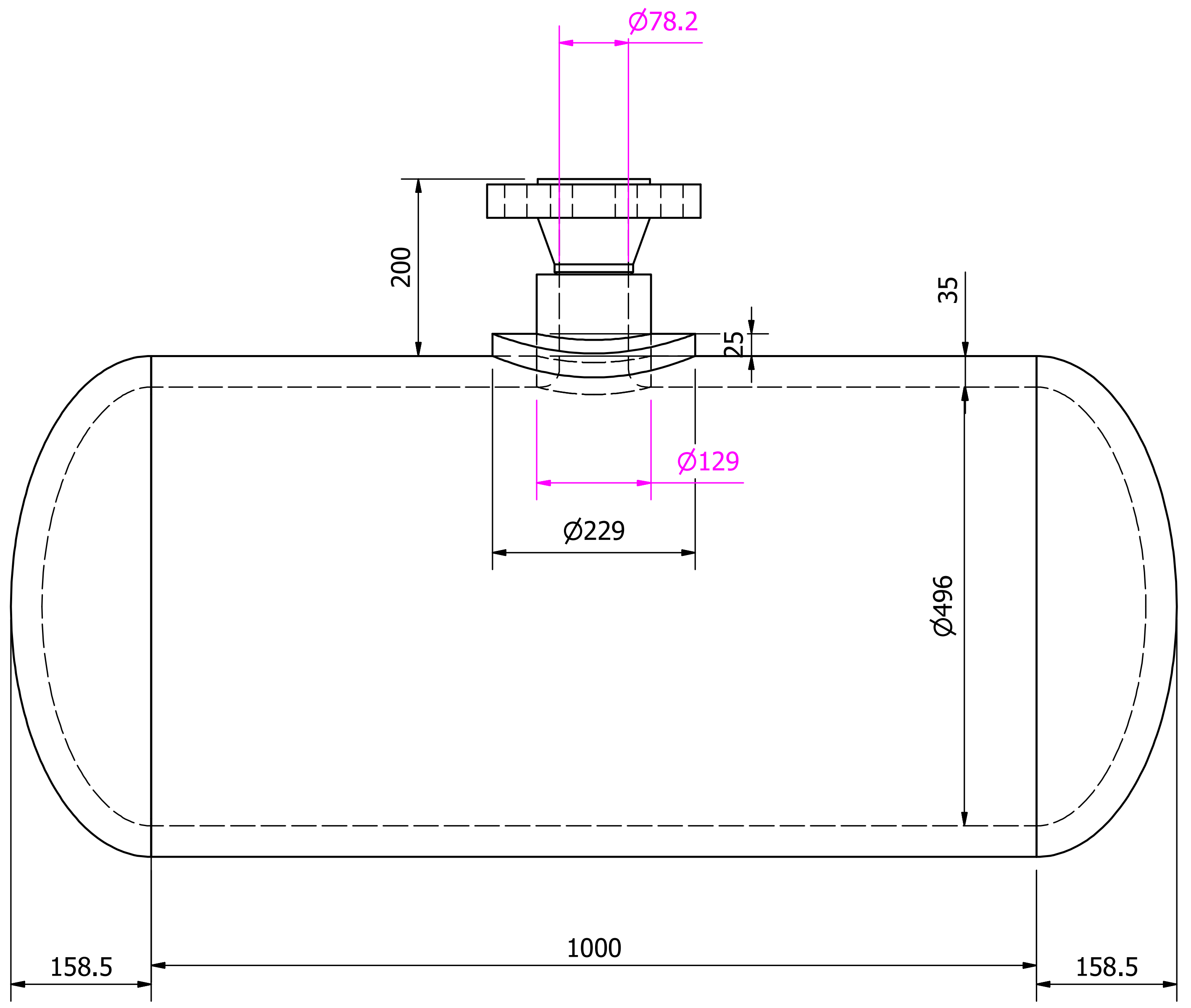
APPENDIX

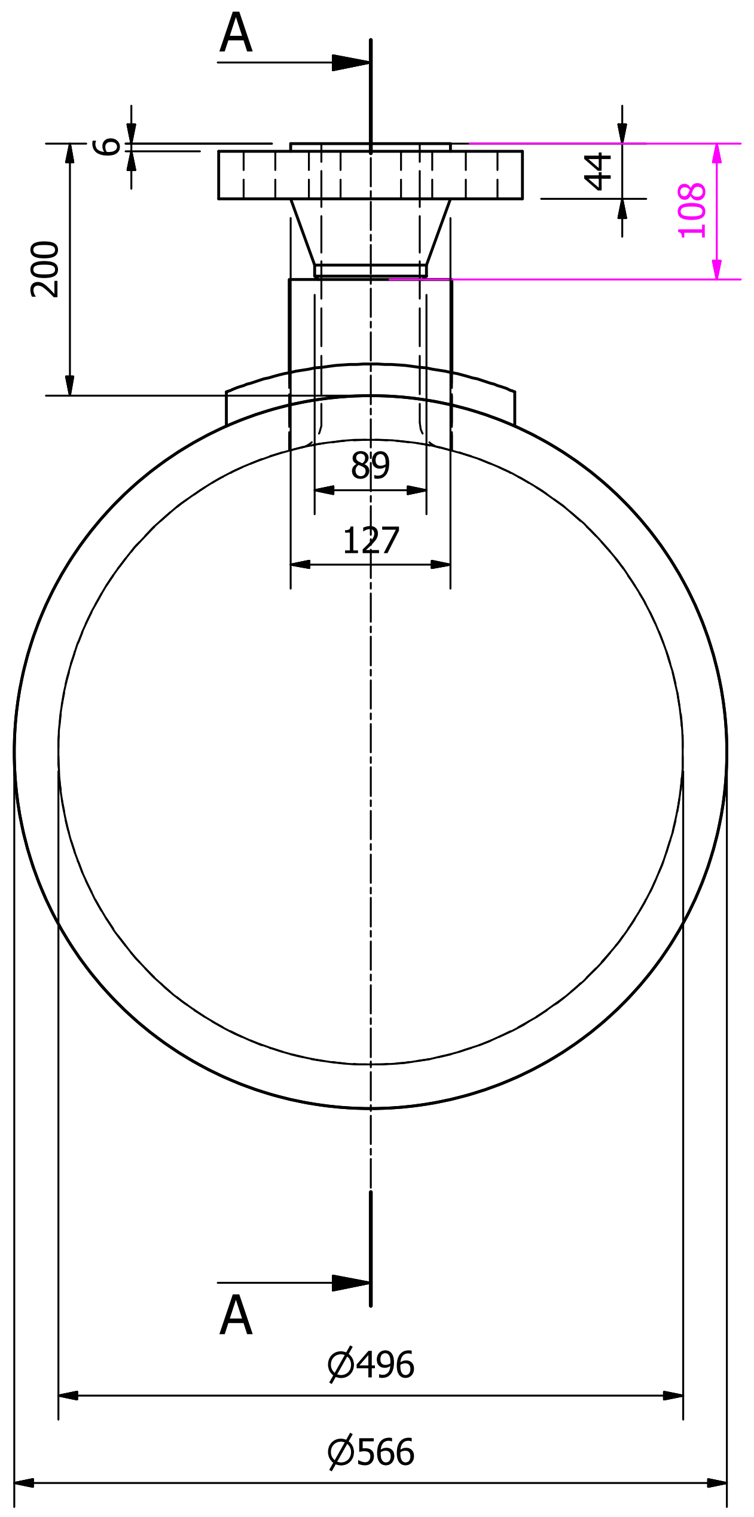
- APPENDIX A: Set of construction drawings
    - 2 sheets of A3 paper
- APPENDIX B: Calculation report from VVD (35 mm @ 100 bar)
    - 26 pages
- APPENDIX C: Calculation report from VVD (70 mm @ 200 bar)
    - 26 pages
- APPENDIX D: Calculation report from ANSYS, Direct Route – NS-EN 13445; 2009, Gross plastic deformation (35 mm @ 100 bar)
    - 13 pages
- APPENDIX E: Calculation report from ANSYS, Direct Route – NS-EN 13445; 2009, Gross plastic deformation (70 mm @ 200 bar)
    - 13 pages
- APPENDIX F: Calculation report from ANSYS, Elastic Stress Analysis - ASME VIII div. 2; 2010, Protection against plastic collapse (35 mm @ 100 bar)
    - 25 pages
- APPENDIX G: Calculation report from ANSYS, Elastic Stress Analysis - ASME VIII div. 2; 2010, Protection against plastic collapse (70 mm @ 200 bar)
    - 25 pages
- APPENDIX H: Calculation report from ANSYS, Elastic - plastic Stress Analysis, ASME VIII div. 2; 2010, Protection against plastic collapse (35 mm @ 100 bar)
    - 20 pages
- APPENDIX I: Calculation report from ANSYS Elastic - plastic Stress Analysis, ASME VIII div. 2; 2010, Protection against plastic collapse (70 mm @ 200 bar)
    - 20 pages

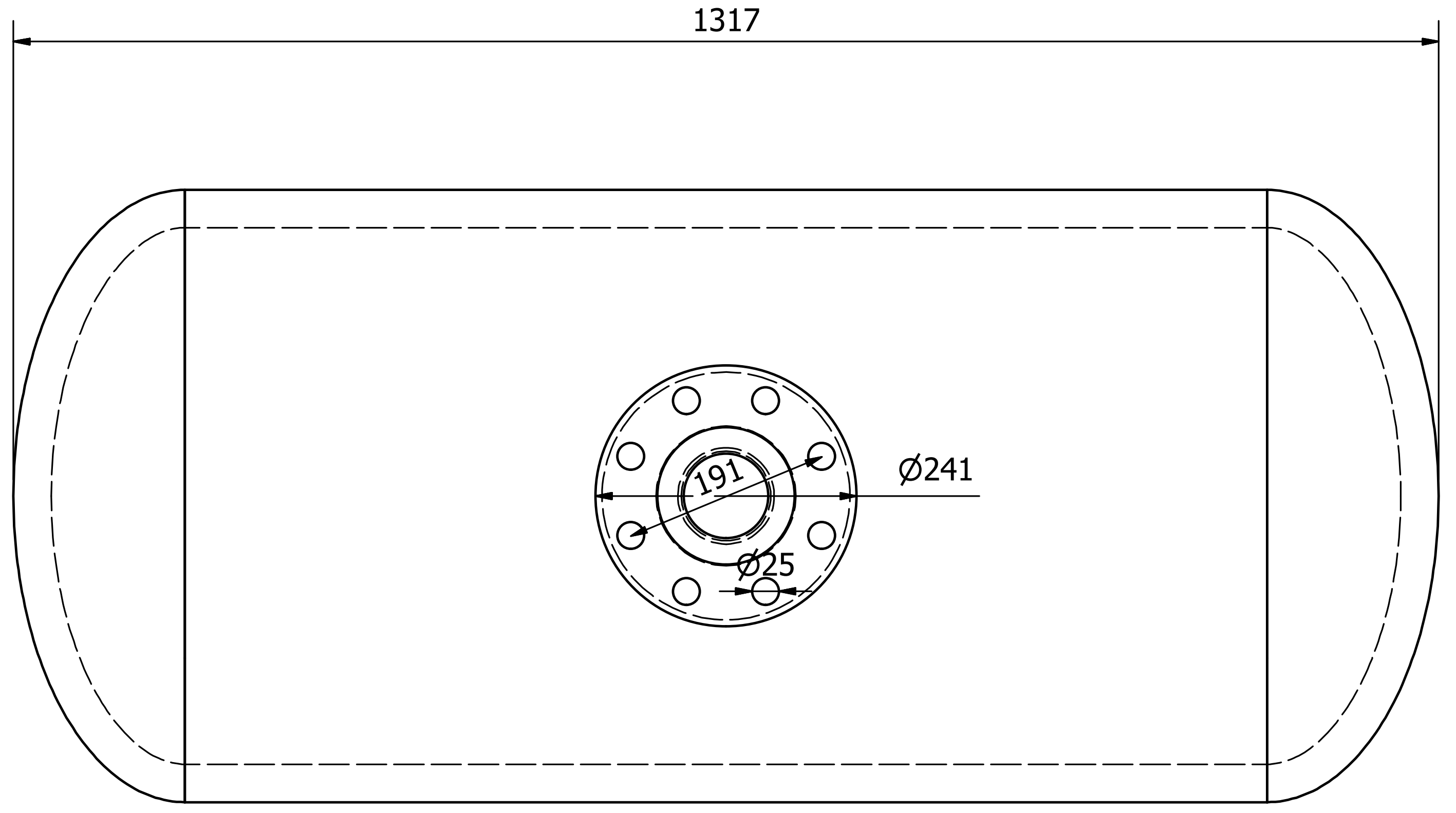
APPENDIX

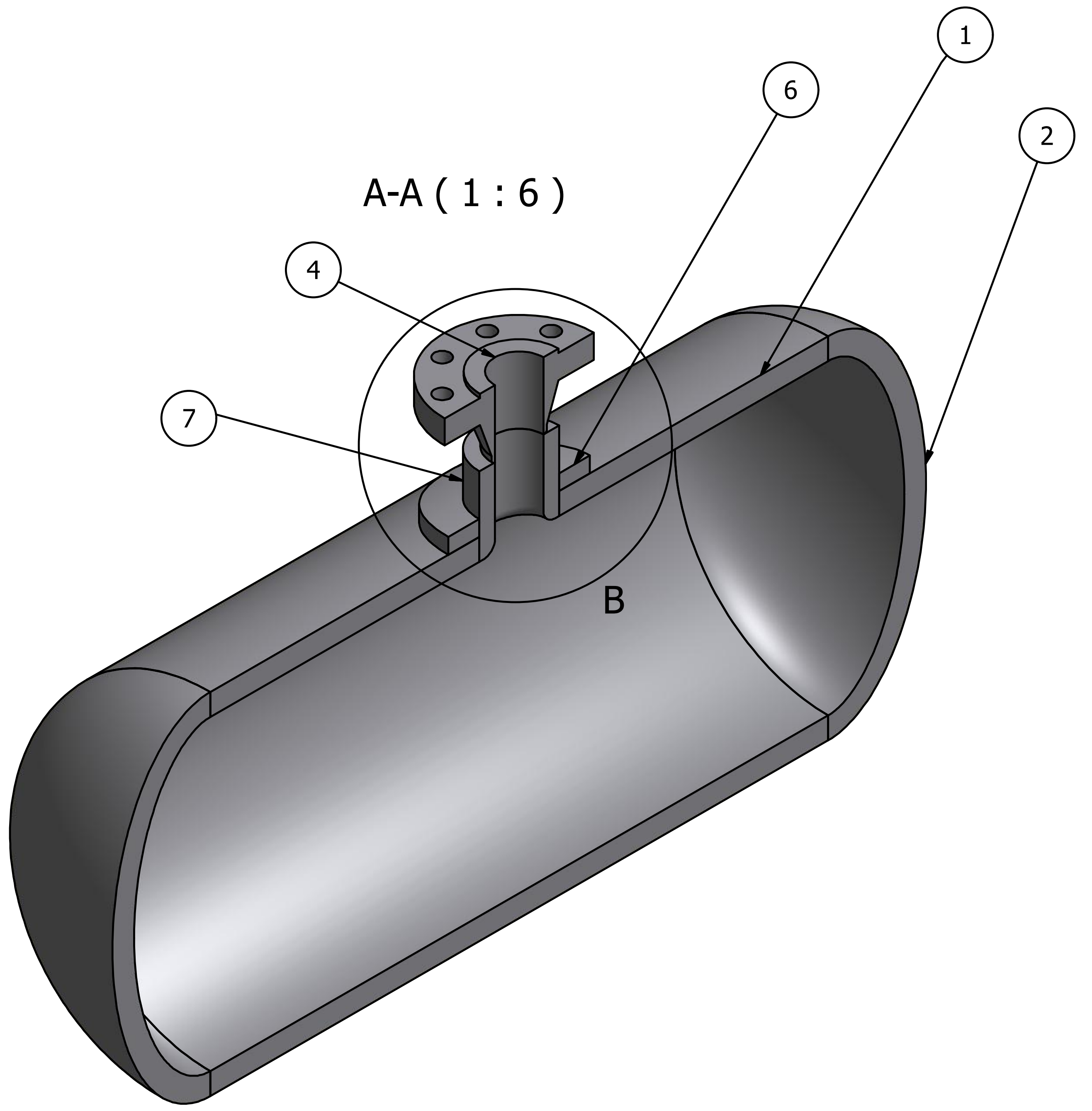
APPENDIX A: Set of construction drawings

NOTES:

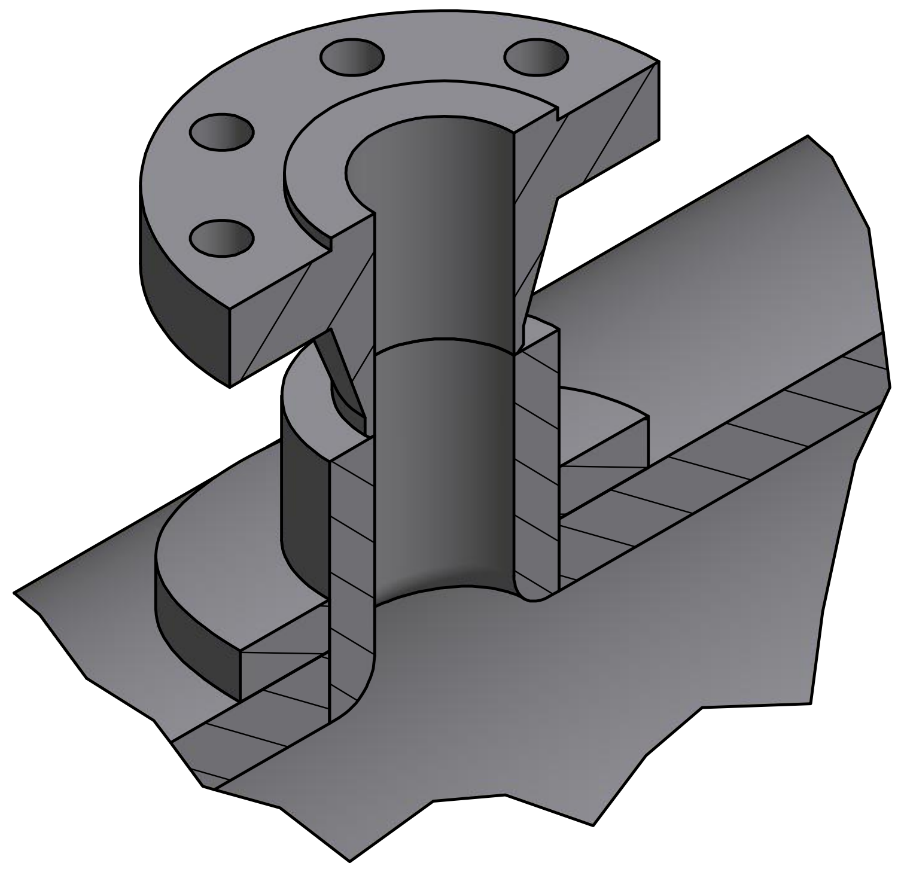








A-A (1 : 6)



B (1 : 3)

Parts List

ITEM	QTY	PART NUMBER	TITLE	MASS	MATERIAL
1	1	Main shell	Main shell	454,739 kg	Steel
2	2	End cap	End cap	85,938 kg	Steel
4	1	3 inch Weldneck Flange 900 RF	3" Weldneck Flange 900# RF	13,329 kg	Steel
6	1	Reinforcement pad2		5,679 kg	Carbon Steel
7	1	Nozzle2		8,302 kg	Carbon Steel

CLIENT REV.	REV.	DATE	DESCRIPTION	Prep'd	Chec'd	Appr'd	
				CONTRACTOR			COMPANY

CLIENT

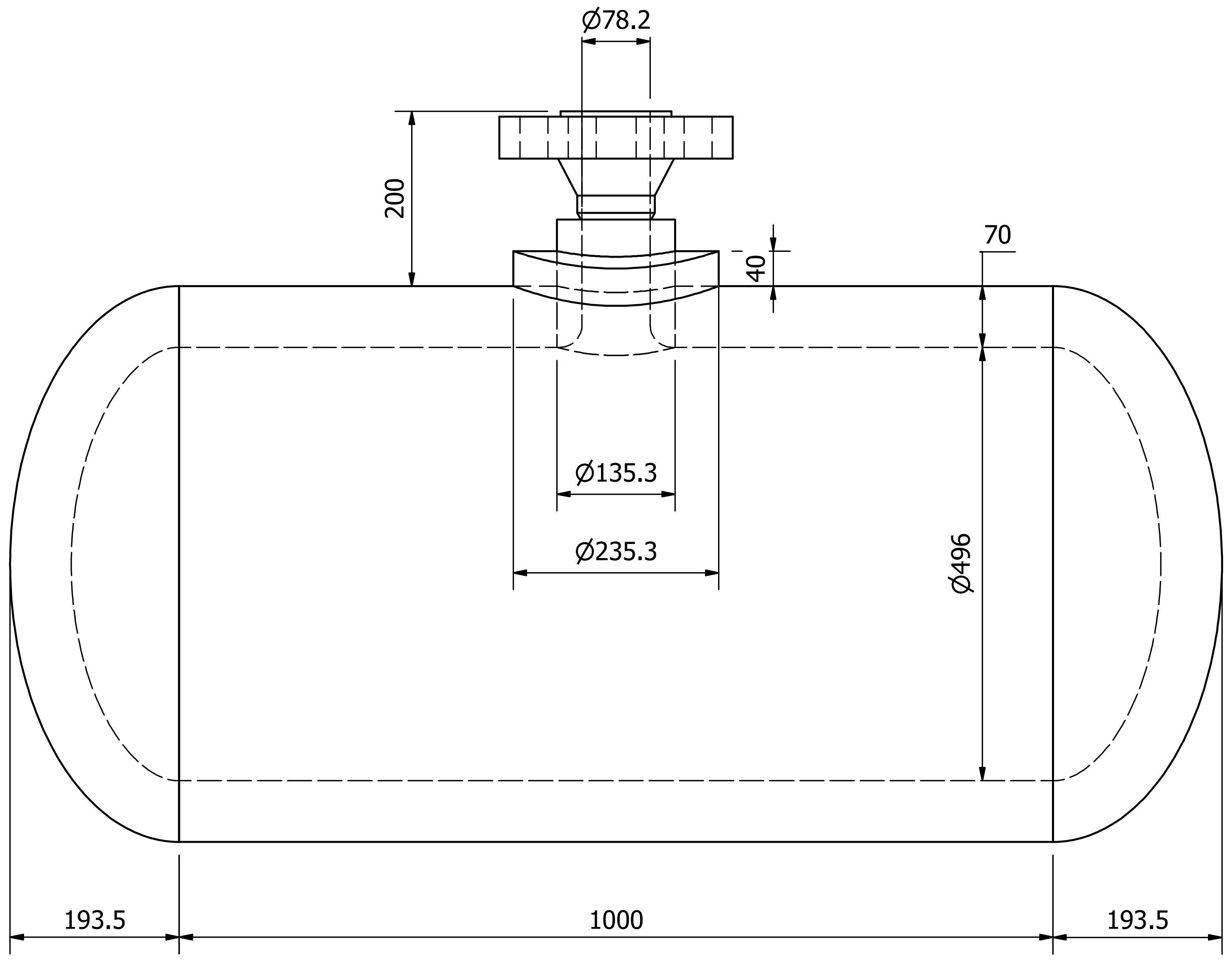
COMPANY

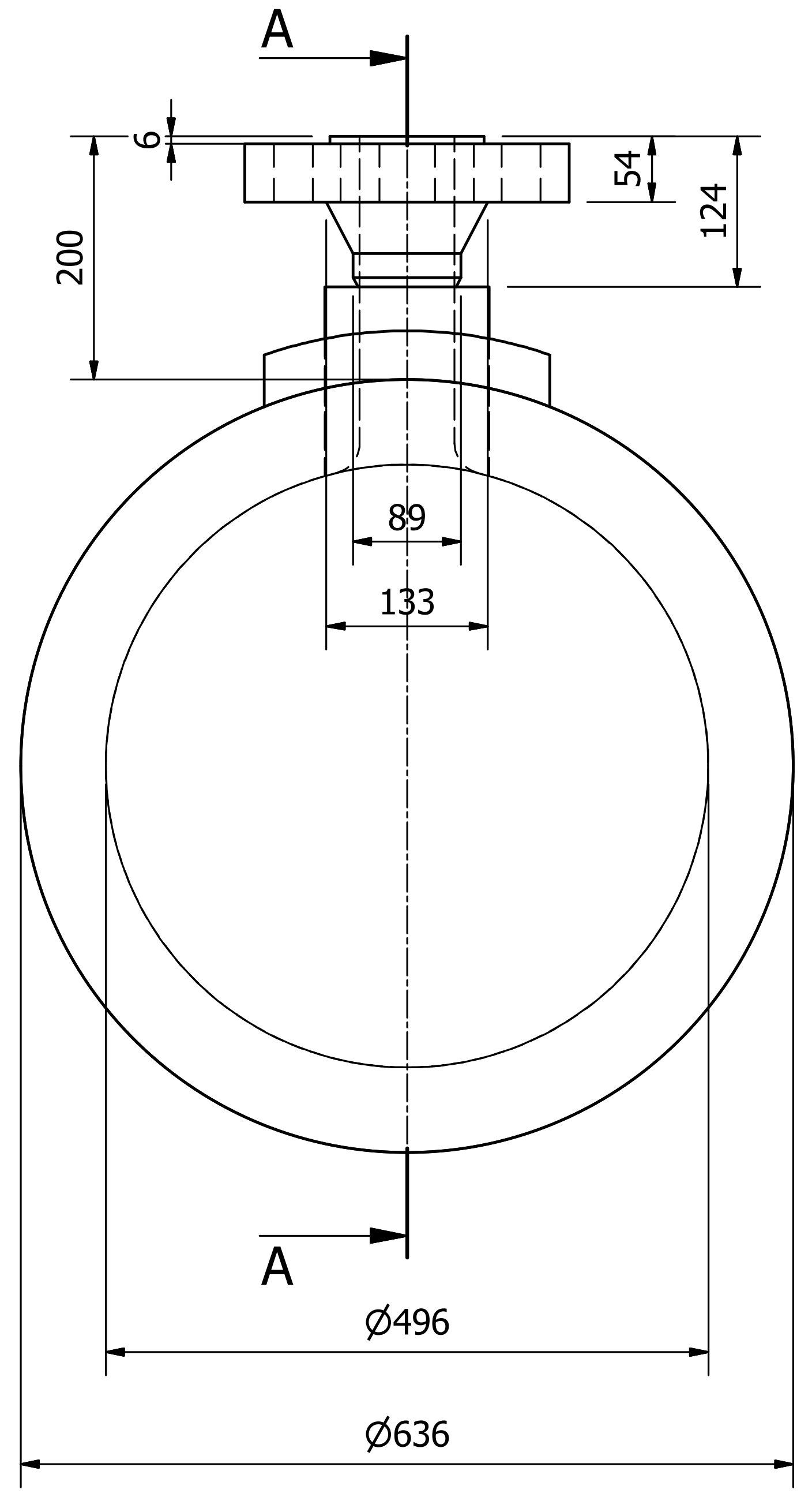
Ocean Design

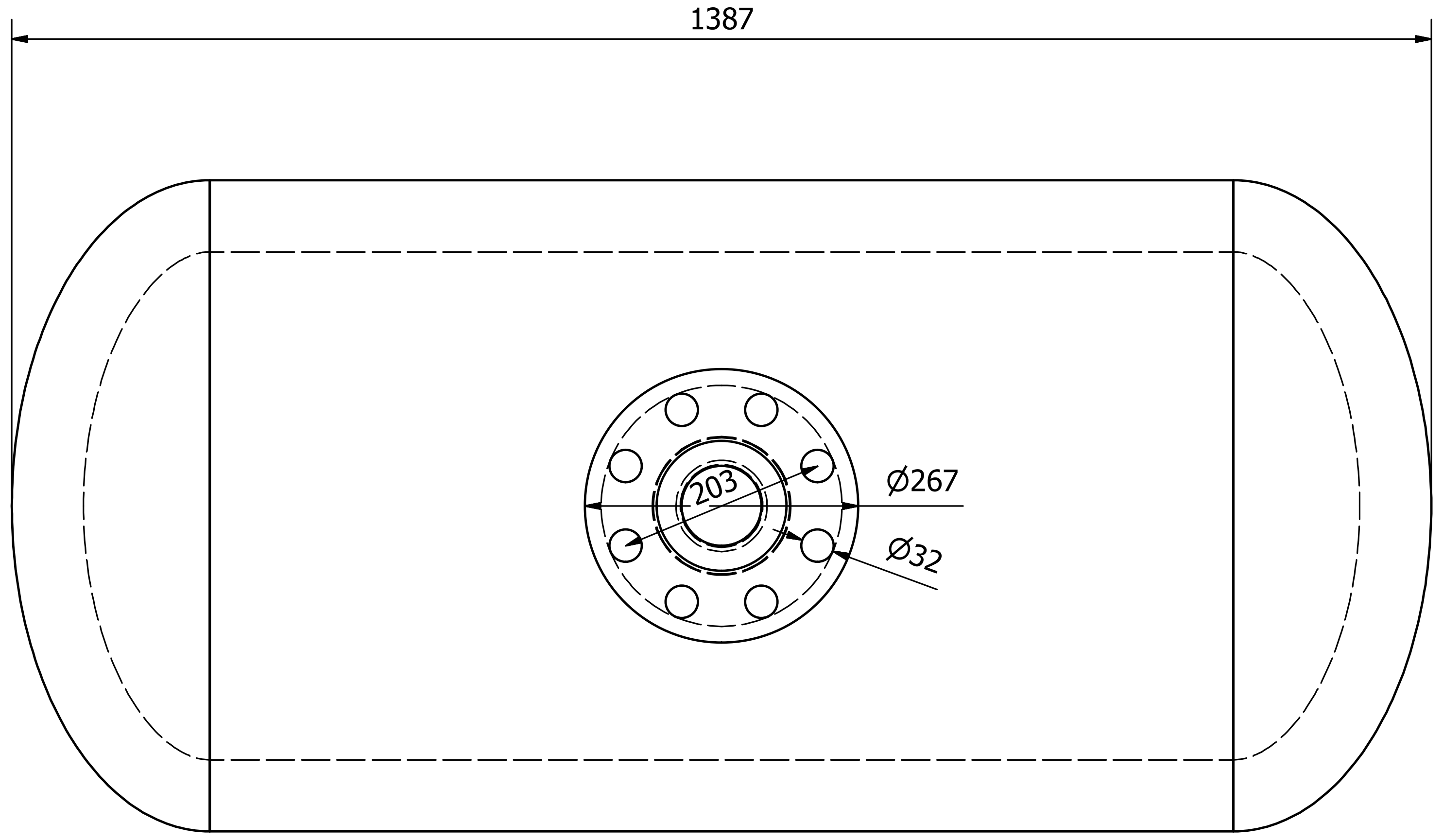
Thin wall configuration -01 01

CONTR. No.
SYSTEM / AREA
PROJECT No.
ORIG. CODE
DISC
DRAW CODE
SEQUENCE No.
SHEET No.
REV.
SCALE
ESTIMATED WEIGHT: 654 kg
TITLE
SIZE: A1
PROJECT
PRO
DRAWN BY: frodet
DRAWING START DATE
TOT. No. OF SHEETS: 1
PART No.: Main assembly
CLIENT DRAWING No.: Add Client Drw. No. In Keyword Field
CLIENT REV.

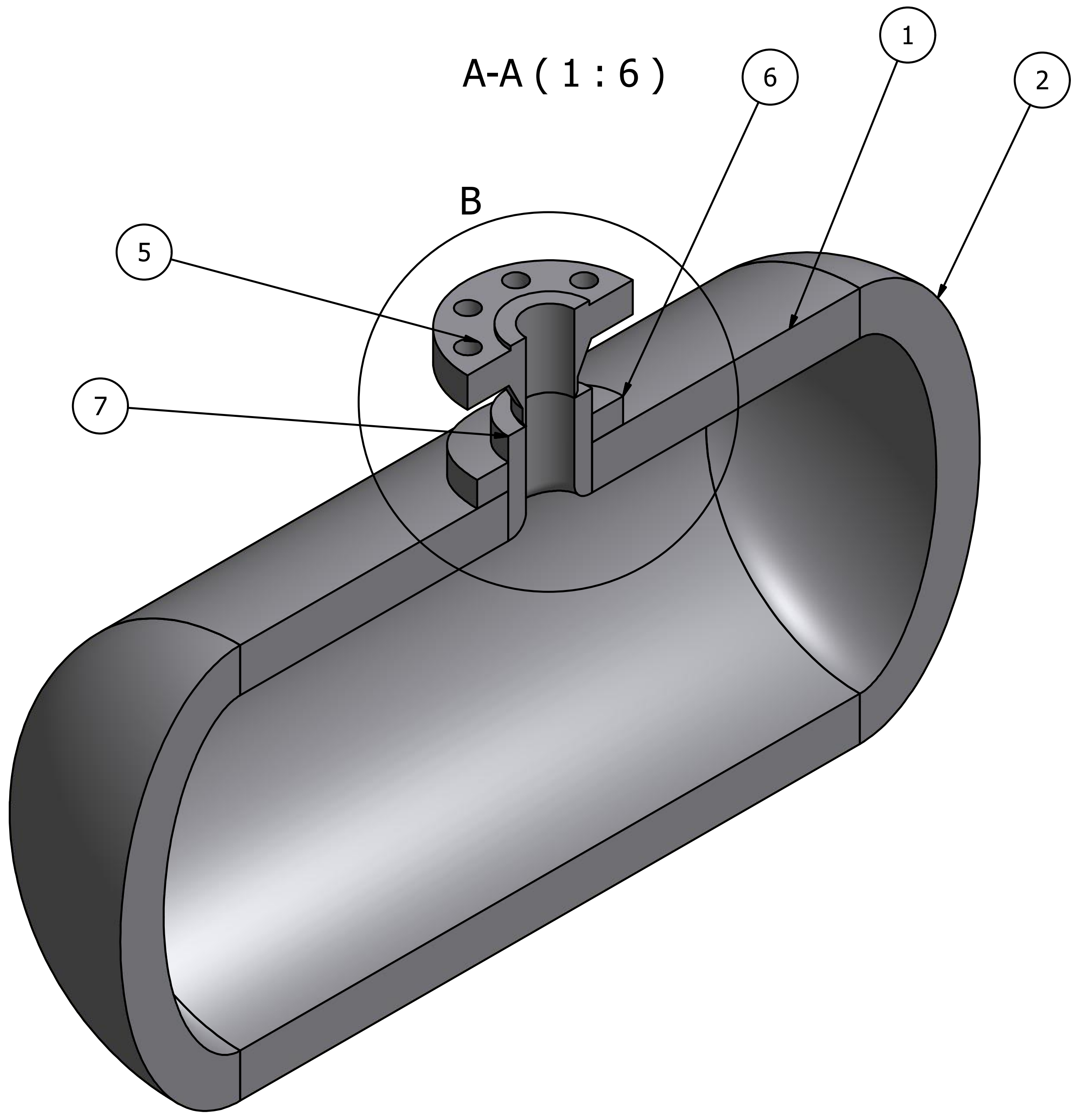
NOTES:



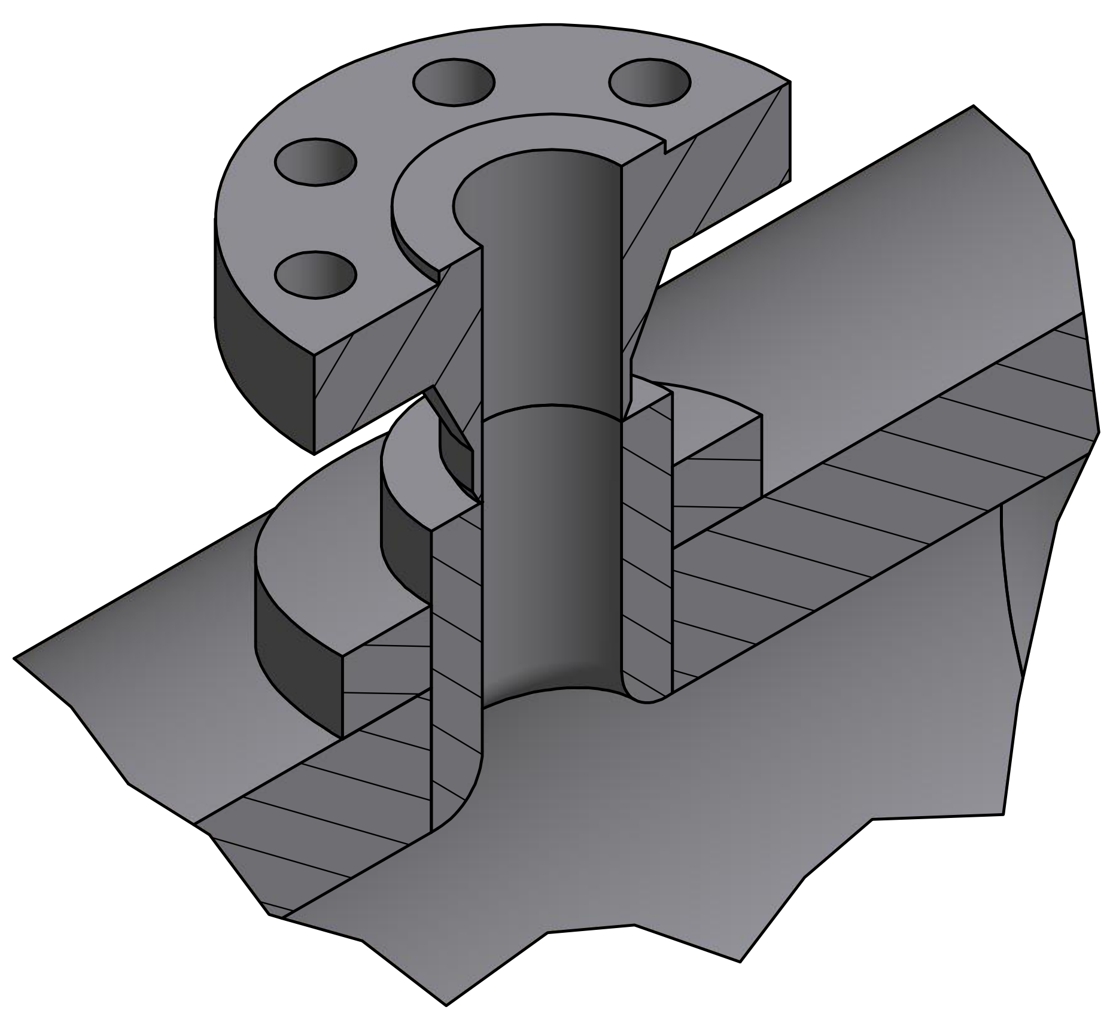




A-A (1 : 6)



B (1 : 3)



CLIENT REV.	REV.	DATE	DESCRIPTION	Prep'd	Chec'd	Appr'd	
				CONTRACTOR			COMPANY

Parts List

ITEM	QTY	PART NUMBER	TITLE	MASS	MATERIAL
1	1	Main shell	Main shell	969,133 kg	Steel
2	2	End cap	End cap	200,486 kg	Steel
5	1	3 inch Weldneck Flange 1500 RF	3 inch Weldneck Flange 1500 RF	19,175 kg	Steel
6	1	Reinforcement pad thick2		0,000 kg	Carbon Steel
7	1	Nozzle thick2		0,000 kg	Carbon Steel

CLIENT

COMPANY

Ocean Design

Thick wall configuration -01 01

CONTR. No.	SYSTEM / AREA	PROJECT No.	ORIG. CODE	DISC	DRAW CODE	SEQUENCE No.	SHEET No.	REV.

SCALE

ESTIMATED WEIGHT
1389 kg

TITLE

SIZE
A1

PROJECT

PRO

DRAWN BY
frodet

DRAWING START DATE

TOT. No. OF SHEETS
1

PART No.
Main assembly

CLIENT DRAWING No.
Add Client Drw. No. In
Keyword Field

CLIENT REV.

APPENDIX B: Calculation report from VVD (35 mm @ 100 bar)

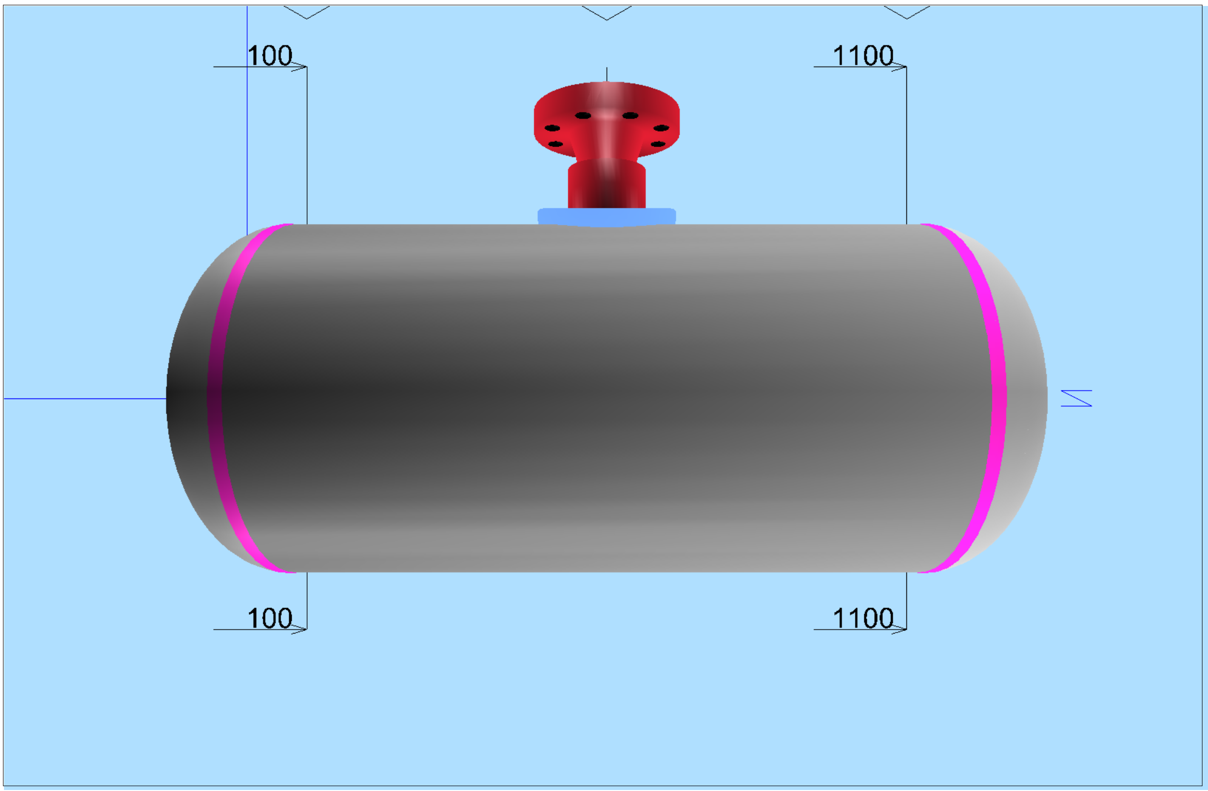
IKM Ocean Design AS

Client :Master thesis Vessel Tag No.:

Visual Vessel Design by OhmTech Ver:10.2c Operator : Rev.:A

Drawing

3D View of Vessel (alter by using the Save User Specified View command)



History of Revisions

Rev	ID	Component Type	Comp. Description	DATE & TIME
A	E2.1	Ellipsoidal End	End cap	17 Apr. 2012 17:19
A	E2.2	Ellipsoidal End	End cap	17 Apr. 2012 17:19
A	N.1	Nozzle,Forging (LWN)	Nozzle	17 Apr. 2012 17:19
A	S1.1	Cylindrical Shell	Main Shell	17 Apr. 2012 17:19

A First Issue 28 Mar. 2012 12:26

Design Data & Process Information

Description	Units	Design Data
Process Card		General Design Data
Design Code & Specifications		ASME VIII Div.1:2007 A09
Internal Design Pressure (MPa)	MPa	10
External Design Pressure (MPa)	MPa	0
Hydrotest Pressure (MPa)	MPa	
Maximum Design Temperature ('C)	'C	100
Minimum Design Temperature ('C)	'C	20
Operating Temperature ('C)	'C	
Corrosion Allowance (mm)	mm	1
Content of Vessel		
Specific Density of Oper.Liq		1

Weight & Volume of Vessel

ID	No.	Wt-UnFinish.	Wt-Finished	Tot.Volume	Test.Liq.Wt	Oper.Liq.Wt
S1.1	1	459.0 kg	455.4 kg	0.196 m3	196.0 kg	96.6 kg
N.1	1	31.0 kg	31.0 kg	0.001 m3	1.0 kg	0.0 kg
E2.1	1	84.0 kg	84.0 kg	0.016 m3	16.0 kg	10.3 kg
E2.2	1	84.0 kg	84.0 kg	0.016 m3	16.0 kg	10.3 kg
Total	**4**	**658.0 kg**	**654.4 kg**	**0.229 m3**	**229.0 kg**	**117.3 kg**

Weight Summary/Condition	Weights
Empty Weight of Vessel incl. 5% Contingency	687 kg
Total Test Weight of Vessel (Testing with Water)	916 kg
Total Operating Weight of Vessel	804 kg

Center of Gravity

ID	X-Empty	Y-Empty	Z-Empty	X-Test	Y-Test	Z-Test	X-Oper	Y-Oper	Z-Oper
S1.1	-2	0	599	0	0	600	-105	0	600
N.1	376	0	600	352	0	600	352	0	600
E2.1	0	0	38	0	0	54	-100	0	-2
E2.2	0	0	1162	0	0	1146	-100	0	1202

CENTER OF GRAVITY AT CONDITIONS BELOW	X	Y	Z
Empty Vessel	16	0	599
Test Condition of Vessel (Testing with Water)	12	0	600
Operating Condition of Vessel	-2	0	600

Max. Allowable Pressure MAWP

ID	Comp. Type	Liq.Head	MAWP New & Cold	MAWP Hot & Corr.
S1.1	Cylindrical Shell	0.000 MPa	17.488 MPa	16.949 MPa
N.1	Nozzle,Forging (LWN)	0.000 MPa	29.000 MPa	27.875 MPa
E2.1	Ellipsoidal End	0.000 MPa	18.666 MPa	18.124 MPa
E2.2	Ellipsoidal End	0.000 MPa	18.666 MPa	18.124 MPa
	MAWP		**17.488 MPa**	**16.949 MPa**

Note : Other components may limit the MAWP than the ones checked above.
Note : The value for MAWP is at top of vessel, with static liquid head subtracted.

Test Pressure

UG-99(b) REQUIRED MINIMUM TEST PRESSURE.

TEST PRESSURE OF VESSEL - NEW & COLD - HORIZONTAL

Design Pressure.........................: 10.000 MPa
Design Temperature......................: 100.0 C

ID	Description	Pdesign	PtMax	PtMin	Wat.Head	PtTop	PtTopMax
S1.1	Cylindrical Shell-Main Shell	10.000	29.423	13.000	0.008	12.992	29.415
N.1	3" ANSI B16.5 900 lbs WN -RF Raised Face	10.000	23.096	NA	0.003	NA	23.092
N.1	Nozzle,Forging (LWN)-Nozzle	10.000	49.000	NA	0.003	NA	48.997
E2.1	Ellipsoidal End-End cap	10.000	31.521	13.000	0.005	12.995	31.516
E2.2	Ellipsoidal End-End cap	10.000	31.521	13.000	0.008	12.992	31.513

HYDRO-TEST

```
REQUIRED TEST PRESSURE AT TOP OF VESSEL PtReq(Hydro Test) ......:    13.000 MPa
MAXIMUM TEST PRESSURE AT TOP OF VESSEL PtLim(Hydro Test) .......:    23.092 MPa

Note : Other components may limit Ptlim than the ones checked above.
```

NOMENCLATURE:

Pdesign- is the design pressure including liquid head at the part under consideration.

PtMax - is the maximum allowed test pressure determined at the part under consideration.

PtMin - is the required test pressure determined at the part under consideration.

Wat.Head - is the water head during hydrotesting at the part under consideration.

PtBot - is the required test pressure at bottom of the vessel, for the part under consideration.

PtTop - is the required test pressure at top of the vessel, for the part under consideration.

PtTopMax - is the maximum test pressure allowed at top of the vessel, for the part under consideration.

PtReq - is the required minimum test pressure (largest value of PtTop) at top of vessel for the listed components.

PtLim - is the maximum allowed test pressure (minimum value for PtTopMax) at top of vessel for the listed components.

Bill of Materials

ID	No	Description	Component Dimensions	Material Standard
E2.1	1	Ellipsoidal End-End cap	Do= 566, t= 35, h= 158.5, Semi-Ellipsoidal Head R:h 2:1	ID 1, SA-516(M) Gr.70, K02700 Plate
E2.2	1	Ellipsoidal End-End cap	Do= 566, t= 35, h= 158.5, Semi-Ellipsoidal Head R:h 2:1	ID 1, SA-516(M) Gr.70, K02700 Plate
N.1	1	Flange:ANSI B16.5:Class 900 lbs	WN Welding Neck, 1a RF Raised Face	1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)
N.1	1	Nozzle,Forging (LWN)-Nozzle	3" do=127,t=25.39,L=243.5,ho=200,PAD OD=229	ID 1, SA-516(M) Gr.70, K02700 Plate
N.1	1	Reinforcement Pad	PAD OD=229, wt= 25, width= 50	ID 1, SA-516(M) Gr.70, K02700 Plate
S1.1	1	Cylindrical Shell-Main Shell	Do= 566, t= 35, L= 1000	ID 1, SA-516(M) Gr.70, K02700 Plate

Notes, Warning & Error Messages

ID & Comp. Description	Notes/Warnings/Error Messages
S1.1 Cylindrical Shell Main Shell	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**
N.1 Nozzle,Forging (LWN) Nozzle	
-	NOTE: Pad thickness is less than recommended minimum value of 75% of shell thickness.
E2.1 Ellipsoidal End End cap	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**
E2.2 Ellipsoidal End End cap	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**

TOTAL No. OF ERRORS/WARNINGS : 3

Nozzle List

ID	Service	SIZE	STANDARD/CLASS	ID	Standout	X	Y	Z	Rot.	Orient.
N.1	Nozzle	3"	ANSI B16.5 900 lbs WN -RF Raised Face CLASS :900 LWN Long Welding Neck	78.19	200	265.5	0	600	0	Radial

Nozzle Loads

ID	Load Desc.	Nozzle Loads
N.1	Nozzle	Fz=30kN,My=0,Mx=0,Mt=0kNm,Fl=0,Fc=0kN

Maximum Component Utilization - Umax

ID	Comp.Type	Umax(%)	Limited by
S1.1	Cylindrical Shell	59.5%	Internal Pressure
N.1	Nozzle,Forging (LWN)	59.7%	PhiAll AT EDGE OF PAD
E2.1	Ellipsoidal End	57.4%	Internal Pressure
E2.2	Ellipsoidal End	57.4%	Internal Pressure

Component with highest utilization Umax = 59.7% N.1 Nozzle

Average utilization of all components Umean= 58.5%

Material Data/Mechanical Properties

ID	Material Name	Temp	ST	SY	SYd	S_d	Sr	ftest	E-mod	Note
1	SA-516(M) Gr.70, K02700 Plate , SG=7.85	100	485	260	239	138.6	138.6	234	0	

Notation:
Thickness in mm, stress in N/mm2, temperature in deg.C
TG : Test Group 1 to 4
Max.T: Maximum thickness for this stress set, 0 or 999 = No limit specified
S/C : CS = Carbon Steel, SS = Stainless Steel
SG : SG = Specific Gravity (Water = 1.0)
ST : MIN.TENSILE STRENGTH at room temp.
SY : MIN. YIELD STRENGTH at room temp.
SYd : MIN. YIELD STRENGTH at calc.temp.
S_d : DESIGN STRESS at calc.temp.
Sr : DESIGN STRESS at room temp.

Comp.Location in Global Coord.System

ID	Comp. Type	X	Y	Z	Teta	Phi	ConnID
E2.1	Ellipsoidal End	0	0	100	0.0	0.0	S1.1
E2.2	Ellipsoidal End	0	0	1100	0.0	0.0	S1.1
N.1	Nozzle,Forging (LWN)	266	0	600	90.0	360.0	S1.1
S1.1	Cylindrical Shell	0	0	100	0.0	0.0	

The report above shows the location of the connecting point (x, y and z) for each component referenced to the coordinate system of the connecting component (ConnID). The connecting point (x, y and z) is always on the center axis of rotational symmetry for the component under consideration, i.e. the connecting point for a nozzle connected to a cylindrical shell will be at the intersection of the nozzle center axis and the mid thickness of the shell referenced to the shell s coordinate system. In addition the orientation of the the center axis of the component is given by the two angles Teta and Phi, where Teta is the angle between the center axis of the two components and Phi is the orientation in the x-y plane

The basis for the coordinate system used by the software is a right handed coordinate system with the z-axis as the center axis of rotational geometry for the components, and Teta as the Polar Angle and Phi as the Azimuthal Angle

MDMT Minimum Design Metal Temperature

Table :

ID-Description	Material Name	tn(mm)	tg(mm)	Ratio	E(*)	Curve
E2.1 End cap - End	SA-516(M) Gr.70, K02700 Plate	35.0	35.0	0.57	1.00	D
E2.2 End cap - End	SA-516(M) Gr.70, K02700 Plate	35.0	35.0	0.57	1.00	D
N.1 Nozzle - Flange	1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)	0.0	0.0	3.52	1.00	NA
N.1 Nozzle - Nozzle	SA-516(M) Gr.70, K02700 Plate	25.4	25.4	0.12	1.00	D
N.1 Nozzle - Pad	SA-516(M) Gr.70, K02700 Plate	25.0	25.0	0.59	1.00	D
S1.1 Main Shell - Shell	SA-516(M) Gr.70, K02700 Plate	35.0	35.0	0.59	1.00	D

Table Continued

ID-Description	T1(C)	T2(C)	MDMT(C)	Comments
E2.1 End cap - End	-27.9	-24.8	-52.7	Material Normalized or Quenced and Tempered
E2.2 End cap - End	-27.9	-24.8	-52.7	Material Normalized or Quenced and Tempered
N.1 Nozzle - Flange	-28.8	0.0	-28.8	The ASME B16.5 nozzle flange has an unadjusted MDMT of -28.8 C
N.1 Nozzle - Nozzle	-35.0	-80.0	-115	Material Normalized or Quenced and Tempered
N.1 Nozzle - Pad	-35.2	-23.7	-58.9	Material Normalized or Quenced and Tempered
S1.1 Main Shell - Shell	-27.9	-23.7	-51.6	Material Normalized or Quenced and Tempered

```
MDMT CALCULATIONS PER UCS-66
MDMT Required        :  20.0 C
MDMT Lowest Allowable: -28.8 C
```

NOMENCLATURE:

tn - Nominal thickness of component under consideration(including corr. allow.).

tg - Governing thickness of component under consideration.

Ratio- tr*E(*)/(tn-c), utiliztion of component for given process conditions.

tr - Required minimum thickness of component at calculation temperature of MDMT.

E(*) - Joint efficiency factor, not lower than 0.8.

Curve- Applicable curve A, B, C or D in Figure UCS-66.

T1 - Unadjusted MDMT/Lowest allowable temperature for given part, value taken from Figure UCS-66 based on curve A, B, C or D.

T2 - Reduction in MDMT without impact testing per Figure UCS-66.1.

NOTES:

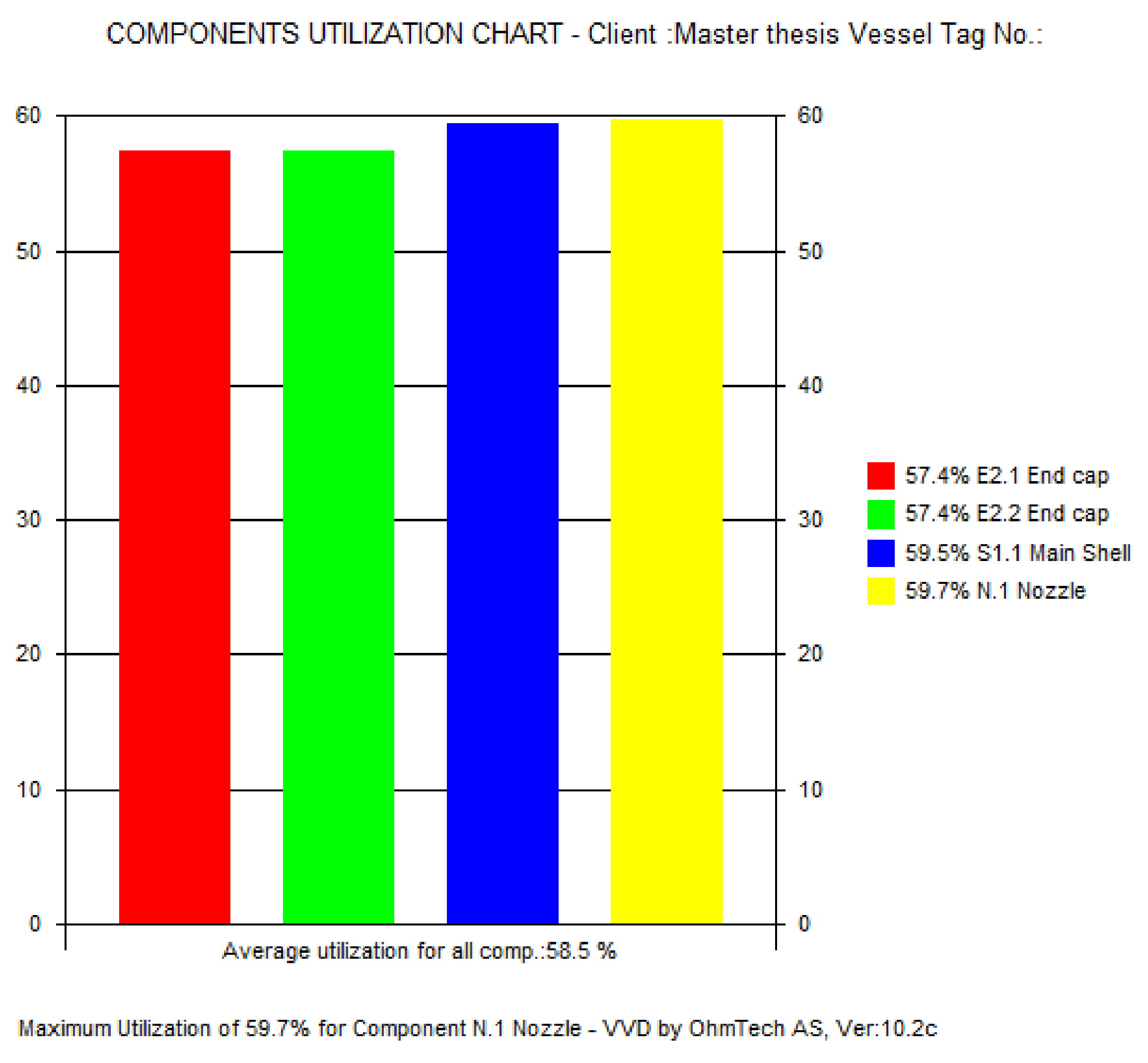
UCS-68(c) If postweld heat treatment is performed when it is not otherwise a requirement, a 17C reduction in impact test exemption temp. may be given to the min. permissible temp. for P.no.1 materials.

The maximum general primary stress in the pads are conservatively assumed to be the same as that in the corroded shell.

NOTE: LOWEST MDMT = -28.8 C (Warmest Value)

Utilization Chart

Utilization Chart



Surface Area

Table Surface Area:

ID	No.	Description	Area Outside(m2)	Area Inside(m2)
S1.1	1	Cylindrical Shell, Main Shell	1.778	1.571
N.1	1	Nozzle,Forging (LWN), Nozzle	0.080	0.048
E2.1	1	Ellipsoidal End, End cap	0.324	0.286
E2.2	1	Ellipsoidal End, End cap	0.324	0.286
Total	**4**		**2.506**	**2.191**

ASME VIII Div.1:2007 A09 - UG-27 CYLINDRICAL SHELL
S1.1 Main Shell 17 Apr. 2012 17:19

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=10.000 MPa, c= 1 mm, Pext=0.000 MPa
```

SHELL DATA

```
CYLINDER FABRICATION: Plate Material
DIAMETER INPUT: Base Design on Shell Inside Diameter
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138.57 Sr=138.57 Stest=234  (N/mm2)
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        500.00 mm
LENGTH OF CYLINDRICAL PART OF SHELL.................:Lcyl     1000.00 mm
AS BUILT WALL THICKNESS (uncorroded)................:tn         35.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
Split shell into several shell courses and include welding information: NO
```

CALCULATION DATA

UG-27 - CYLINDRICAL SHELLS UNDER INTERNAL PRESSURE

```
Inside Radius of Shell
R = Di / 2 =500/2=                                         250.00 mm
```

»Thin Cylinder Check P=10 <= 0.385 * S * E=53.35[MPa] « » OK«

```
Required Minimum Shell Thickness Excl.Allow. tmin :
tmin = P * R / (S * E - 0.6 * P)                                 (UG-27(1))
=10*250/(138.57*1-0.6*10)=                               18.86 mm
```

»Thin Cylinder Check tmin=18.86 < 0.5 * R=125[mm] « » OK«

```
Required Minimum Shell Thickness Incl.Allow. :
tmina = tmin + c + th =18.86+1+1=                          20.86 mm

Analysis Thickness
ta = tn - c - th =35-1-1=                                  33.00 mm
```

»Internal Pressure tmina=20.86 <= tn=35[mm] « » (U= 59.5%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :

```
Outside Diameter of Shell
Do = Di + 2 * ta =500+2*33=                                566.00 mm
Outside Radius of Shell
Ro = Do / 2 =566/2=                                        283.00 mm
MAWP HOT & CORR. (Corroded condition at design temp.)
MAWPHC = S * E * ta / (R + 0.6 * ta)
=138.57*1*33/(250+0.6*33)=                               16.95 MPa

MAWP NEW & COLD (Uncorroded condition at ambient temp.)
MAWPNC = Sr * E * (ta + c) / (R - c + 0.6 * (ta + c))
=138.57*1*(33+1)/(250-1+0.6*(33+1))=                     17.49 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)
Ptmax = SYtest * Etest * (ta + c) / (R + 0.6 * (ta + c))
=234*1*(33+1)/(250+0.6*(33+1))=                          29.42 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*10*138.57/138.57=             13.00 MPa
```

»Test Pressure Ptmin=13 <= Ptmax=29.42[MPa] « » (U= 44.1%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 50 * tn / Rf * (1 - Rf / INFINITY)
=50*35/265.5*(1-265.5/Infinity)=                              6.59 %
```

CALCULATION SUMMARY

UG-27 - CYLINDRICAL SHELLS UNDER INTERNAL PRESSURE

```
Required Minimum Shell Thickness Excl.Allow. tmin :
tmin = P * R / (S * E - 0.6 * P)                                     (UG-27(1))
=10*250/(138.57*1-0.6*10)=                                   18.86 mm

Required Minimum Shell Thickness Incl.Allow. :
tmina = tmin + c + th =18.86+1+1=                              20.86 mm
```

»Internal Pressure tmina=20.86 <= tn=35[mm] « » (U= 59.5%) OK«

```
MAWP HOT & CORR. (Corroded condition at design temp.)
MAWPHC = S * E * ta / (R + 0.6 * ta)
=138.57*1*33/(250+0.6*33)=                                   16.95 MPa

MAWP NEW & COLD (Uncorroded condition at ambient temp.)
MAWPNC = Sr * E * (ta + c) / (R - c + 0.6 * (ta + c))
=138.57*1*(33+1)/(250-1+0.6*(33+1))=                         17.49 MPa
```

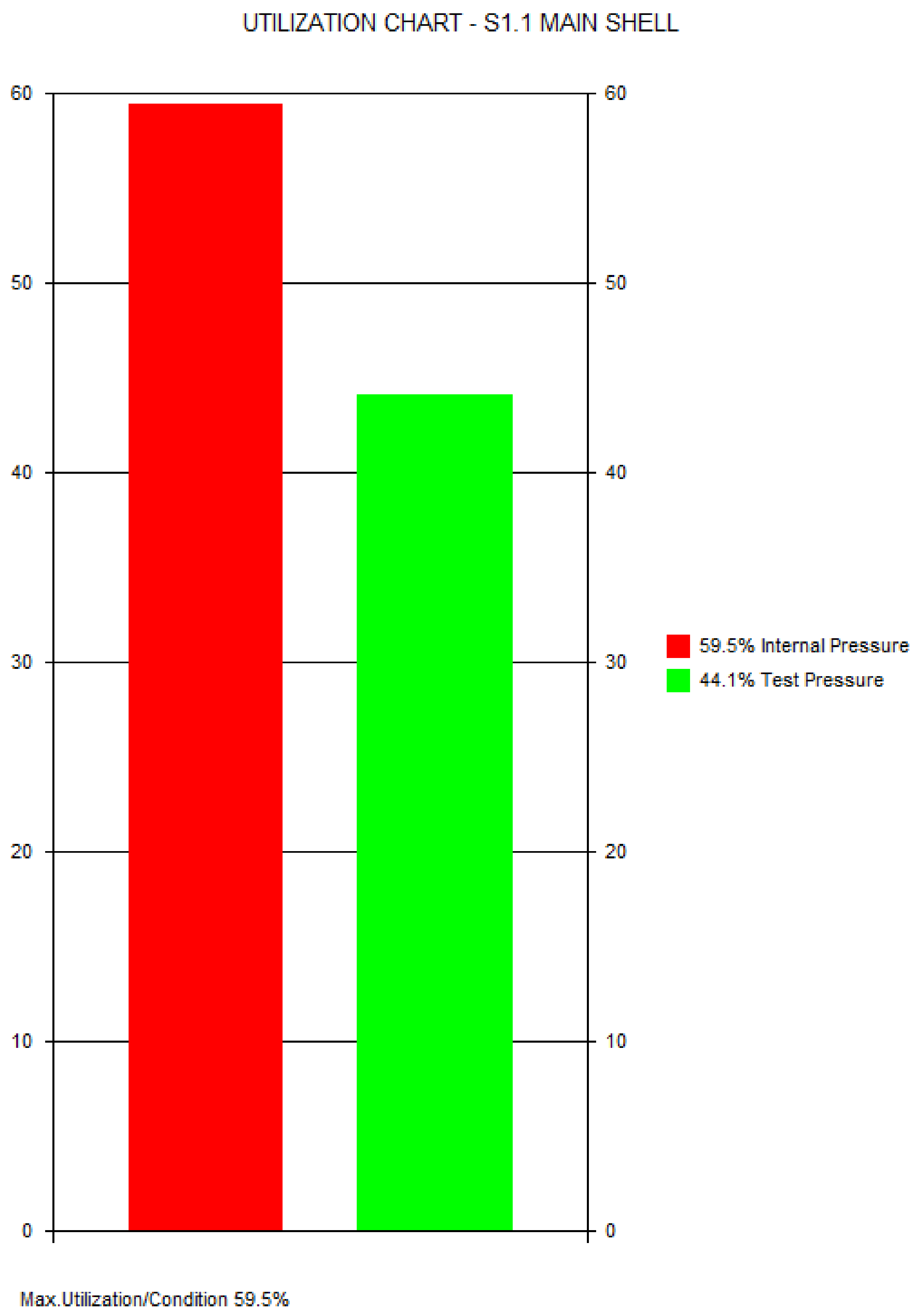
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)
Ptmax = SYtest * Etest * (ta + c) / (R + 0.6 * (ta + c))
=234*1*(33+1)/(250+0.6*(33+1))=                              29.42 MPa
```

»Test Pressure Ptmin=13 <= Ptmax=29.42[MPa] « » (U= 44.1%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

Volume:0.2 m3 Weight:458.3 kg (SG= 7.85)


UTILIZATION CHART - S1.1 MAIN SHELL
60
50
40
30
20
10
0
60
50
40
30
20
10
0
🟥 59.5% Internal Pressure
🟩 44.1% Test Pressure
Max.Utilization/Condition 59.5%

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
Location: Along z-axis zo=  100
```

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=10.000 MPa, c= 1 mm, Pext=0.000 MPa
```

DIMENSIONS OF END

```
Design Diameter: Base Design on Inside Diameter
Type of Ellipsoidal End: Semi-Ellipsoidal Head R:h 2:1
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        498.00 mm
LENGTH OF CYLINDRICAL PART OF END...................:Lcyl        0.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
AS BUILT THICKNESS OF HEAD/END (uncorroded).........:tn         35.00 mm
```

MATERIAL DATA FOR END

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138.57 Sr=138.57 Stest=234  (N/mm2)
MODULUS OF ELASTICITY at design temp................:E      1,9993E05 N/mm2
```

CALCULATION DATA

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Factor K from Appendix 1 Article 1-4(c)
K = 1 =1=                                                       1.00
```

Application of Rules for Ellipsoidal Heads:

»Geometry Check Di/2h=2 >= 1.0=1« » OK«

»Geometry Check Di/2h=2 <= 3.0=3« » OK«

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                           (APP.1-4(c))
=10*498*1/(2*138.57*1-0.2*10)=                             18.10 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =18.1+1+1=                             20.10 mm
```

»Internal Pressure tmina=20.1 <= tn=35[mm] « » (U= 57.4%) OK«

```
Analysis Thickness
ta = tn - c - th =35-1-1=                                    33.00 mm
Outside Diameter of Shell
Do = Di + 2 * (tn - c) =498+2*(35-1)=                       566.00 mm
Mean Diameter of Shell
Dm = (Do + Di) / 2 =(566+498)/2=                            532.00 mm
h = Di / 4 + (tn - c) =498/4+(35-1)=                        158.50 mm
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*34/(1*498+0.2*34)=                             18.67 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*33/(1*498+0.2*33)=                             18.12 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*34/(1*498+0.2*34)=                                  31.52 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*10*138.57/138.57=              13.00 MPa
```

»Test Pressure Ptmin=13 <= Ptmax=31.52[MPa] « » (U= 41.2%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 75 * tn / Rf * (1 - Rf / INFINITY)
=75*35/87.615*(1-87.615/Infinity)=                           29.96 %
```

CALCULATION SUMMARY

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                         (APP.1-4(c))
=10*498*1/(2*138.57*1-0.2*10)=                               18.10 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =18.1+1+1=                               20.10 mm
```

»Internal Pressure tmina=20.1 <= tn=35[mm] « » (U= 57.4%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*34/(1*498+0.2*34)=                               18.67 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*33/(1*498+0.2*33)=                               18.12 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*34/(1*498+0.2*34)=                                  31.52 MPa
```

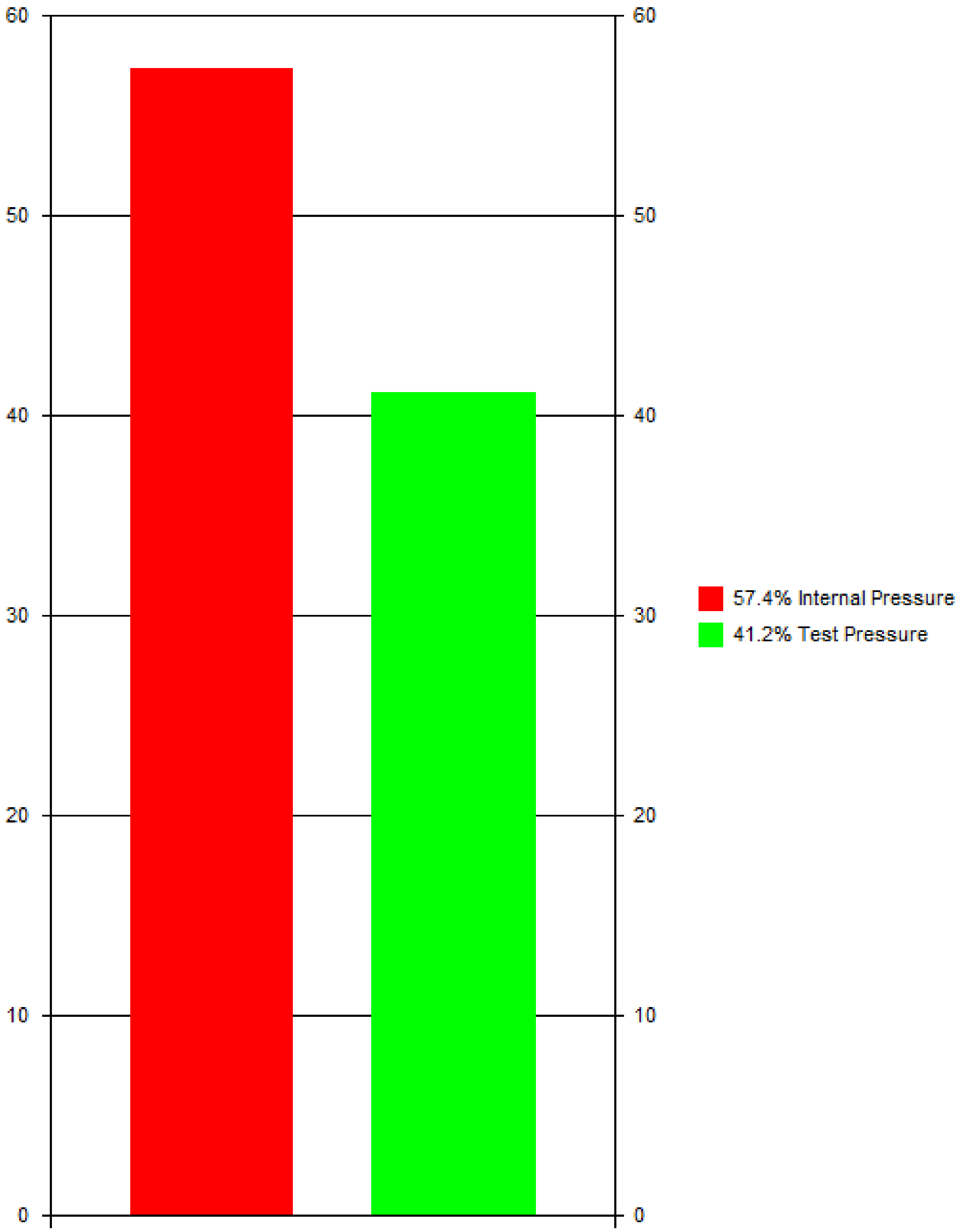
»Test Pressure Ptmin=13 <= Ptmax=31.52[MPa] « » (U= 41.2%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

Volume:0.02 m3 Weight:83.8 kg (SG= 7.85)

ASME VIII Div.1:2007 A09 - UG-32 ELLIPSOIDAL HEADS
E2.1 End cap 17 Apr. 2012 17:19 ConnID:S1.1

UTILIZATION CHART - E2.1 END CAP



Max.Utilization/Condition 57.4%

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
Location: Along z-axis z1=  1100
```

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=10.000 MPa, c= 1 mm, Pext=0.000 MPa
```

DIMENSIONS OF END

```
Design Diameter: Base Design on Inside Diameter
Type of Ellipsoidal End: Semi-Ellipsoidal Head R:h 2:1
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        498.00 mm
LENGTH OF CYLINDRICAL PART OF END...................:Lcyl        0.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
AS BUILT THICKNESS OF HEAD/END (uncorroded).........:tn         35.00 mm
```

MATERIAL DATA FOR END

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138.57 Sr=138.57 Stest=234  (N/mm2)
MODULUS OF ELASTICITY at design temp................:E      1,9993E05 N/mm2
```

CALCULATION DATA

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Factor K from Appendix 1 Article 1-4(c)
K = 1 =1=                                                       1.00
```

Application of Rules for Ellipsoidal Heads:

»Geometry Check Di/2h=2 >= 1.0=1« » OK«

»Geometry Check Di/2h=2 <= 3.0=3« » OK«

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                           (APP.1-4(c))
=10*498*1/(2*138.57*1-0.2*10)=                                 18.10 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =18.1+1+1=                                 20.10 mm
```

»Internal Pressure tmina=20.1 <= tn=35[mm] « » (U= 57.4%) OK«

```
Analysis Thickness
ta = tn - c - th =35-1-1=                                        33.00 mm
Outside Diameter of Shell
Do = Di + 2 * (tn - c) =498+2*(35-1)=                           566.00 mm
Mean Diameter of Shell
Dm = (Do + Di) / 2 =(566+498)/2=                                532.00 mm
h = Di / 4 + (tn - c) =498/4+(35-1)=                            158.50 mm
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*34/(1*498+0.2*34)=                                 18.67 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*33/(1*498+0.2*33)=                                 18.12 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*34/(1*498+0.2*34)=                            31.52 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*10*138.57/138.57=         13.00 MPa
```

»Test Pressure Ptmin=13 <= Ptmax=31.52[MPa] « » (U= 41.2%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 75 * tn / Rf * (1 - Rf / INFINITY)
=75*35/87.615*(1-87.615/Infinity)=                     29.96 %
```

CALCULATION SUMMARY

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                     (APP.1-4(c))
=10*498*1/(2*138.57*1-0.2*10)=                         18.10 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =18.1+1+1=                         20.10 mm
```

»Internal Pressure tmina=20.1 <= tn=35[mm] « » (U= 57.4%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*34/(1*498+0.2*34)=                         18.67 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138.57*1*33/(1*498+0.2*33)=                         18.12 MPa
```

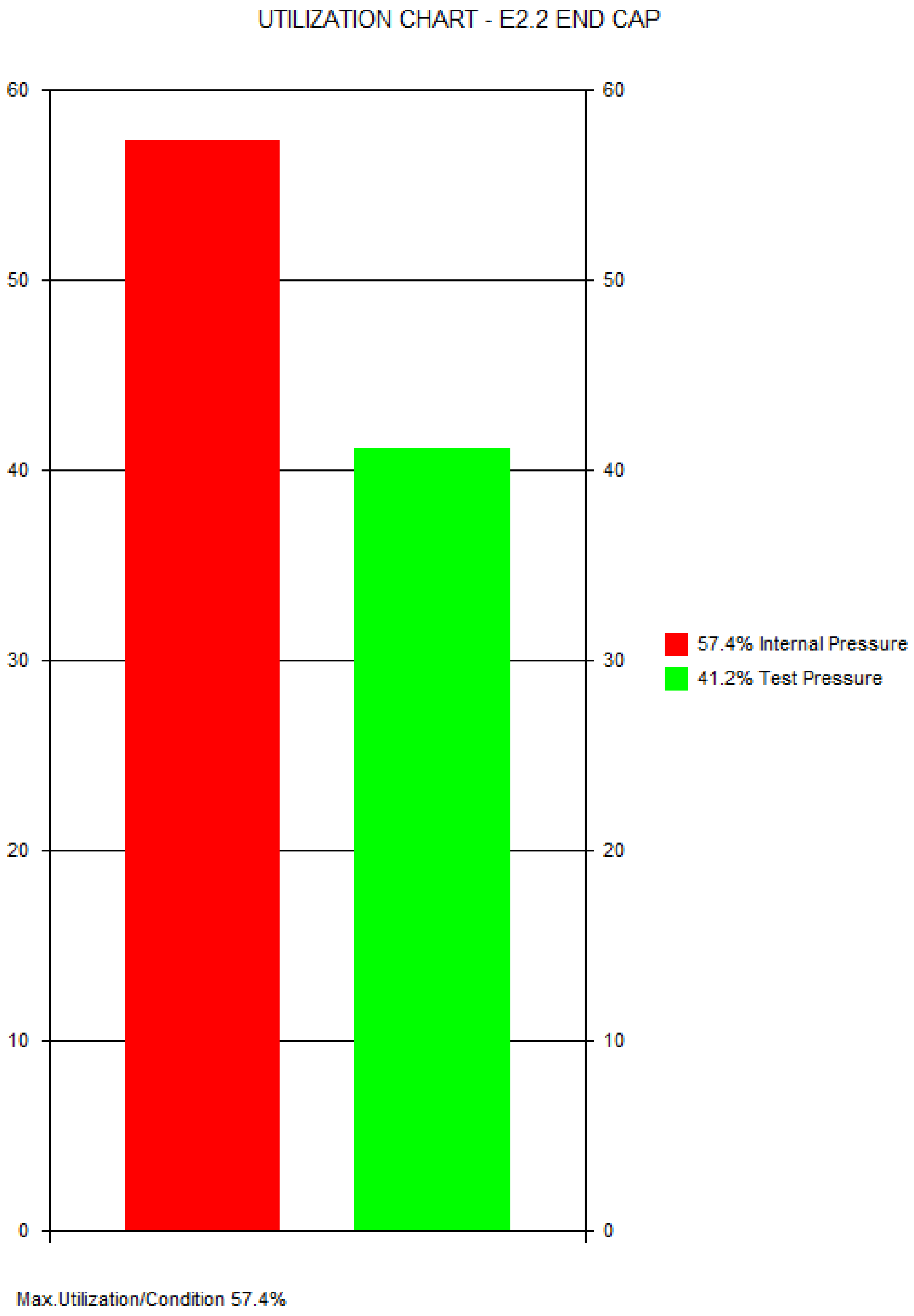
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*34/(1*498+0.2*34)=                            31.52 MPa
```

»Test Pressure Ptmin=13 <= Ptmax=31.52[MPa] « » (U= 41.2%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

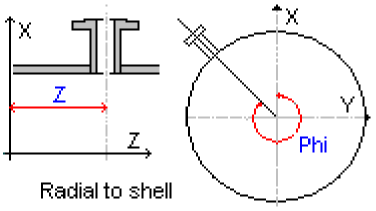
Volume:0.02 m3 Weight:83.8 kg (SG= 7.85)


UTILIZATION CHART - E2.2 END CAP
60
50
40
30
20
10
0
🟥 57.4% Internal Pressure
🟩 41.2% Test Pressure
Max.Utilization/Condition 57.4%

INPUT DATA

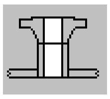
COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
```

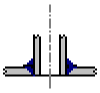


```
Orientation & Location of Nozzle: Radial to Shell
z-location of nozzle along axis of attacment........:z         600.00 mm
Angle of Rotation of nozzle axis projected in the x-y plane:Phi   360.00 Degr.
```

GENERAL DESIGN DATA



```
Type of Opening: Standard ANSI or DIN/EN Flange Attachment
```



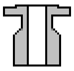
```
Nozzle Type: Set In Flush Nozzle
Nozzle Weld Intersect: Nozzle Does NOT Intersect with a Welded Shell Seam
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=10.000 MPa, c= 1 mm, Pext=0.000 MPa
Include Nozzle Load Calculation: YES
```

SHELL DATA (S1.1)

```
Shell Type: Cylindrical Shell
OUTSIDE DIAMETER OF SHELL............................:Do        566.00 mm
AS BUILT WALL THICKNESS (uncorroded)................:tn         35.00 mm
WELD JOINT EFFICIENCY FACTOR.........................:E1          1.00
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sv=138.57 Sr=138.57 Stest=234  (N/mm2)
```

NOZZLE DATA

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sn=138.57 Sr=138.57 Stest=234  (N/mm2)
Nozzle without pipe connections(access/inspection openings): NO
```



```
Delivery Form: Forging (LWN)
Base calculations on Forging OD: NO
INSIDE DIAMETER OF NOZZLE (corroded)................:d          78.19 mm
AS BUILT NOZZLE THICKNESS (uncorroded)..............:tnb        25.39 mm
Size of Flange and Nozzle: 3"
Comment (Optional): CLASS :900#    LWN Long Welding Neck
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:            1.00 mm
NOZZLE STANDOUT MEASURED FROM VESSEL OD.............:ho        200.00 mm
```

FLANGE DATA

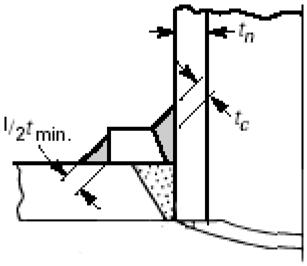
A: Flange Standard: ANSI B16.5 Flanges
E: Pressure Class: ANSI B16.5:Class 900 lbs
C: Flange Type: WN Welding Neck
D: Facing Sketch/ANSI facing (Table 3.8.3(2)): 1a RF Raised Face
Flange Material Category:
1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)

DATA FOR REINFORCEMENT PAD

Type of Pad: Single Pad
THICKNESS OF THE REINFORCEMENT PAD..................:te 25.00 mm
WIDTH OF THE REINFORCEMENT PAD......................:Lp 50.00 mm
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sp=138.57 Sr=138.57 Stest=234 (N/mm2)

WELDING DATA

Nozzle to Shell Welding Area: Include Area of Nozzle to Shell Weld as Min.Required



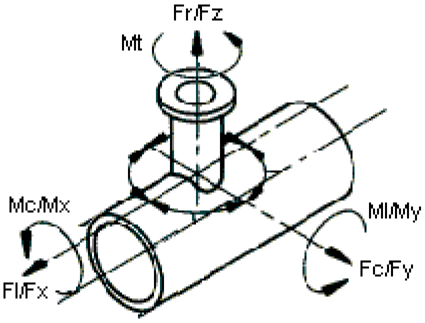
Weld Connection:
Full Penetration Weld + Outward Fillet Weld (to=tc) + PAD Fillet Weld (tp=0.5*tmin)

LIMITS OF REINFORCEMENT

Reduction of Limits of Reinforcement: No Reduction Required

EXTERNAL LOADS ON NOZZLE

FACTOR C4:
C4 = 1.1 Nozzle is Attached to a Piping System with due Allowance for Expansion and Thrust



TYPE OF LOAD INPUT: Load Cases
External Nozzle Loads: User Specified Loads

LOADING DATA

Table NOZZLE LOADS:

Load Description	*ID*	*Units*	*Load Case 1*
Pressure	*P*	*MPa*	*10*
Radial Load	*Fz*	*kN*	*30*

ASME VIII Div.1:2007 A09 - UG-37 REINFORCEMENT REQUIRED FOR OPENINGS IN SHELLS
N.1 Nozzle 17 Apr. 2012 17:19 ConnID:S1.1

Load Description	*ID*	*Units*	*Load Case 1*
Longitudinal Moment	*My*	*kNm*	
Circumferential Moment:	*Mx*	*kNm*	
Longitudinal Shear Force	*Fl*	*kN*	
Circumferential Shear Force	*Fc*	*kN*	
Torsional Moment	*Mt*	*kNm*	

CALCULATION DATA

FLANGE RATING

```
ANSI 900lb-Flange Rating(at 100C)= 13.915 MPa, Max.Test Pressure = 23.095 MPa
```

GEOMETRIC LIMITATIONS

Material Strength Reduction Factor fr1-4

```
Strength Reduction Factor for Nozzle Inserted Through Vessel Wall fr1
fr1 = MIN( Sn / Sv, 1) =MIN(138.57/138.57,1)=                   1.00
Strength Reduction Factor for Nozzle fr2
fr2 = MIN( Sn / Sv, 1) =MIN(138.57/138.57,1)=                   1.00
Strength Reduction Factor for Pad fr3
fr3 = MIN( MIN( Sn, Sp) / Sv, 1)
=MIN(MIN(138.57,138.57)/138.57,1)=                            1.00
Strength Reduction Factor for Pad fr4
fr4 = MIN( Sp / Sv, 1) =MIN(138.57/138.57,1)=                   1.00
```

PRELIMINARY CALCULATIONS

```
Shell Analysis Thickness t
t = tn - c - th =35-1-1=                                       33.00 mm
Nozzle Analysis Thickness tn
tn = tnb - c =25.39-1=                                         24.39 mm
Reinf.Pad Analysis Thickness te
te = MIN( te, 2.5 * t) =MIN(25,2.5*33)=                        25.00 mm
Inside Radius of Shell R
L = Do / 2 - t =566/2-33=                                     250.00 mm
Required Thickness of a Seamless Shell tr
tr = P * L / (Sv * E1 - 0.6 * P)
=10*250/(138.57*1-0.6*10)=                                   18.86 mm

deb = d + 2 * tn =78.19+2*24.39=                              126.97 mm
deb = d + 2 * tn =78.19+2*24.39=                              126.97 mm
Inside Radius of Nozzle Rn
Rn = d / 2 =78.19/2=                                           39.10 mm
Minimum nozzle thickness due to pressure
trn = P * Rn / (Sn * E - 0.6 * P)
=10*39.095/(138.57*1-0.6*10)=                                 2.95 mm
```

UG-40 LIMITS OF REINFORCEMENT

```
Parallel to Vessel Wall (half diameter limit)
Lv = MAX( d , d / 2 + t + tn)
=MAX(78.19,78.19/2+33+24.39)=                                96.49 mm
Normal to Vessel Wall Outside
Lno = MIN( 2.5 * t , 2.5 * tn + te)
=MIN(2.5*33,2.5*24.39+25)=                                   82.50 mm
Effective Material Diameter Limit
deff = 2 * Lv =2*96.485=                                      192.97 mm
```

UG-37 Calculation of Stress Loaded Areas Effective as Reinforcement

Area Available in Shell A1

```
A1 = (deff-d)*(E1*t-F*tr)-2*tn*(E1*t-F*tr)*(1-fr1)
=(192.97-78.19)*(1*33-1*18.86)-2*24.39*(1*33-1*18.86)*(1-1)=  1623.22 mm2
```

Area Available in Nozzle Projecting Outward A2

```
A2 = 2 * (tn - trn) * fr2 * MIN( Lno, ho)
=2*(24.39-2.95)*1*MIN(82.5,200)=                       3537.76 mm2
```

Area Available in Welds A4

```
Area Available in Nozzle Outward Weld A41
A41 = Leg41 ^ 2 * fr3 =8.57^2*1=                          73.44 mm2
Area Available in Outer Weld A42
A42 = Leg42 ^ 2 * fr4 =13.57^2*1=                        184.14 mm2
A4 = A41 + A42 + A43 =73.44+184.14+0=                    257.59 mm2
```

Area Available in Reinforcement Pad A5

```
Limit of Reinforcement Along Pad
wp = Min( Lp , Lv - deb / 2) =Min(50,96.485-126.97/2)=    33.00 mm
te = Min( te, 2.5 * t) =Min(25,2.5*33)=                   25.00 mm
A5 = 2 * wp * te * fr4 =2*33*25*1=                      1650.00 mm2
```

Total Area Available Aavail

```
Aavail = A1 + A2 + A3 + A4 + A5
=1623.22+3537.76+0+257.59+1650=                        7068.58 mm2
```

UG-37(c) Total Area Required

```
Total Area Required Areq
Areq = d * tr * F + 2 * tn * tr * F * (1 - fr1)
=78.19*18.86*1+2*24.39*18.86*1*(1-1)=                  1474.50 mm2
```

»UG-37 Nozzle Reinforcement Aavail=7068.58 >= Areq=1474.5[mm2] «» (U= 20.8%) OK«

UG-41.1 WELD STRENGTH AND WELD LOADS (Sketch a or b)

```
NOTE: UW-15(b) Strength calculations for attachment welds are NOT required for
this detail.
```

MAXIMUM ALLOWABLE WORKING PRESSURE

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax (t,tn,Sv,Sn)(33,24.4,138.6,138.6) = ==             27.88 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax (t,tn,Sv,Sn)(34,25.4,138.6,138.6) = ==             29.00 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax (t,tn,Sv,Sn)(34,25.4,234,234) = ==                 49.00 MPa
```

UW-16(c) DIMENSIONS OF FILLET WELDS:

```
Throat dimension of fillet welds on nozzle:
- at outward nozzle weld at nozzle OD, tmin = lesser of 19.0, tn or t/te)= 19 mm
to(min)=MIN(6, 0.7*tmin)= 6 mm
- at pad OD, tmin = lesser of 19.0, t or te)= 19 mm
tp(min)=( .5*tmin)= 9.5 mm
Minimum length of legs:
- at outward nozzle weld at nozzle OD, Leg41(min) =  8.57 mm
- at pad OD, Leg42(min) = 13.57 mm
```

»UG-16(c) Outward Nozzle Fillet Weld, Leg Size Leg41=8.57 >= Leg41(min)=8.57[mm] «» OK«

»UG-16(c) Fillet Weld at Pad OD, Leg Size Leg42=13.57 >= Leg42(min)=13.57[mm] «» OK«

UG-45 NOZZLE NECK THICKNESS

```
UG-45(a) Required Thickness of a Seamless Nozzle Wall trn
UG45a = MAX( trn , textn) + c =MAX(2.95,0)+1=                3.95 mm
UG45b1 = MAX( tr , tmin16) + c =MAX(18.86,1.6)+1=           19.86 mm
UG45b2 = 0 =0=                                               0.00 mm
UG45b3 = MAX( UG45b1 , UG45b2) =MAX(19.86,0)=               19.86 mm
UG45b4(Stnd.wall thk.minus 12.5% neg.tolerance)+c= ==        6.77 mm
UG45b = MIN( UG45b3 , UG45b4) =MIN(19.86,6.77)=              6.77 mm
Minimum Thickness of Nozzle Neck to UG45
UG45 = MAX( UG45a , UG45b) =MAX(3.95,6.77)=                  6.77 mm
```

»UG-45 Min.Nozzle Neck Thk. UG45=6.77 <= tnb-tolerance=24.39[mm] «» (U= 27.7%) OK«

```
Type of Design Method:
16.5 - EN13445 -LOCAL LOADS ON NOZZLES IN CYLINDRICAL SHELLS
```

16.5 - EN13445 -LOCAL LOADS ON NOZZLES IN CYLINDRICAL SHELLS

PRELIMINARY CALCULATIONS

```
Shell Analysis Thickness eas
eas = tn - c - th =35-1-1=                                  33.00 mm
Nozzle Analysis Thickness eb
eb = tnb - c - NegDev =25.39-1-1=                           24.39 mm
Mean diameter of shell
D = Do - ea =566-33=                                       533.00 mm
Mean radius of shell
R = D / 2 =533/2=                                          266.50 mm
```

16.5.3 CONDITIONS OF APPLICABILITY

»a) ea/D=0.0619 >= 0.001« » OK«

»a) ea/D=0.0619 <= 0.1« » OK«

»b) LamdaC=0.5834 <= 10« » OK«

```
»c) Dist.to any other local load shall not be less than SQR(D*ec)=  175.8 mm
»d) Nozzle thickness shall be maintained over a distance of SQR(d*eb)=  50 mm
```

LOAD CASE NO: 1 - Load Case 1

```
Total Moment
MB = Sqr( Mx ^ 2 + My ^ 2) =Sqr(0^2+0^2)=                    0.00 kNm
```

STRESSES AT OUTER DIAMETER OF NOZZLE

```
Mean Diameter of Nozzle
d = deb - eb =126.97-24.39=                                102.58 mm
Combined Analysis Thickness
ec = ea + te * Min( fp / f 1)
=33+25*Min(138.57/138.57,1)=                              58.00 mm
LamdaC = d / Sqr( D * ec) =102.58/Sqr(533*58)=            0.5834
Ratio1 = eb / ec =24.39/58=                               0.4205
Ratio2 = D / ec =533/58=                                     9.19
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    1.810  C2 =    4.900  C3 =    6.183
```

16.5.5 MAXIMUM ALLOWABLE INDIVIDUAL LOADS

```
Permissible Pressure Pmax:
Pmax (from nozzle calculation) = Pmax =17.88=               17.88 MPa

Allowable Axial Load Fzmax:
Fzmax = f * ec ^ 2 * C1 =138.57*58^2*1.81=                 843.73 kN

Allowable Circumferential Moment Mxmax:
Mxmax = f * ec ^ 2 * d / 4 * C2
=138.57*58^2*102.58/4*4.9=                                58.58 kNm

Allowable Longitudinal Moment Mxmax:
Mymax = f * ec ^ 2 * d / 4 * C3
=138.57*58^2*102.58/4*6.18=                               73.92 kNm
```

SHEAR STRESS FORMULAES (PD5500 Section G.2.3.6.3)

```
Shear Stresses due to Longitudinal Shear Force, TauFl:
TauFl = 2 * Fl / (PI * deb * ec)
=2*0/(3.14*126.97*58)=                                      0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * Fc / (PI * deb * ec)
=2*0/(3.14*126.97*58)=                                      0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * Mt / (PI * deb ^ 2 * ec)
=2*0/(3.14*126.97^2*58)=                                    0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                            0.00 N/mm2
```

16.5.6 COMBINATIONS OF EXTERNAL LOADS AND INTERNAL PRESSURE

```
PhiP = P / Pmax =10/17.88=                                 0.5593

PhiZ = Fz / Fzmax =30/843.73=                              0.0356

PhiTau = Tau / (0.5 * f) =0/(0.5*138.57)=                    0.00

PhiB = Sqr(( Mx / Mxmax) ^ 2 + (My / Mymax) ^ 2)
=Sqr((0/58.58)^2+(0/73.92)^2)=                              0.00

MaxAll = MAX(Abs(PhiP/C4+PhiZ),Abs(PhiZ),Abs(PhiP/C4-0.2*PhiZ)
=MAX(Abs(0.5593/1.1+0.0356,)Abs(0.0356,)Abs(0.5593/1.1-0.2*0.0356)=  0.5440

PhiAll = Sqr( MaxAll ^ 2 + PhiB ^ 2 + PhiTau ^ 2)
=Sqr(0.544^2+0^2+0^2)=                                   0.5440
```

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT NOZZLE OD PhiP=0.5593 <= 1.0 =1« » (U= 55.9%) OK«

»PhiZ AT NOZZLE OD PhiZ=0.0356 <= 1.0=1« » (U= 3.5%) OK«

»PhiB AT NOZZLE OD PhiB=0 <= 1.0=1« » (U= 0%) OK«

»PhiTau AT NOZZLE OD PhiTau=0 <= 1.0=1« » (U= 0%) OK«

»PhiAll AT NOZZLE OD PhiAll=0.544 <= 1.0=1« » (U= 54.3%) OK«

STRESSES AT OUTER EDGE OF PAD

```
Diameter at Edge of Reinforcement Pad
d = deb + 2 * Lp =126.97+2*50=                              226.97 mm
Combined Analysis Thickness
ec = ea =33=                                                  33.00 mm
LamdaC = d / Sqr( D * ec) =226.97/Sqr(533*33)=                 1.71
Ratio1 = MAX( eb / ec , 0.5) =MAX(24.39/33,0.5)=           0.7391
Ratio2 = D / ec =533/33=                                      16.15
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    2.238  C2 =    5.003  C3 =   11.424
```

16.5.5 MAXIMUM ALLOWABLE INDIVIDUAL LOADS

```
Permissible Pressure Pmax:
Pmax (from nozzle calculation) = Pmax =17.88=                 17.88 MPa

Allowable Axial Load Fzmax:
Fzmax = f * ec ^ 2 * C1 =138.57*33^2*2.24=                  337.73 kN

Allowable Circumferential Moment Mxmax:
Mxmax = f * ec ^ 2 * d / 4 * C2
=138.57*33^2*226.97/4*5.=                                  42.84 kNm

Allowable Longitudinal Moment Mxmax:
Mymax = f * ec ^ 2 * d / 4 * C3
=138.57*33^2*226.97/4*11.42=                               97.82 kNm
```

SHEAR STRESS FORMULAES (PD5500 Section G.2.3.6.3)

```
Shear Stresses due to Longitudinal Shear Force, TauFl:
TauFl = 2 * Fl / (PI * deb * ec)
=2*0/(3.14*126.97*33)=                                  0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * Fc / (PI * deb * ec)
=2*0/(3.14*126.97*33)=                                  0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * Mt / (PI * deb ^ 2 * ec)
=2*0/(3.14*126.97^2*33)=                                0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                        0.00 N/mm2
```

16.5.6 COMBINATIONS OF EXTERNAL LOADS AND INTERNAL PRESSURE

```
PhiP = P / Pmax =10/17.88=                             0.5593

PhiZ = Fz / Fzmax =30/337.73=                          0.0888

PhiTau = Tau / (0.5 * f) =0/(0.5*138.57)=                0.00

PhiB = Sqr(( Mx / Mxmax) ^ 2 + (My / Mymax) ^ 2)
=Sqr((0/42.84)^2+(0/97.82)^2)=                          0.00

MaxAll = MAX(Abs(PhiP/C4+PhiZ),Abs(PhiZ),Abs(PhiP/C4-0.2*PhiZ)
=MAX(Abs(0.5593/1.1+0.0888,)Abs(0.0888,)Abs(0.5593/1.1-0.2*0.0888)=  0.5973

PhiAll = Sqr( MaxAll ^ 2 + PhiB ^ 2 + PhiTau ^ 2)
=Sqr(0.5973^2+0^2+0^2)=                               0.5973
```

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT EDGE OF PAD PhiP=0.5593 <= 1.0 =1« » (U= 55.9%) OK«
»PhiZ AT EDGE OF PAD PhiZ=0.0888 <= 1.0=1« » (U= 8.8%) OK«
»PhiB AT EDGE OF PAD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT EDGE OF PAD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT EDGE OF PAD PhiAll=0.5973 <= 1.0=1« » (U= 59.7%) OK«

16.5.7 STRESS RANGES AND THEIR COMBINATIONS

16.5.7.1 LOAD RANGES

```
DeltaP = Max( Pmax , 0) - Min( Pmin , 0)
=Max(10,0)-Min(0,0)=                                   10.00 MPa

DeltaFz = Max( Fzmax , 0) - Min( Fzmin , 0)
=Max(30,0)-Min(0,0)=                                   30.00 kN

DeltaMx = Max( Mxmax , 0) - Min( Mxmin , 0)
=Max(0,0)-Min(0,0)=                                     0.00 kNm

DeltaMy = Max( Mymax , 0) - Min( Mymin , 0)
=Max(0,0)-Min(0,0)=                                     0.00 kNm

DeltaFl = Max( Flmax , 0) - Min( Flmin , 0)
=Max(0,0)-Min(0,0)=                                     0.00 kN

DeltaFc = Max( Fcmax , 0) - Min( Fcmin , 0)
=Max(0,0)-Min(0,0)=                                     0.00 kN

DeltaFshear = Sqr( DeltaFl ^ 2 + DeltaFc ^ 2)
=Sqr(0^2+0^2)=                                          0.00 kN

DeltaMt = Max( Mtmax , 0) - Min( Mtmin , 0)
=Max(0,0)-Min(0,0)=                                     0.00 kNm
```

16.5.7.2 EQVIVALENT SHELL THICKNESS

```
eeq = ea+Min(te*Lp/Sqr(D*(ea+te)),te)*Min(fp/f1)
=33+Min(25*50/Sqr(533*(33+25,))25)*Min(138.57/138.57,1)=    40.11 mm
```

16.5.7.3 STRESSES

```
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    1.810  C2 =    4.900  C3 =    7.002
Tmp1 = Sqr( d * eb / (D * eeq))
=Sqr(78.19*24.39/(533*40.11))=                              0.3421
Tmp2 = (2+2*d/D*Tmp1+1.25*d/D*Sqr(D/eeq))/(1+eb/eeq*Tmp1)
=(2+2*78.19/533*0.3421+1.25*78.19/533*Sqr(533/40.11))/(1+24.39/40.11*0.3421
)=                                                            =     2.49
Stresses due to Pressure Range
SigP = DeltaP * D / (2 * eeq) * Tmp2                                (16.5-21)
=10*533/(2*40.11)*2.49=                                      165.48 N/mm2

Stresses due to Axial Load Range
SigFz = 2.25 / C1 * (DeltaFz / eeq ^ 2)                             (16.5-22)
=2.25/1.81*(30/40.11^2)=                                      23.18 N/mm2

Stresses due to Circumferential Moment Range
SigMx = 2.25 / C2 * (4 * DeltaMx / (eeq ^ 2 * d))                   (16.5-23)
=2.25/4.9*(4*0/(40.11^2*78.19))=                               0.00 N/mm2

Stresses due to Longitudinal Moment Range
SigMy = 2.25 / C3 * (4 * DeltaMy / (eeq ^ 2 * d))                   (16.5-24)
=2.25/7.*(4*0/(40.11^2*78.19))=                                0.00 N/mm2

Shear Stresses due to Longitudinal Shear Force, DeltaFl:
TauFl = 2 * DeltaFl / (PI * deb * eeq)
=2*0/(3.14*126.97*40.11)=                                      0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * DeltaFc / (PI * deb * eeq)
=2*0/(3.14*126.97*40.11)=                                      0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * DeltaMt / (PI * deb ^ 2 * eeq)
=2*0/(3.14*126.97^2*40.11)=                                    0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                               0.00 N/mm2

Total Stress Intensity due to Load Range
SigTot = Abs(SigT+Sqr((SigP+SigFz)^2+SigMx^2+SigMy^2+4*Tau^2))     (16.5-25)
=Abs(0+Sqr((165.48+23.18)^2+0^2+0^2+4*0^2))=                 188.66 N/mm2
```

»Total Stress in Shell SigTot=188.66 <= 3*f=415.71[N/mm2] «» (U= 45.3%) OK«

16.5.8 NOZZLE LONGITUDINAL STRESSES

```
Maximum Longitudinal Stresses in Nozzle
SigLong = P*d/(4*eb)+4*MB/(PI*d^2*eb)+Fz/(PI*d*eb)
=10*78.19/(4*24.39)+4*0/(3.14*78.19^2*24.39)+30000/(3.14*78.19*24.39)
=    14.33 N/mm2
```

»Nozzle Long.Stress SigLong=14.33 <= fb=138.57[N/mm2] « » (U= 10.3%) OK«

16.14.6 COMPRESSIVE STRESS LIMITS

```
K = 1.21 * E * ea / (Sige * D)                                      (16.14-15)
=1.21*200000*24.39/(239*102.58)=                             240.75
alfa = 0.83 / Sqr( 1 + 0.005 * D / ea)                              (16.14-16)
=0.83/Sqr(1+0.005*102.58/24.39)=                            0.8214
delta = (1 - 0.4123 / (alfa * K) ^ 0.6) / S                         (16.14-19)
=(1-0.4123/(0.8214*240.75)^0.6)/1.5=                        0.6551
Maximum Allowable Compressive Stress
Sigcall = Sige * delta (16.14-20) =239*0.6551=                 156.58 N/mm2
```

16.14.4 PERMISSIBLE INDIVIDUAL LOADS

```
Maximum Tensile Force Ftmax
Ftmax = PI * D * ea * f (16.14-1) =3.14*102.58*24.39*138.57=  1089.16 kN

Maximum Compressive Force Fcmax
Fcmax = PI * D * ea * Sigcall                                  (16.14-2)
=3.14*102.58*24.39*156.58=                              1230.72 kN

Maximum Bending Moment Mmax
Mmax = PI / 4 * D ^ 2 * ea * Sigcall                           (16.14-3)
=3.14/4*102.58^2*24.39*156.58=                            31.56 kNm

Longitudinal Stability Check (P=0)
LongStab = MB / Mmax + Abs( Fzmin) / Fcmax
=0/31.56+Abs(0)/1230.72=                                   0.00
```

»Nozzle Long.Stability LongStab=0 <= 1.0=1« » (U= 0%) OK«

```
Weight of Nozzle: 9.6kg  Pad: 8kg  Flange: 15kg
```

CALCULATION SUMMARY

```
Aavail = A1 + A2 + A3 + A4 + A5
=1623.22+3537.76+0+257.59+1650=                         7068.58 mm2

Total Area Required Areq
Areq = d * tr * F + 2 * tn * tr * F * (1 - fr1)
=78.19*18.86*1+2*24.39*18.86*1*(1-1)=                   1474.50 mm2
```

»UG-37 Nozzle Reinforcement Aavail=7068.58 >= Areq=1474.5[mm2] «» (U= 20.8%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax (t,tn,Sv,Sn)(33,24.4,138.6,138.6) = ==               27.88 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax (t,tn,Sv,Sn)(34,25.4,138.6,138.6) = ==               29.00 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax (t,tn,Sv,Sn)(34,25.4,234,234) = ==                   49.00 MPa
```

»UG-45 Min.Nozzle Neck Thk. UG45=6.77 <= tnb-tolerance=24.39[mm] «» (U= 27.7%) OK«

LOAD CASE NO: 1 - Load Case 1

STRESSES AT OUTER DIAMETER OF NOZZLE

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT NOZZLE OD PhiP=0.5593 <= 1.0 =1« » (U= 55.9%) OK«
»PhiZ AT NOZZLE OD PhiZ=0.0356 <= 1.0=1« » (U= 3.5%) OK«
»PhiB AT NOZZLE OD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT NOZZLE OD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT NOZZLE OD PhiAll=0.544 <= 1.0=1« » (U= 54.3%) OK«

STRESSES AT OUTER EDGE OF PAD

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT EDGE OF PAD PhiP=0.5593 <= 1.0 =1« » (U= 55.9%) OK«

»PhiZ AT EDGE OF PAD PhiZ=0.0888 <= 1.0=1« » (U= 8.8%) OK«
»PhiB AT EDGE OF PAD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT EDGE OF PAD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT EDGE OF PAD PhiAll=0.5973 <= 1.0=1« » (U= 59.7%) OK«

16.5.7 STRESS RANGES AND THEIR COMBINATIONS

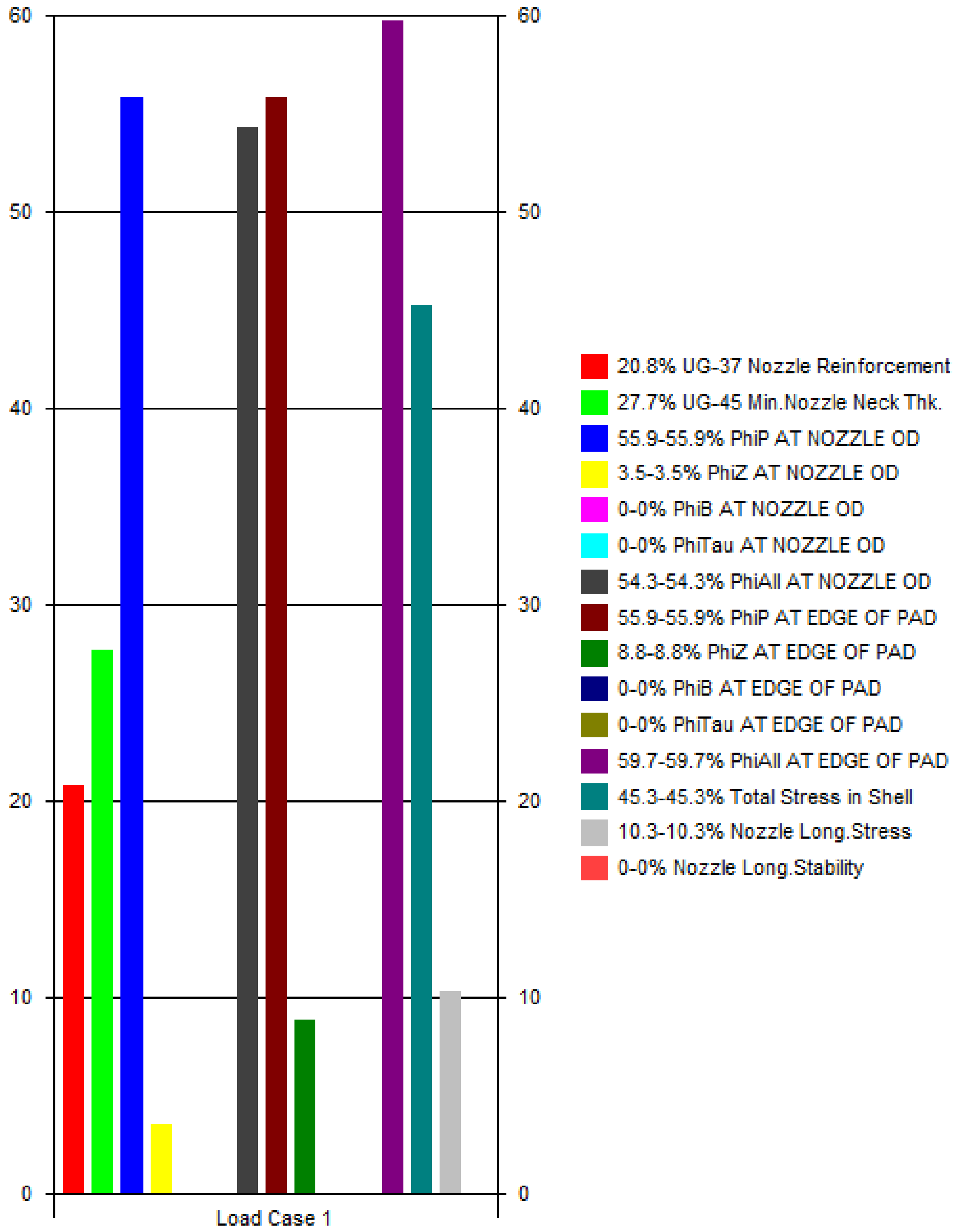
»Total Stress in Shell SigTot=188.66 <= 3*f=415.71[N/mm2] «» (U= 45.3%) OK«

16.5.8 NOZZLE LONGITUDINAL STRESSES

»Nozzle Long.Stress SigLong=14.33 <= fb=138.57[N/mm2] « » (U= 10.3%) OK«
»Nozzle Long.Stability LongStab=0 <= 1.0=1« » (U= 0%) OK«

Volume:0 m3 Weight:30.9 kg (SG= 7.85)

UTILIZATION CHART - N.1 NOZZLE

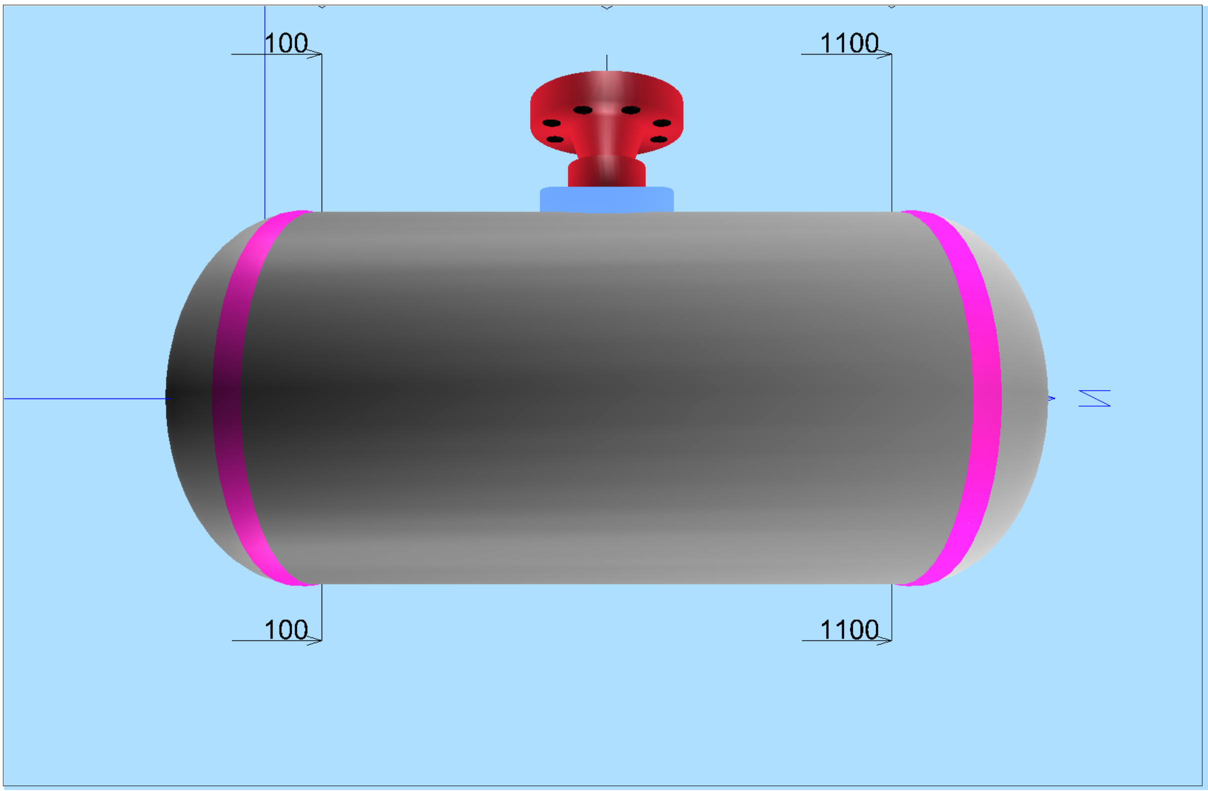


Max.Utilization/Condition 59.7% CASE:Load Case 1

APPENDIX C: Calculation report from VVD (70 mm @ 200 bar)

Drawing

3D View of Vessel (alter by using the Save User Specified View command)



History of Revisions

Rev	ID	Component Type	Comp. Description	DATE & TIME
A	E2.1	Ellipsoidal End	End cap	17 Apr. 2012 17:26
A	E2.2	Ellipsoidal End	End cap	17 Apr. 2012 17:26
A	N.1	Nozzle,Forging (LWN)	Nozzle	17 Apr. 2012 17:27
A	S1.1	Cylindrical Shell	Main Shell	17 Apr. 2012 17:26

A First Issue 28 Mar. 2012 12:58

Design Data & Process Information

Description	Units	Design Data
Process Card		General Design Data
Design Code & Specifications		ASME VIII Div.1:2007 A09
Internal Design Pressure (MPa)	MPa	20
External Design Pressure (MPa)	MPa	0
Hydrotest Pressure (MPa)	MPa	
Maximum Design Temperature ('C)	'C	100
Minimum Design Temperature ('C)	'C	20
Operating Temperature ('C)	'C	
Corrosion Allowance (mm)	mm	1
Content of Vessel		
Specific Density of Oper.Liq		1

Weight & Volume of Vessel

ID	No.	Wt-UnFinish.	Wt-Finished	Tot.Volume	Test.Liq.Wt	Oper.Liq.Wt
S1.1	1	978.0 kg	970.1 kg	0.196 m3	196.0 kg	96.6 kg
N.1	1	46.0 kg	46.0 kg	0.001 m3	1.0 kg	0.0 kg
E2.1	1	197.0 kg	197.0 kg	0.016 m3	16.0 kg	13.8 kg
E2.2	1	197.0 kg	197.0 kg	0.016 m3	16.0 kg	13.8 kg
Total	**4**	**1418.0 kg**	**1410.1 kg**	**0.229 m3**	**229.0 kg**	**124.3 kg**

Weight Summary/Condition	Weights
Empty Weight of Vessel incl. 5% Contingency	1481 kg
Total Test Weight of Vessel (Testing with Water)	1710 kg
Total Operating Weight of Vessel	1605 kg

Center of Gravity

ID	X-Empty	Y-Empty	Z-Empty	X-Test	Y-Test	Z-Test	X-Oper	Y-Oper	Z-Oper
S1.1	-2	0	600	0	0	600	-105	0	600
N.1	400	0	600	361	0	600	361	0	600
E2.1	0	0	38	0	0	54	-103	0	23
E2.2	0	0	1162	0	0	1146	-103	0	1177

CENTER OF GRAVITY AT CONDITIONS BELOW	X	Y	Z
Empty Vessel	11	0	600
Test Condition of Vessel (Testing with Water)	10	0	600
Operating Condition of Vessel	2	0	600

Max. Allowable Pressure MAWP

ID	Comp. Type	Liq.Head	MAWP New & Cold	MAWP Hot & Corr.
S1.1	Cylindrical Shell	0.000 MPa	32.789 MPa	32.270 MPa
N.1	Nozzle,Forging (LWN)	0.000 MPa	42.250 MPa	41.500 MPa
E2.1	Ellipsoidal End	0.000 MPa	37.210 MPa	36.685 MPa
E2.2	Ellipsoidal End	0.000 MPa	37.210 MPa	36.685 MPa
	MAWP		**32.789 MPa**	**32.270 MPa**

Note : Other components may limit the MAWP than the ones checked above.
Note : The value for MAWP is at top of vessel, with static liquid head subtracted.

Test Pressure

UG-99(b) REQUIRED MINIMUM TEST PRESSURE.

TEST PRESSURE OF VESSEL - NEW & COLD - HORIZONTAL

Design Pressure........................: 20.000 MPa
Design Temperature.....................: 100.0 C

ID	Description	Pdesign	PtMax	PtMin	Wat.Head	PtTop	PtTopMax
S1.1	Cylindrical Shell-Main Shell	20.000	55.408	26.000	0.009	25.991	55.400
N.1	3" ANSI B16.5 1500 lbs WN -RF Raised Face	20.000	38.435	NA	0.004	NA	38.431
N.1	Nozzle,Forging (LWN)-Nozzle	20.000	71.750	NA	0.004	NA	71.746
E2.1	Ellipsoidal End-End cap	20.000	63.095	26.000	0.006	25.994	63.089
E2.2	Ellipsoidal End-End cap	20.000	63.095	26.000	0.009	25.991	63.086

HYDRO-TEST

```
REQUIRED TEST PRESSURE AT TOP OF VESSEL PtReq(Hydro Test) ......:    26.000 MPa
MAXIMUM TEST PRESSURE AT TOP OF VESSEL PtLim(Hydro Test) .......:    38.431 MPa

Note : Other components may limit Ptlim than the ones checked above.
```

NOMENCLATURE:

Pdesign- is the design pressure including liquid head at the part under consideration.

PtMax - is the maximum allowed test pressure determined at the part under consideration.

PtMin - is the required test pressure determined at the part under consideration.

Wat.Head - is the water head during hydrotesting at the part under consideration.

PtBot - is the required test pressure at bottom of the vessel, for the part under consideration.

PtTop - is the required test pressure at top of the vessel, for the part under consideration.

PtTopMax - is the maximum test pressure allowed at top of the vessel, for the part under consideration.

PtReq - is the required minimum test pressure (largest value of PtTop) at top of vessel for the listed components.

PtLim - is the maximum allowed test pressure (minimum value for PtTopMax) at top of vessel for the listed components.

Bill of Materials

ID	No	Description	Component Dimensions	Material Standard
E2.1	1	Ellipsoidal End-End cap	Do= 636, t= 70, h= 193.5, Semi-Ellipsoidal Head R:h 2:1	ID 1, SA-516(M) Gr.70, K02700 Plate
E2.2	1	Ellipsoidal End-End cap	Do= 636, t= 70, h= 193.5, Semi-Ellipsoidal Head R:h 2:1	ID 1, SA-516(M) Gr.70, K02700 Plate
N.1	1	Flange:ANSI B16.5:Class 1500 lbs	WN Welding Neck, 1a RF Raised Face	1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)
N.1	1	Nozzle,Forging (LWN)-Nozzle	3" do=133.3,t=28.57,L=279.4,ho=200,PAD OD=235.3	ID 1, SA-516(M) Gr.70, K02700 Plate
N.1	1	Reinforcement Pad	PAD OD=235.3, wt= 40, width= 50	ID 1, SA-516(M) Gr.70, K02700 Plate
S1.1	1	Cylindrical Shell-Main Shell	Do= 636, t= 70, L= 1000	ID 1, SA-516(M) Gr.70, K02700 Plate

Notes, Warning & Error Messages

ID & Comp. Description	Notes/Warnings/Error Messages
S1.1 Cylindrical Shell Main Shell	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**
N.1 Nozzle,Forging (LWN) Nozzle	
-	**ERROR:a) ; ea/D <= 0.1 Outside Valid Range**
-	NOTE: Pad thickness is less than recommended minimum value of 75% of shell thickness.
E2.1 Ellipsoidal End End cap	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**
E2.2 Ellipsoidal End End cap	
-	**WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.**

TOTAL No. OF ERRORS/WARNINGS : 4

Nozzle List

ID	Service	SIZE	STANDARD/CLASS	ID	Standout	X	Y	Z	Rot.	Orient.
N.1	Nozzle	3"	ANSI B16.5 1500 lbs WN -RF Raised Face CLASS :1500 LWN Long Welding Neck	78.19	200	283	0	600	0	Radial

Nozzle Loads

ID	Load Desc.	Nozzle Loads
N.1	Nozzle	Fz=30kN,My=0,Mx=0,Mt=0kNm,Fl=0,Fc=0kN

Maximum Component Utilization - Umax

ID	Comp.Type	Umax(%)	Limited by
S1.1	Cylindrical Shell	59.5%	Internal Pressure
N.1	Nozzle,Forging (LWN)	48.1%	PhiP AT EDGE OF PAD
E2.1	Ellipsoidal End	55.1%	Internal Pressure
E2.2	Ellipsoidal End	55.1%	Internal Pressure

```
Component with highest utilization     Umax = 59.5% S1.1     Main Shell

Average utilization of all components Umean= 54.4%
```

Material Data/Mechanical Properties

ID	Material Name	Temp	ST	SY	SYd	S_d	Sr	ftest	E-mod	Note
1	SA-516(M) Gr.70, K02700 Plate , SG=7.85	100	485	260	239	138	138	234	199925	G10,S1,T2

```
Notation:
Thickness in mm, stress in N/mm2, temperature in deg.C
TG   : Test Group 1 to 4
Max.T: Maximum thickness for this stress set, 0 or 999 = No limit specified
S/C  : CS = Carbon Steel, SS = Stainless Steel
SG   : SG = Specific Gravity (Water = 1.0)
ST   : MIN.TENSILE STRENGTH at room temp.
SY   : MIN. YIELD STRENGTH at room temp.
SYd  : MIN. YIELD STRENGTH at calc.temp.
S_d  : DESIGN STRESS at calc.temp.
Sr   : DESIGN STRESS at room temp.
Note : G10 = Upon prolonged exposure to temperatures above 425°C, the carbide
phase of carbon steel may be converted to graphite.  See Appendix A, A-240.
Note : S1 = For Section I applications, stress values at temperatures of 450°C and
above are permissible but, except for tubular products 75 mm  O.D. or less
enclosed within the boiler setting, use of these materials at these temperatures
is not current practice.
Note : T2 = Allowable stresses for temperatures of 400°C and above are values
obtained from time-dependent properties.
```

Comp.Location in Global Coord.System

ID	Comp. Type	X	Y	Z	Teta	Phi	ConnID
E2.1	Ellipsoidal End	0	0	100	0.0	0.0	S1.1
E2.2	Ellipsoidal End	0	0	1100	0.0	0.0	S1.1
N.1	Nozzle,Forging (LWN)	283	0	600	90.0	360.0	S1.1
S1.1	Cylindrical Shell	0	0	100	0.0	0.0	

```
The report above shows the location of the connecting point (x, y and z) for
each component referenced to the coordinate system of the connecting component
```

(ConnID). The connecting point (x, y and z) is always on the center axis of rotational symmetry for the component under consideration, i.e. the connecting point for a nozzle connected to a cylindrical shell will be at the intersection of the nozzle center axis and the mid thickness of the shell referenced to the shell s coordinate system. In addition the orientation of the the center axis of the component is given by the two angles Teta and Phi, where Teta is the angle between the center axis of the two components and Phi is the orientation in the x-y plane
The basis for the coordinate system used by the software is a right handed coordinate system with the z-axis as the center axis of rotational geometry for the components, and Teta as the Polar Angle and Phi as the Azimuthal Angle

MDMT Minimum Design Metal Temperature

Table :

ID-Description	Material Name	tn(mm)	tg(mm)	Ratio	E(*)	Curve
E2.1 End cap - End	SA-516(M) Gr.70, K02700 Plate	70.0	70.0	0.55	1.00	D
E2.2 End cap - End	SA-516(M) Gr.70, K02700 Plate	70.0	70.0	0.55	1.00	D
N.1 Nozzle - Flange	1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)	0.0	0.0	3.09	1.00	NA
N.1 Nozzle - Nozzle	SA-516(M) Gr.70, K02700 Plate	28.6	28.6	0.22	1.00	D
N.1 Nozzle - Pad	SA-516(M) Gr.70, K02700 Plate	40.0	40.0	0.62	1.00	D
S1.1 Main Shell - Shell	SA-516(M) Gr.70, K02700 Plate	70.0	70.0	0.62	1.00	D

Table Continued

ID-Description	T1(C)	T2(C)	MDMT(C)	Comments
E2.1 End cap - End	-13.0	-26.4	-39.4	- Material Normalized
E2.2 End cap - End	-13.0	-26.4	-39.4	- Material Normalized
N.1 Nozzle - Flange	-28.8	0.0	-28.8	The ASME B16.5 nozzle flange has an unadjusted MDMT of -28.8 C
N.1 Nozzle - Nozzle	-32.0	-80.0	-112	- Material Normalized
N.1 Nozzle - Pad	-24.8	-21.8	-46.6	- Material Normalized
S1.1 Main Shell - Shell	-13.0	-21.8	-34.8	

MDMT CALCULATIONS PER UCS-66
MDMT Required : 20.0 C
MDMT Lowest Allowable: -28.8 C

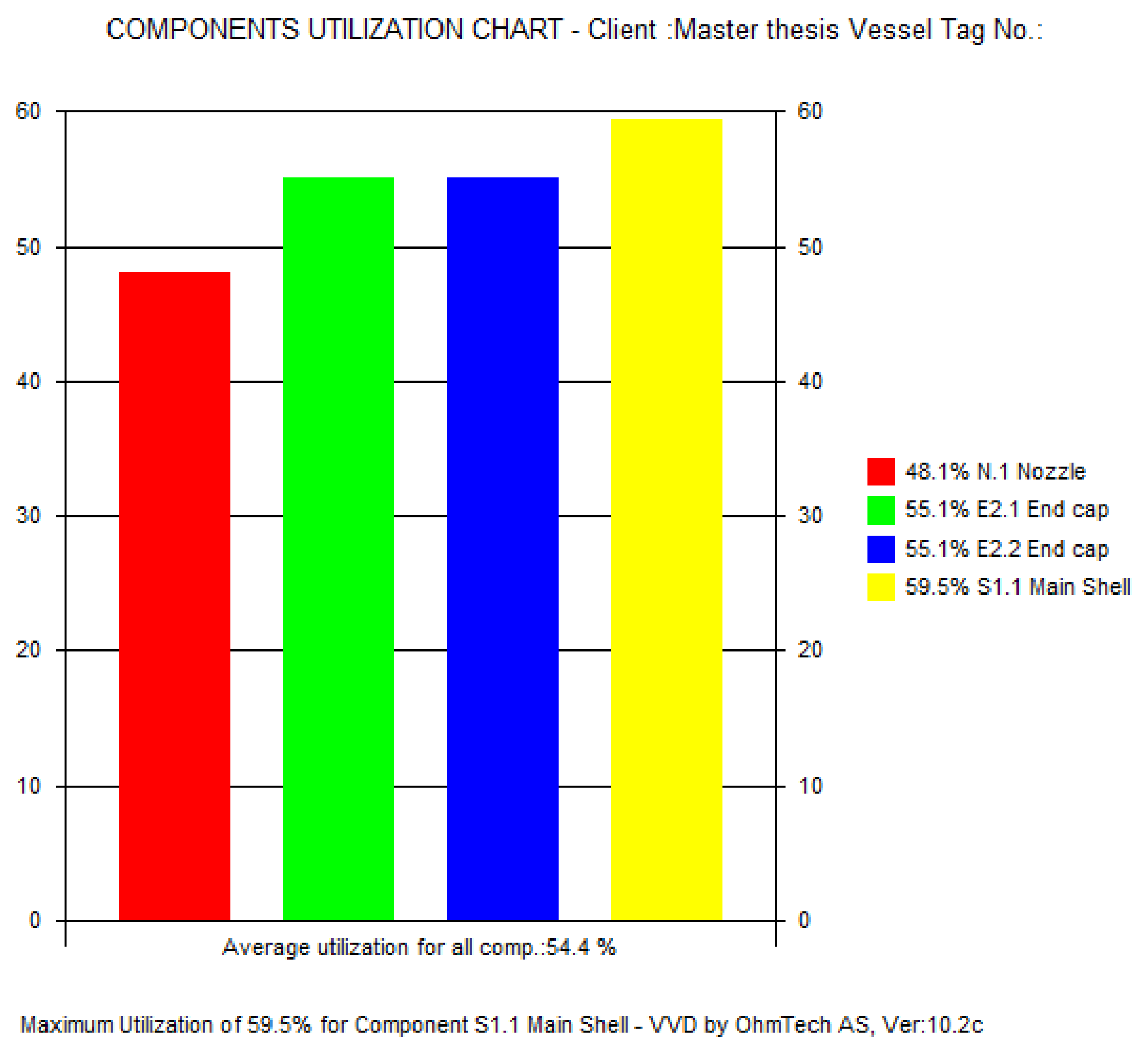
NOMENCLATURE:
tn - Nominal thickness of component under consideration(including corr. allow.).
tg - Governing thickness of component under consideration.
Ratio- tr*E(*)/(tn-c), utiliztion of component for given process conditions.
tr - Required minimum thickness of component at calculation temperature of MDMT.
E(*) - Joint efficiency factor, not lower than 0.8.
Curve- Applicable curve A, B, C or D in Figure UCS-66.
T1 - Unadjusted MDMT/Lowest allowable temperature for given part, value taken from Figure UCS-66 based on curve A, B, C or D.
T2 - Reduction in MDMT without impact testing per Figure UCS-66.1.

NOTES:
UCS-68(c) If postweld heat treatment is performed when it is not otherwise a requirement, a 17C reduction in impact test exemption temp. may be given to the min. permissible temp. for P.no.1 materials.
The maximum general primary stress in the pads are conservatively assumed to be the same as that in the corroded shell.

NOTE: LOWEST MDMT = -28.8 C (Warmest Value)

Utilization Chart

Utilization Chart



Surface Area

Table Surface Area:

ID	No.	Description	Area Outside(m2)	Area Inside(m2)
S1.1	1	Cylindrical Shell, Main Shell	1.998	1.571
N.1	1	Nozzle,Forging (LWN), Nozzle	0.084	0.048
E2.1	1	Ellipsoidal End, End cap	0.398	0.319
E2.2	1	Ellipsoidal End, End cap	0.398	0.319
Total	**4**		**2.878**	**2.257**

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=20.000 MPa, c= 1 mm, Pext=0.000 MPa
```

SHELL DATA

```
CYLINDER FABRICATION: Plate Material
DIAMETER INPUT: Base Design on Shell Inside Diameter
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138 Sr=138 Stest=234  (N/mm2)
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        500.00 mm
LENGTH OF CYLINDRICAL PART OF SHELL.................:Lcyl     1000.00 mm
AS BUILT WALL THICKNESS (uncorroded)................:tn         70.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
Split shell into several shell courses and include welding information: NO
```

CALCULATION DATA

UG-27 - CYLINDRICAL SHELLS UNDER INTERNAL PRESSURE

```
Inside Radius of Shell
R = Di / 2 =500/2=                                         250.00 mm
```

»Thin Cylinder Check P=20 <= 0.385 * S * E=53.13[MPa] « » OK«

```
Required Minimum Shell Thickness Excl.Allow. tmin :
tmin = P * R / (S * E - 0.6 * P)                                 (UG-27(1))
=20*250/(138*1-0.6*20)=                                   39.68 mm
```

»Thin Cylinder Check tmin=39.68 < 0.5 * R=125[mm] « » OK«

```
Required Minimum Shell Thickness Incl.Allow. :
tmina = tmin + c + th =39.68+1+1=                           41.68 mm

Analysis Thickness
ta = tn - c - th =70-1-1=                                   68.00 mm
```

»Internal Pressure tmina=41.68 <= tn=70[mm] « » (U= 59.5%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :

```
Outside Diameter of Shell
Do = Di + 2 * ta =500+2*68=                                636.00 mm
Outside Radius of Shell
Ro = Do / 2 =636/2=                                        318.00 mm
MAWP HOT & CORR. (Corroded condition at design temp.)
MAWPHC = S * E * ta / (R + 0.6 * ta)
=138*1*68/(250+0.6*68)=                                   32.27 MPa

MAWP NEW & COLD (Uncorroded condition at ambient temp.)
MAWPNC = Sr * E * (ta + c) / (R - c + 0.6 * (ta + c))
=138*1*(68+1)/(250-1+0.6*(68+1))=                         32.79 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)
Ptmax = SYtest * Etest * (ta + c) / (R + 0.6 * (ta + c))
=234*1*(68+1)/(250+0.6*(68+1))=                           55.41 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*20*138/138=                          26.00 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=55.41[MPa] « » (U= 46.9%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 50 * tn / Rf * (1 - Rf / INFINITY)
=50*70/283*(1-283/Infinity)=                                        12.37 %
```

CALCULATION SUMMARY

UG-27 - CYLINDRICAL SHELLS UNDER INTERNAL PRESSURE

```
Required Minimum Shell Thickness Excl.Allow. tmin :
tmin = P * R / (S * E - 0.6 * P)                                           (UG-27(1))
=20*250/(138*1-0.6*20)=                                             39.68 mm

Required Minimum Shell Thickness Incl.Allow. :
tmina = tmin + c + th =39.68+1+1=                                   41.68 mm
```

»Internal Pressure tmina=41.68 <= tn=70[mm] « » (U= 59.5%) OK«

```
MAWP HOT & CORR. (Corroded condition at design temp.)
MAWPHC = S * E * ta / (R + 0.6 * ta)
=138*1*68/(250+0.6*68)=                                             32.27 MPa

MAWP NEW & COLD (Uncorroded condition at ambient temp.)
MAWPNC = Sr * E * (ta + c) / (R - c + 0.6 * (ta + c))
=138*1*(68+1)/(250-1+0.6*(68+1))=                                   32.79 MPa
```

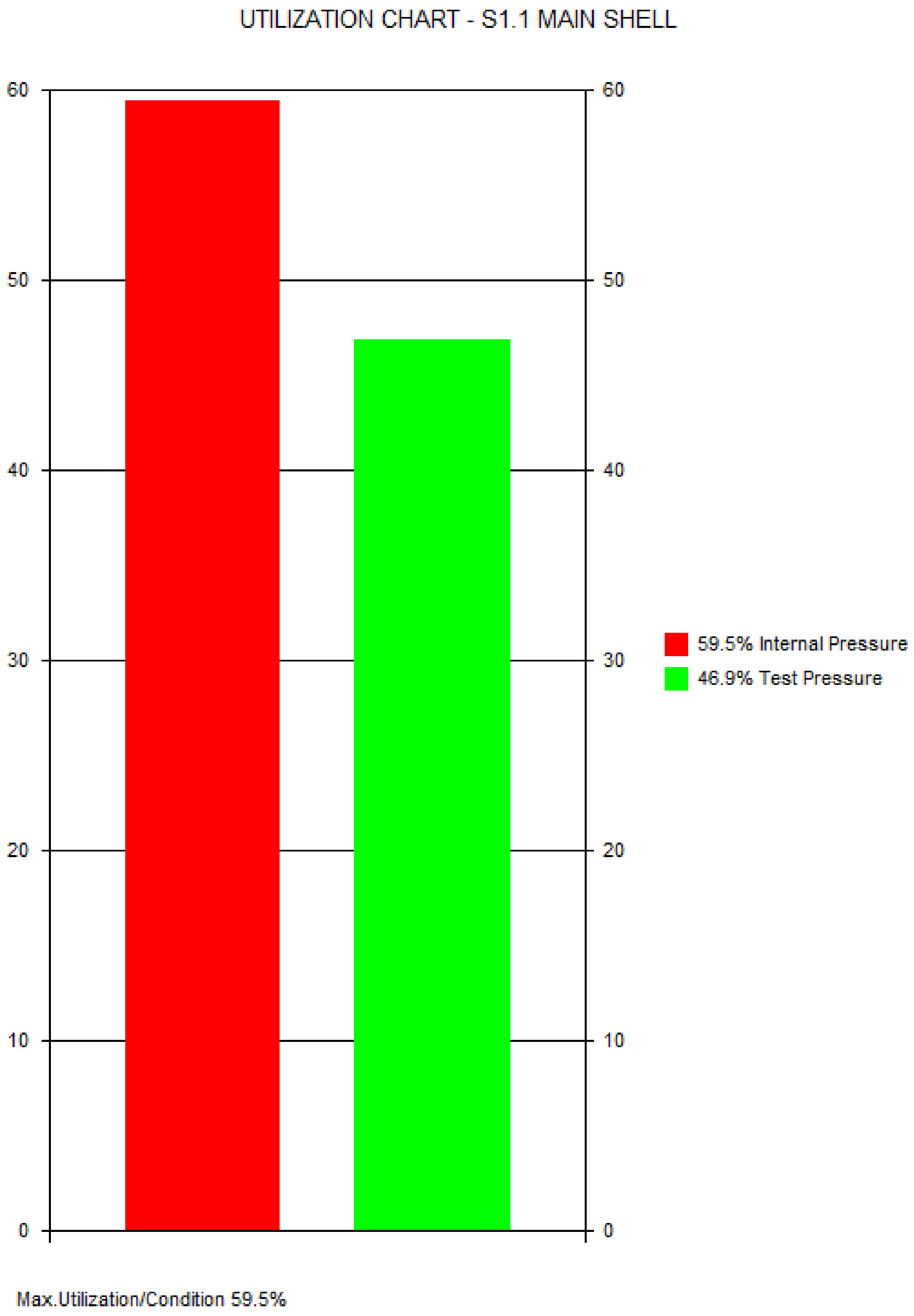
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)
Ptmax = SYtest * Etest * (ta + c) / (R + 0.6 * (ta + c))
=234*1*(68+1)/(250+0.6*(68+1))=                                     55.41 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=55.41[MPa] « » (U= 46.9%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

Volume:0.2 m3 Weight:977.1 kg (SG= 7.85)


UTILIZATION CHART - S1.1 MAIN SHELL
60
50
40
30
20
10
0
60
50
40
30
20
10
0
🟥 59.5% Internal Pressure
🟩 46.9% Test Pressure
Max.Utilization/Condition 59.5%

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
Location: Along z-axis zo=  100
```

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=20.000 MPa, c= 1 mm, Pext=0.000 MPa
```

DIMENSIONS OF END

```
Design Diameter: Base Design on Inside Diameter
Type of Ellipsoidal End: Semi-Ellipsoidal Head R:h 2:1
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        498.00 mm
LENGTH OF CYLINDRICAL PART OF END...................:Lcyl        0.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
AS BUILT THICKNESS OF HEAD/END (uncorroded).........:tn         70.00 mm
```

MATERIAL DATA FOR END

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138 Sr=138 Stest=234  (N/mm2)
MODULUS OF ELASTICITY at design temp................:E      1,9993E05 N/mm2
```

CALCULATION DATA

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Factor K from Appendix 1 Article 1-4(c)
K = 1 =1=                                                   1.00
```

Application of Rules for Ellipsoidal Heads:

»Geometry Check Di/2h=2 >= 1.0=1« » OK«

»Geometry Check Di/2h=2 <= 3.0=3« » OK«

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                       (APP.1-4(c))
=20*498*1/(2*138*1-0.2*20)=                              36.62 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =36.62+1+1=                          38.62 mm
```

»Internal Pressure tmina=38.62 <= tn=70[mm] « » (U= 55.1%) OK«

```
Analysis Thickness
ta = tn - c - th =70-1-1=                                  68.00 mm
Outside Diameter of Shell
Do = Di + 2 * (tn - c) =498+2*(70-1)=                     636.00 mm
Mean Diameter of Shell
Dm = (Do + Di) / 2 =(636+498)/2=                          567.00 mm
h = Di / 4 + (tn - c) =498/4+(70-1)=                      193.50 mm
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*69/(1*498+0.2*69)=                              37.21 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*68/(1*498+0.2*68)=                              36.68 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*69/(1*498+0.2*69)=                           63.09 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*20*138/138=              26.00 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=63.09[MPa] « » (U= 41.2%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 75 * tn / Rf * (1 - Rf / INFINITY)
=75*70/93.39*(1-93.39/Infinity)=                      56.22 %
```

CALCULATION SUMMARY

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                    (APP.1-4(c))
=20*498*1/(2*138*1-0.2*20)=                           36.62 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =36.62+1+1=                       38.62 mm
```

»Internal Pressure tmina=38.62 <= tn=70[mm] « » (U= 55.1%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*69/(1*498+0.2*69)=                           37.21 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*68/(1*498+0.2*68)=                           36.68 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

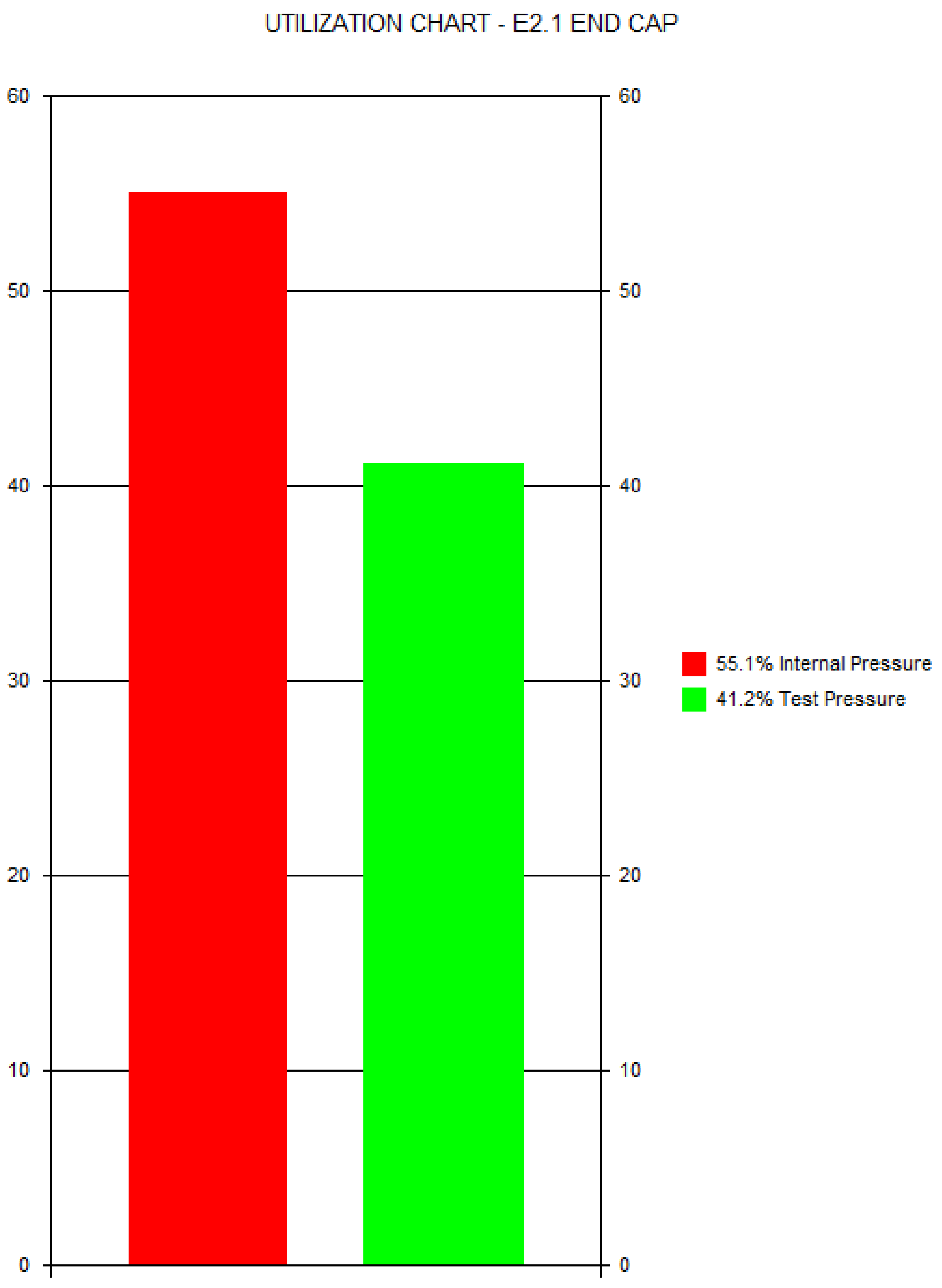
```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*69/(1*498+0.2*69)=                           63.09 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=63.09[MPa] « » (U= 41.2%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

Volume:0.02 m3 Weight:196.8 kg (SG= 7.85)

ASME VIII Div.1:2007 A09 - UG-32 ELLIPSOIDAL HEADS
E2.1 End cap 17 Apr. 2012 17:26 ConnID:S1.1



Max.Utilization/Condition 55.1%

INPUT DATA

COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
Location: Along z-axis z1=  1100
```

GENERAL DESIGN DATA

```
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=20.000 MPa, c= 1 mm, Pext=0.000 MPa
```

DIMENSIONS OF END

```
Design Diameter: Base Design on Inside Diameter
Type of Ellipsoidal End: Semi-Ellipsoidal Head R:h 2:1
WELD JOINT EFFICIENCY FACTOR: Full RT UW-11(a) Type 1 (E=1.0)
INSIDE SHELL DIAMETER (corroded)....................:Di        498.00 mm
LENGTH OF CYLINDRICAL PART OF END...................:Lcyl        0.00 mm
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th          1.00 mm
AS BUILT THICKNESS OF HEAD/END (uncorroded).........:tn         70.00 mm
```

MATERIAL DATA FOR END

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 S=138 Sr=138 Stest=234  (N/mm2)
MODULUS OF ELASTICITY at design temp................:E      1,9993E05 N/mm2
```

CALCULATION DATA

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Factor K from Appendix 1 Article 1-4(c)
K = 1 =1=                                                   1.00
```

Application of Rules for Ellipsoidal Heads:

»Geometry Check Di/2h=2 >= 1.0=1« » OK«

»Geometry Check Di/2h=2 <= 3.0=3« » OK«

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                        (APP.1-4(c))
=20*498*1/(2*138*1-0.2*20)=                              36.62 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =36.62+1+1=                          38.62 mm
```

»Internal Pressure tmina=38.62 <= tn=70[mm] « » (U= 55.1%) OK«

```
Analysis Thickness
ta = tn - c - th =70-1-1=                                  68.00 mm
Outside Diameter of Shell
Do = Di + 2 * (tn - c) =498+2*(70-1)=                     636.00 mm
Mean Diameter of Shell
Dm = (Do + Di) / 2 =(636+498)/2=                          567.00 mm
h = Di / 4 + (tn - c) =498/4+(70-1)=                      193.50 mm
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*69/(1*498+0.2*69)=                              37.21 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*68/(1*498+0.2*68)=                              36.68 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*69/(1*498+0.2*69)=                                   63.09 MPa
```

UG-99(b) REQUIRED MINIMUM TEST PRESSURE: NEW AT AMBIENT TEMP. Ptmin

```
Ptmin = 1.3 * Pd * Sr / S =1.3*20*138/138=                      26.00 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=63.09[MPa] « » (U= 41.2%) OK«

UCS-79 Extreme Fiber Elongation

```
fext = 75 * tn / Rf * (1 - Rf / INFINITY)
=75*70/93.39*(1-93.39/Infinity)=                              56.22 %
```

CALCULATION SUMMARY

UG-32(d) ELLIPSOIDAL HEADS UNDER INTERNAL PRESSURE

```
Required Minimum Head Thickness Excl.Allow. tmin :
tmin = P * Di * K / (2 * S * E - 0.2 * P)                           (APP.1-4(c))
=20*498*1/(2*138*1-0.2*20)=                                   36.62 mm

Required Minimum End Thickness Incl.Allow. :
tmina = tmin + c + th =36.62+1+1=                               38.62 mm
```

»Internal Pressure tmina=38.62 <= tn=70[mm] « » (U= 55.1%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*69/(1*498+0.2*69)=                                   37.21 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*138*1*68/(1*498+0.2*68)=                                   36.68 MPa
```

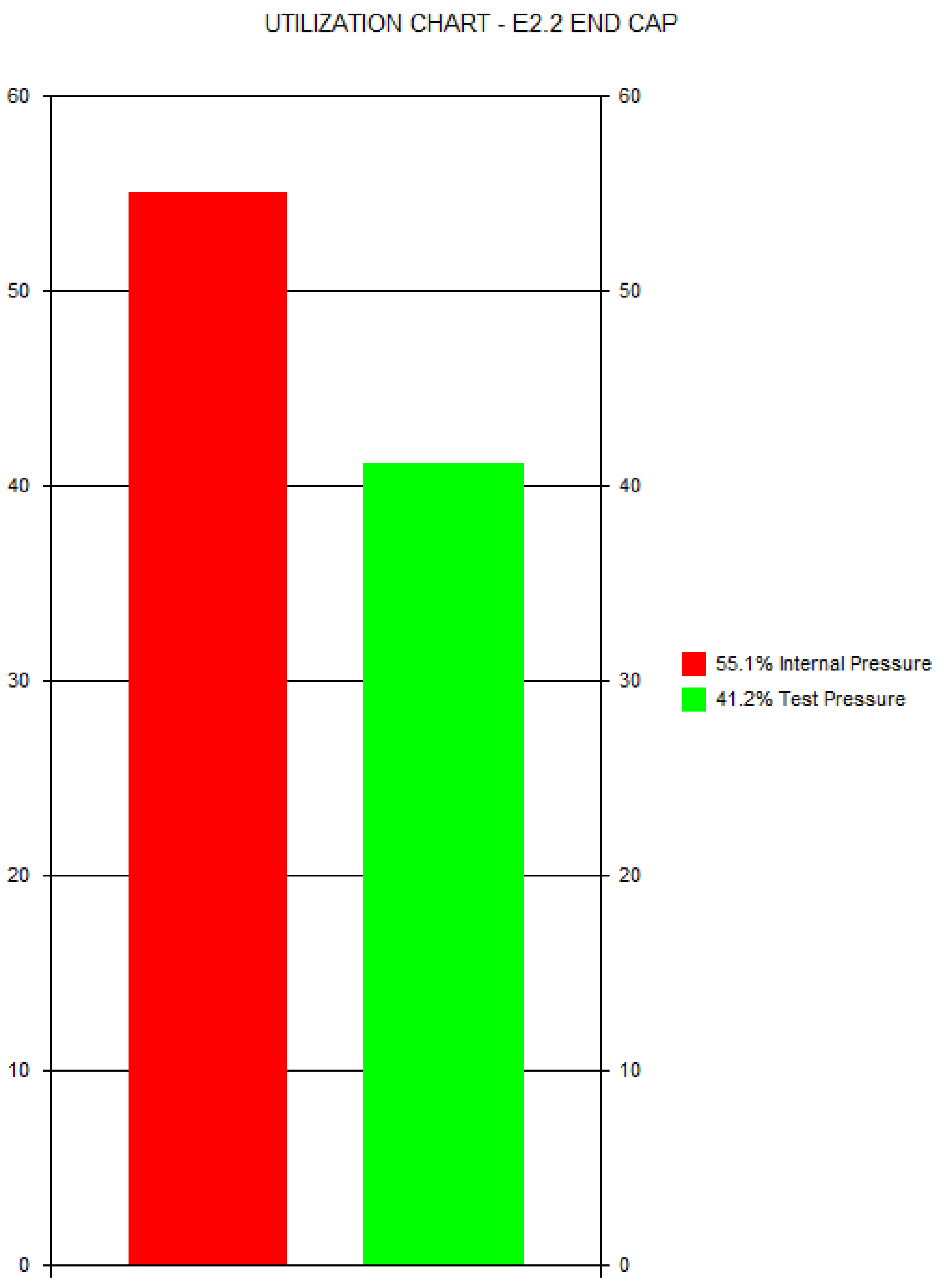
MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax = 2 * S * E * ta / (K * Di + 0.2 * ta)
=2*234*1*69/(1*498+0.2*69)=                                   63.09 MPa
```

»Test Pressure Ptmin=26 <= Ptmax=63.09[MPa] « » (U= 41.2%) OK«

WARNING: UCS-79 EXTREME FIBER ELONGATION EXCEEDS 5%, HEAT TREATMENT MAY BE REQUIRED.

Volume:0.02 m3 Weight:196.8 kg (SG= 7.85)

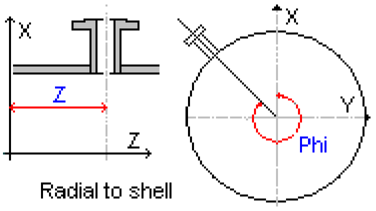


Max.Utilization/Condition 55.1%

INPUT DATA

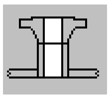
COMPONENT ATTACHMENT/LOCATION

```
Attachment: S1.1    Cylindrical Shell    Main Shell
```

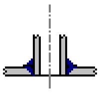


```
Orientation & Location of Nozzle: Radial to Shell
z-location of nozzle along axis of attacment........:z          600.00 mm
Angle of Rotation of nozzle axis projected in the x-y plane:Phi   360.00 Degr.
```

GENERAL DESIGN DATA



```
Type of Opening: Standard ANSI or DIN/EN Flange Attachment
```



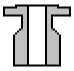
```
Nozzle Type: Set In Flush Nozzle
Nozzle Weld Intersect: Nozzle Does NOT Intersect with a Welded Shell Seam
PRESSURE LOADING: Design Component for Internal Pressure Only
PROCESS CARD:
General Design Data : Temp= 100°C, P=20.000 MPa, c= 1 mm, Pext=0.000 MPa
Include Nozzle Load Calculation: YES
```

SHELL DATA (S1.1)

```
Shell Type: Cylindrical Shell
OUTSIDE DIAMETER OF SHELL...........................:Do         636.00 mm
AS BUILT WALL THICKNESS (uncorroded)................:tn          70.00 mm
WELD JOINT EFFICIENCY FACTOR........................:E1           1.00
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:th           1.00 mm
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sv=138 Sr=138 Stest=234  (N/mm2)
```

NOZZLE DATA

```
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sn=138 Sr=138 Stest=234  (N/mm2)
Nozzle without pipe connections(access/inspection openings): NO
```



```
Delivery Form: Forging (LWN)
Base calculations on Forging OD: NO
INSIDE DIAMETER OF NOZZLE (corroded)................:d           78.19 mm
AS BUILT NOZZLE THICKNESS (uncorroded)..............:tnb         28.57 mm
Size of Flange and Nozzle: 3"
Comment (Optional): CLASS :1500#   LWN Long Welding Neck
NEGATIVE TOLERANCE/THINNING ALLOWANCE...............:            1.00 mm
NOZZLE STANDOUT MEASURED FROM VESSEL OD.............:ho         200.00 mm
```

FLANGE DATA

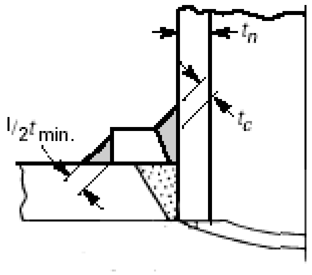
A: Flange Standard: ANSI B16.5 Flanges
E: Pressure Class: ANSI B16.5:Class 1500 lbs
C: Flange Type: WN Welding Neck
D: Facing Sketch/ANSI facing (Table 3.8.3(2)): 1a RF Raised Face
Flange Material Category:
1.1 - Carbon Steel - A105, A515 70, A516 70, A350 LF2 (BS 1503 164 490, BS 1504 161 480)

DATA FOR REINFORCEMENT PAD

Type of Pad: Single Pad
THICKNESS OF THE REINFORCEMENT PAD..................:te 40.00 mm
WIDTH OF THE REINFORCEMENT PAD......................:Lp 50.00 mm
SA-516(M) Gr.70, K02700 Plate THK<=0mm 100'C
ST=485 SY=260 SYd=239 Sp=138 Sr=138 Stest=234 (N/mm2)

WELDING DATA

Nozzle to Shell Welding Area: Include Area of Nozzle to Shell Weld as Min.Required



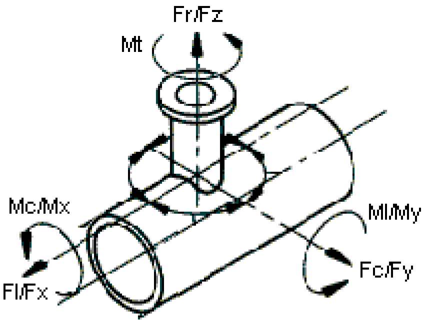
Weld Connection:
Full Penetration Weld + Outward Fillet Weld (to=tc) + PAD Fillet Weld (tp=0.5*tmin)

LIMITS OF REINFORCEMENT

Reduction of Limits of Reinforcement: No Reduction Required

EXTERNAL LOADS ON NOZZLE

FACTOR C4:
C4 = 1.1 Nozzle is Attached to a Piping System with due Allowance for Expansion and Thrust



TYPE OF LOAD INPUT: Load Cases
External Nozzle Loads: User Specified Loads

LOADING DATA

Table NOZZLE LOADS:

Load Description	*ID*	*Units*	*Load Case 1*
Pressure	*P*	*MPa*	*20*
Radial Load	*Fz*	*kN*	*30*

ASME VIII Div.1:2007 A09 - UG-37 REINFORCEMENT REQUIRED FOR OPENINGS IN SHELLS
N.1   Nozzle   17 Apr. 2012 17:27 ConnID:S1.1

Load Description	*ID*	*Units*	*Load Case 1*
Longitudinal Moment	*My*	*kNm*	
Circumferential Moment:	*Mx*	*kNm*	
Longitudinal Shear Force	*Fl*	*kN*	
Circumferential Shear Force	*Fc*	*kN*	
Torsional Moment	*Mt*	*kNm*	

CALCULATION DATA

FLANGE RATING

```
ANSI 1500lb-Flange Rating(at 100C)= 23.189 MPa, Max.Test Pressure = 38.435 MPa
```

GEOMETRIC LIMITATIONS

Material Strength Reduction Factor fr1-4

```
Strength Reduction Factor for Nozzle Inserted Through Vessel Wall fr1
fr1 = MIN( Sn / Sv, 1) =MIN(138/138,1)=                        1.00
Strength Reduction Factor for Nozzle fr2
fr2 = MIN( Sn / Sv, 1) =MIN(138/138,1)=                        1.00
Strength Reduction Factor for Pad fr3
fr3 = MIN( MIN( Sn, Sp) / Sv, 1)
=MIN(MIN(138,138)/138,1)=                                    1.00
Strength Reduction Factor for Pad fr4
fr4 = MIN( Sp / Sv, 1) =MIN(138/138,1)=                        1.00
```

PRELIMINARY CALCULATIONS

```
Shell Analysis Thickness t
t = tn - c - th =70-1-1=                                      68.00 mm
Nozzle Analysis Thickness tn
tn = tnb - c =28.57-1=                                        27.57 mm
Reinf.Pad Analysis Thickness te
te = MIN( te, 2.5 * t) =MIN(40,2.5*68)=                       40.00 mm
Inside Radius of Shell R
L = Do / 2 - t =636/2-68=                                   250.00 mm
Required Thickness of a Seamless Shell tr
tr = P * L / (Sv * E1 - 0.6 * P)
=20*250/(138*1-0.6*20)=                                    39.68 mm

deb = d + 2 * tn =78.19+2*27.57=                            133.33 mm
deb = d + 2 * tn =78.19+2*27.57=                            133.33 mm
Inside Radius of Nozzle Rn
Rn = d / 2 =78.19/2=                                          39.10 mm
Minimum nozzle thickness due to pressure
trn = P * Rn / (Sn * E - 0.6 * P)
=20*39.095/(138*1-0.6*20)=                                  6.21 mm
```

UG-40 LIMITS OF REINFORCEMENT

```
Parallel to Vessel Wall (half diameter limit)
Lv = MAX( d , d / 2 + t + tn)
=MAX(78.19,78.19/2+68+27.57)=                             134.67 mm
Normal to Vessel Wall Outside
Lno = MIN( 2.5 * t , 2.5 * tn + te)
=MIN(2.5*68,2.5*27.57+40)=                                108.92 mm
Effective Material Diameter Limit
deff = 2 * Lv =2*134.67=                                    269.33 mm
```

UG-37 Calculation of Stress Loaded Areas Effective as Reinforcement

Area Available in Shell A1

```
A1 = (deff-d)*(E1*t-F*tr)-2*tn*(E1*t-F*tr)*(1-fr1)
=(269.33-78.19)*(1*68-1*39.68)-2*27.57*(1*68-1*39.68)*(1-1)=  5412.60 mm2
```

Area Available in Nozzle Projecting Outward A2

```
A2 = 2 * (tn - trn) * fr2 * MIN( Lno, ho)
=2*(27.57-6.21)*1*MIN(108.92,200)=                         4654.24 mm2
```

Area Available in Welds A4

```
Area Available in Nozzle Outward Weld A41
A41 = Leg41 ^ 2 * fr3 =8.57^2*1=                              73.44 mm2
Area Available in Outer Weld A42
A42 = Leg42 ^ 2 * fr4 =13.57^2*1=                            184.14 mm2
A4 = A41 + A42 + A43 =73.44+184.14+0=                        257.59 mm2
```

Area Available in Reinforcement Pad A5

```
Limit of Reinforcement Along Pad
wp = Min( Lp , Lv - deb / 2) =Min(50,134.67-133.33/2)=        50.00 mm
te = Min( te, 2.5 * t) =Min(40,2.5*68)=                       40.00 mm
A5 = 2 * wp * te * fr4 =2*50*40*1=                          4000.00 mm2
```

Total Area Available Aavail

```
Aavail = A1 + A2 + A3 + A4 + A5
=5412.6+4654.24+0+257.59+4000=                            14324.43 mm2
```

UG-37(c) Total Area Required

```
Total Area Required Areq
Areq = d * tr * F + 2 * tn * tr * F * (1 - fr1)
=78.19*39.68*1+2*27.57*39.68*1*(1-1)=                      3102.78 mm2
```

»UG-37 Nozzle Reinforcement Aavail=14324.43 >= Areq=3102.78[mm2] «» (U= 21.6%) OK«

UG-41.1 WELD STRENGTH AND WELD LOADS (Sketch a or b)

```
NOTE: UW-15(b) Strength calculations for attachment welds are NOT required for
this detail.
```

MAXIMUM ALLOWABLE WORKING PRESSURE

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax (t,tn,Sv,Sn)(68,27.6,138,138) = ==                    41.50 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax (t,tn,Sv,Sn)(69,28.6,138,138) = ==                    42.25 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax (t,tn,Sv,Sn)(69,28.6,234,234) = ==                    71.75 MPa
```

UW-16(c) DIMENSIONS OF FILLET WELDS:

```
Throat dimension of fillet welds on nozzle:
- at outward nozzle weld at nozzle OD, tmin = lesser of 19.0, tn or t/te)= 19 mm
to(min)=MIN(6, 0.7*tmin)= 6 mm
- at pad OD, tmin = lesser of 19.0, t or te)= 19 mm
tp(min)=( .5*tmin)= 9.5 mm
Minimum length of legs:
- at outward nozzle weld at nozzle OD, Leg41(min) =  8.57 mm
- at pad OD, Leg42(min) = 13.57 mm
```

»UG-16(c) Outward Nozzle Fillet Weld, Leg Size Leg41=8.57 >= Leg41(min)=8.57[mm] «» OK«

»UG-16(c) Fillet Weld at Pad OD, Leg Size Leg42=13.57 >= Leg42(min)=13.57[mm] «» OK«

UG-45 NOZZLE NECK THICKNESS

```
UG-45(a) Required Thickness of a Seamless Nozzle Wall trn
UG45a = MAX( trn , textn) + c =MAX(6.21,0)+1=                  7.21 mm
UG45b1 = MAX( tr , tmin16) + c =MAX(39.68,1.6)+1=             40.68 mm
UG45b2 = 0 =0=                                                 0.00 mm
UG45b3 = MAX( UG45b1 , UG45b2) =MAX(40.68,0)=                 40.68 mm
UG45b4(Stnd.wall thk.minus 12.5% neg.tolerance)+c= ==          6.77 mm
UG45b = MIN( UG45b3 , UG45b4) =MIN(40.68,6.77)=                6.77 mm
Minimum Thickness of Nozzle Neck to UG45
UG45 = MAX( UG45a , UG45b) =MAX(7.21,6.77)=                    7.21 mm
```

»UG-45 Min.Nozzle Neck Thk. UG45=7.21 <= tnb-tolerance=27.57[mm] «» (U= 26.1%) OK«

```
Type of Design Method:
16.5 - EN13445 -LOCAL LOADS ON NOZZLES IN CYLINDRICAL SHELLS
```

16.5 - EN13445 -LOCAL LOADS ON NOZZLES IN CYLINDRICAL SHELLS

PRELIMINARY CALCULATIONS

```
Shell Analysis Thickness eas
eas = tn - c - th =70-1-1=                                    68.00 mm
Nozzle Analysis Thickness eb
eb = tnb - c - NegDev =28.57-1-1=                             27.57 mm
Mean diameter of shell
D = Do - ea =636-68=                                         568.00 mm
Mean radius of shell
R = D / 2 =568/2=                                            284.00 mm
```

16.5.3 CONDITIONS OF APPLICABILITY

»a) ea/D=0.1197 >= 0.001« » OK«

»a) ea/D=0.1197 <= 0.1=0.1« » NOT OK«

»b) LamdaC=0.427 <= 10« » OK«

```
»c) Dist.to any other local load shall not be less than SQR(D*ec)=  247.7 mm
»d) Nozzle thickness shall be maintained over a distance of SQR(d*eb)=  54 mm
```

LOAD CASE NO: 1 - Load Case 1

```
Total Moment
MB = Sqr( Mx ^ 2 + My ^ 2) =Sqr(0^2+0^2)=                      0.00 kNm
```

STRESSES AT OUTER DIAMETER OF NOZZLE

```
Mean Diameter of Nozzle
d = deb - eb =133.33-27.57=                                  105.76 mm
Combined Analysis Thickness
ec = ea + te * Min( fp / f 1) =68+40*Min(138/138,1)=         108.00 mm
LamdaC = d / Sqr( D * ec) =105.76/Sqr(568*108)=             0.4270
Ratio1 = eb / ec =27.57/108=                                0.2553
Ratio2 = D / ec =568/108=                                     5.26
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    1.810  C2 =    4.900  C3 =    5.188
```

16.5.5 MAXIMUM ALLOWABLE INDIVIDUAL LOADS

```
Permissible Pressure Pmax:
Pmax (from nozzle calculation) = Pmax =41.5=                    41.50 MPa

Allowable Axial Load Fzmax:
Fzmax = f * ec ^ 2 * C1 =138*108^2*1.81=                      2913.43 kN

Allowable Circumferential Moment Mxmax:
Mxmax = f * ec ^ 2 * d / 4 * C2
=138*108^2*105.76/4*4.9=                                     208.54 kNm

Allowable Longitudinal Moment Mxmax:
Mymax = f * ec ^ 2 * d / 4 * C3
=138*108^2*105.76/4*5.19=                                    220.78 kNm
```

SHEAR STRESS FORMULAES (PD5500 Section G.2.3.6.3)

```
Shear Stresses due to Longitudinal Shear Force, TauFl:
TauFl = 2 * Fl / (PI * deb * ec)
=2*0/(3.14*133.33*108)=                                      0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * Fc / (PI * deb * ec)
=2*0/(3.14*133.33*108)=                                      0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * Mt / (PI * deb ^ 2 * ec)
=2*0/(3.14*133.33^2*108)=                                    0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                             0.00 N/mm2
```

16.5.6 COMBINATIONS OF EXTERNAL LOADS AND INTERNAL PRESSURE

```
PhiP = P / Pmax =20/41.5=                                   0.4819

PhiZ = Fz / Fzmax =30/2913.43=                              0.0103

PhiTau = Tau / (0.5 * f) =0/(0.5*138)=                         0.00

PhiB = Sqr(( Mx / Mxmax) ^ 2 + (My / Mymax) ^ 2)
=Sqr((0/208.54)^2+(0/220.78)^2)=                             0.00

MaxAll = MAX(Abs(PhiP/C4+PhiZ),Abs(PhiZ),Abs(PhiP/C4-0.2*PhiZ)
=MAX(Abs(0.4819/1.1+0.0103,)Abs(0.0103,)Abs(0.4819/1.1-0.2*0.0103)=  0.4484

PhiAll = Sqr( MaxAll ^ 2 + PhiB ^ 2 + PhiTau ^ 2)
=Sqr(0.4484^2+0^2+0^2)=                                   0.4484
```

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT NOZZLE OD PhiP=0.4819 <= 1.0 =1« » (U= 48.1%) OK«
»PhiZ AT NOZZLE OD PhiZ=0.0103 <= 1.0=1« » (U= 1%) OK«
»PhiB AT NOZZLE OD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT NOZZLE OD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT NOZZLE OD PhiAll=0.4484 <= 1.0=1« » (U= 44.8%) OK«

STRESSES AT OUTER EDGE OF PAD

```
Diameter at Edge of Reinforcement Pad
d = deb + 2 * Lp =133.33+2*50=                               233.33 mm
Combined Analysis Thickness
ec = ea =68=                                                  68.00 mm
LamdaC = d / Sqr( D * ec) =233.33/Sqr(568*68)=                 1.19
Ratio1 = MAX( eb / ec , 0.5) =MAX(27.57/68,0.5)=            0.5000
Ratio2 = D / ec =568/68=                                       8.35
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    1.810  C2 =    4.900  C3 =    9.004
```

16.5.5 MAXIMUM ALLOWABLE INDIVIDUAL LOADS

```
Permissible Pressure Pmax:
Pmax (from nozzle calculation) = Pmax =41.5=                  41.50 MPa

Allowable Axial Load Fzmax:
Fzmax = f * ec ^ 2 * C1 =138*68^2*1.81=                     1154.98 kN

Allowable Circumferential Moment Mxmax:
Mxmax = f * ec ^ 2 * d / 4 * C2
=138*68^2*233.33/4*4.9=                                    182.39 kNm

Allowable Longitudinal Moment Mxmax:
Mymax = f * ec ^ 2 * d / 4 * C3 =138*68^2*233.33/4*9.=       335.16 kNm
```

SHEAR STRESS FORMULAES (PD5500 Section G.2.3.6.3)

```
Shear Stresses due to Longitudinal Shear Force, TauFl:
TauFl = 2 * Fl / (PI * deb * ec)
=2*0/(3.14*133.33*68)=                                     0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * Fc / (PI * deb * ec)
=2*0/(3.14*133.33*68)=                                     0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * Mt / (PI * deb ^ 2 * ec)
=2*0/(3.14*133.33^2*68)=                                   0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                           0.00 N/mm2
```

16.5.6 COMBINATIONS OF EXTERNAL LOADS AND INTERNAL PRESSURE

```
PhiP = P / Pmax =20/41.5=                                 0.4819

PhiZ = Fz / Fzmax =30/1154.98=                            0.0260

PhiTau = Tau / (0.5 * f) =0/(0.5*138)=                      0.00

PhiB = Sqr(( Mx / Mxmax) ^ 2 + (My / Mymax) ^ 2)
=Sqr((0/182.39)^2+(0/335.16)^2)=                           0.00

MaxAll = MAX(Abs(PhiP/C4+PhiZ),Abs(PhiZ),Abs(PhiP/C4-0.2*PhiZ)
=MAX(Abs(0.4819/1.1+0.026,)Abs(0.026,)Abs(0.4819/1.1-0.2*0.026)=  0.4641

PhiAll = Sqr( MaxAll ^ 2 + PhiB ^ 2 + PhiTau ^ 2)
=Sqr(0.4641^2+0^2+0^2)=                                 0.4641
```

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT EDGE OF PAD PhiP=0.4819 <= 1.0 =1« » (U= 48.1%) OK«
»PhiZ AT EDGE OF PAD PhiZ=0.026 <= 1.0=1« » (U= 2.5%) OK«
»PhiB AT EDGE OF PAD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT EDGE OF PAD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT EDGE OF PAD PhiAll=0.4641 <= 1.0=1« » (U= 46.4%) OK«

16.5.7 STRESS RANGES AND THEIR COMBINATIONS

16.5.7.1 LOAD RANGES

```
DeltaP = Max( Pmax , 0) - Min( Pmin , 0)
=Max(20,0)-Min(0,0)=                                      20.00 MPa

DeltaFz = Max( Fzmax , 0) - Min( Fzmin , 0)
=Max(30,0)-Min(0,0)=                                      30.00 kN

DeltaMx = Max( Mxmax , 0) - Min( Mxmin , 0)
=Max(0,0)-Min(0,0)=                                        0.00 kNm

DeltaMy = Max( Mymax , 0) - Min( Mymin , 0)
=Max(0,0)-Min(0,0)=                                        0.00 kNm

DeltaFl = Max( Flmax , 0) - Min( Flmin , 0)
=Max(0,0)-Min(0,0)=                                        0.00 kN

DeltaFc = Max( Fcmax , 0) - Min( Fcmin , 0)
=Max(0,0)-Min(0,0)=                                        0.00 kN

DeltaFshear = Sqr( DeltaFl ^ 2 + DeltaFc ^ 2)
=Sqr(0^2+0^2)=                                             0.00 kN

DeltaMt = Max( Mtmax , 0) - Min( Mtmin , 0)
=Max(0,0)-Min(0,0)=                                        0.00 kNm
```

16.5.7.2 EQVIVALENT SHELL THICKNESS

```
eeq = ea+Min(te*Lp/Sqr(D*(ea+te)),te)*Min(fp/f1)
=68+Min(40*50/Sqr(568*(68+40,))40)*Min(138/138,1)=          76.08 mm
```

16.5.7.3 STRESSES

```
VALUES FOR C1, C2 AND C3 FROM FIGURES16.5-2 to 16.5-4
C1 =    1.810  C2 =    4.900  C3 =    5.733
Tmp1 = Sqr( d * eb / (D * eeq))
=Sqr(78.19*27.57/(568*76.08))=                            0.2598
Tmp2 = (2+2*d/D*Tmp1+1.25*d/D*Sqr(D/eeq))/(1+eb/eeq*Tmp1)
=(2+2*78.19/568*0.2598+1.25*78.19/568*Sqr(568/76.08))/(1+27.57/76.08*0.2598
)=                                                          =     2.50
Stresses due to Pressure Range
SigP = DeltaP * D / (2 * eeq) * Tmp2                               (16.5-21)
=20*568/(2*76.08)*2.5=                                     186.48 N/mm2

Stresses due to Axial Load Range
SigFz = 2.25 / C1 * (DeltaFz / eeq ^ 2)                            (16.5-22)
=2.25/1.81*(30/76.08^2)=                                     6.44 N/mm2

Stresses due to Circumferential Moment Range
SigMx = 2.25 / C2 * (4 * DeltaMx / (eeq ^ 2 * d))                  (16.5-23)
=2.25/4.9*(4*0/(76.08^2*78.19))=                             0.00 N/mm2

Stresses due to Longitudinal Moment Range
SigMy = 2.25 / C3 * (4 * DeltaMy / (eeq ^ 2 * d))                  (16.5-24)
=2.25/5.73*(4*0/(76.08^2*78.19))=                            0.00 N/mm2

Shear Stresses due to Longitudinal Shear Force, DeltaFl:
TauFl = 2 * DeltaFl / (PI * deb * eeq)
=2*0/(3.14*133.33*76.08)=                                    0.00 N/mm2

Shear Stresses due to Circumferential Force, TauFc:
TauFc = 2 * DeltaFc / (PI * deb * eeq)
=2*0/(3.14*133.33*76.08)=                                    0.00 N/mm2

Shear Stresses due to Torsional Moment, TauMt:
TauMt = 2 * DeltaMt / (PI * deb ^ 2 * eeq)
=2*0/(3.14*133.33^2*76.08)=                                  0.00 N/mm2

Total Shear Stresses, Tau:
Tau = Sqr( TauFc ^ 2 + TauFl ^ 2) + TauMt
=Sqr(0^2+0^2)+0=                                             0.00 N/mm2

Total Stress Intensity due to Load Range
SigTot = Abs(SigT+Sqr((SigP+SigFz)^2+SigMx^2+SigMy^2+4*Tau^2))     (16.5-25)
=Abs(0+Sqr((186.48+6.44)^2+0^2+0^2+4*0^2))=                192.92 N/mm2
```

»Total Stress in Shell SigTot=192.92 <= 3*f=414[N/mm2] « » (U= 46.5%) OK«

16.5.8 NOZZLE LONGITUDINAL STRESSES

```
Maximum Longitudinal Stresses in Nozzle
SigLong = P*d/(4*eb)+4*MB/(PI*d^2*eb)+Fz/(PI*d*eb)
=20*78.19/(4*27.57)+4*0/(3.14*78.19^2*27.57)+30000/(3.14*78.19*27.57)
=    22.46 N/mm2
```

»Nozzle Long.Stress SigLong=22.46 <= fb=138[N/mm2] « » (U= 16.2%) OK«

16.14.6 COMPRESSIVE STRESS LIMITS

```
K = 1.21 * E * ea / (Sige * D)                                     (16.14-15)
=1.21*199925*27.57/(239*105.76)=                           263.86
alfa = 0.83 / Sqr( 1 + 0.005 * D / ea)                             (16.14-16)
=0.83/Sqr(1+0.005*105.76/27.57)=                          0.8222
delta = (1 - 0.4123 / (alfa * K) ^ 0.6) / S                        (16.14-19)
=(1-0.4123/(0.8222*263.86)^0.6)/1.5=                      0.6558
Maximum Allowable Compressive Stress
Sigcall = Sige * delta (16.14-20) =239*0.6558=               156.73 N/mm2
```

16.14.4 PERMISSIBLE INDIVIDUAL LOADS

```
Maximum Tensile Force Ftmax
Ftmax = PI * D * ea * f (16.14-1) =3.14*105.76*27.57*138=   1264.12 kN

Maximum Compressive Force Fcmax
Fcmax = PI * D * ea * Sigcall                                      (16.14-2)
=3.14*105.76*27.57*156.73=                                1435.68 kN

Maximum Bending Moment Mmax
Mmax = PI / 4 * D ^ 2 * ea * Sigcall                               (16.14-3)
=3.14/4*105.76^2*27.57*156.73=                              37.96 kNm

Longitudinal Stability Check (P=0)
LongStab = MB / Mmax + Abs( Fzmin) / Fcmax
=0/37.96+Abs(0)/1435.68=                                     0.00
```

»Nozzle Long.Stability LongStab=0 <= 1.0=1« » (U= 0%) OK«

```
Weight of Nozzle: 12.7kg  Pad: 12kg  Flange: 21kg
```

CALCULATION SUMMARY

```
Aavail = A1 + A2 + A3 + A4 + A5
=5412.6+4654.24+0+257.59+4000=                           14324.43 mm2

Total Area Required Areq
Areq = d * tr * F + 2 * tn * tr * F * (1 - fr1)
=78.19*39.68*1+2*27.57*39.68*1*(1-1)=                     3102.78 mm2
```

»UG-37 Nozzle Reinforcement Aavail=14324.43 >= Areq=3102.78[mm2] «» (U= 21.6%) OK«

MAXIMUM ALLOWABLE WORKING PRESSURE

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :HOT & CORR

```
Pmax (t,tn,Sv,Sn)(68,27.6,138,138) = ==                     41.50 MPa
```

MAXIMUM ALLOWABLE WORKING PRESSURE MAWP :NEW & COLD

```
Pmax (t,tn,Sv,Sn)(69,28.6,138,138) = ==                     42.25 MPa
```

MAX TEST PRESSURE (Uncorroded cond.at ambient temp.)

```
Pmax (t,tn,Sv,Sn)(69,28.6,234,234) = ==                     71.75 MPa
```

»UG-45 Min.Nozzle Neck Thk. UG45=7.21 <= tnb-tolerance=27.57[mm] «» (U= 26.1%) OK«

LOAD CASE NO: 1 - Load Case 1

STRESSES AT OUTER DIAMETER OF NOZZLE

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT NOZZLE OD PhiP=0.4819 <= 1.0 =1« » (U= 48.1%) OK«
»PhiZ AT NOZZLE OD PhiZ=0.0103 <= 1.0=1« » (U= 1%) OK«
»PhiB AT NOZZLE OD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT NOZZLE OD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT NOZZLE OD PhiAll=0.4484 <= 1.0=1« » (U= 44.8%) OK«

STRESSES AT OUTER EDGE OF PAD

16.5.6.4 Check of Individual Load Ratio Limits

»PhiP AT EDGE OF PAD PhiP=0.4819 <= 1.0 =1« » (U= 48.1%) OK«

»PhiZ AT EDGE OF PAD PhiZ=0.026 <= 1.0=1« » (U= 2.5%) OK«
»PhiB AT EDGE OF PAD PhiB=0 <= 1.0=1« » (U= 0%) OK«
»PhiTau AT EDGE OF PAD PhiTau=0 <= 1.0=1« » (U= 0%) OK«
»PhiAll AT EDGE OF PAD PhiAll=0.4641 <= 1.0=1« » (U= 46.4%) OK«

16.5.7 STRESS RANGES AND THEIR COMBINATIONS

»Total Stress in Shell SigTot=192.92 <= 3*f=414[N/mm2] « » (U= 46.5%) OK«

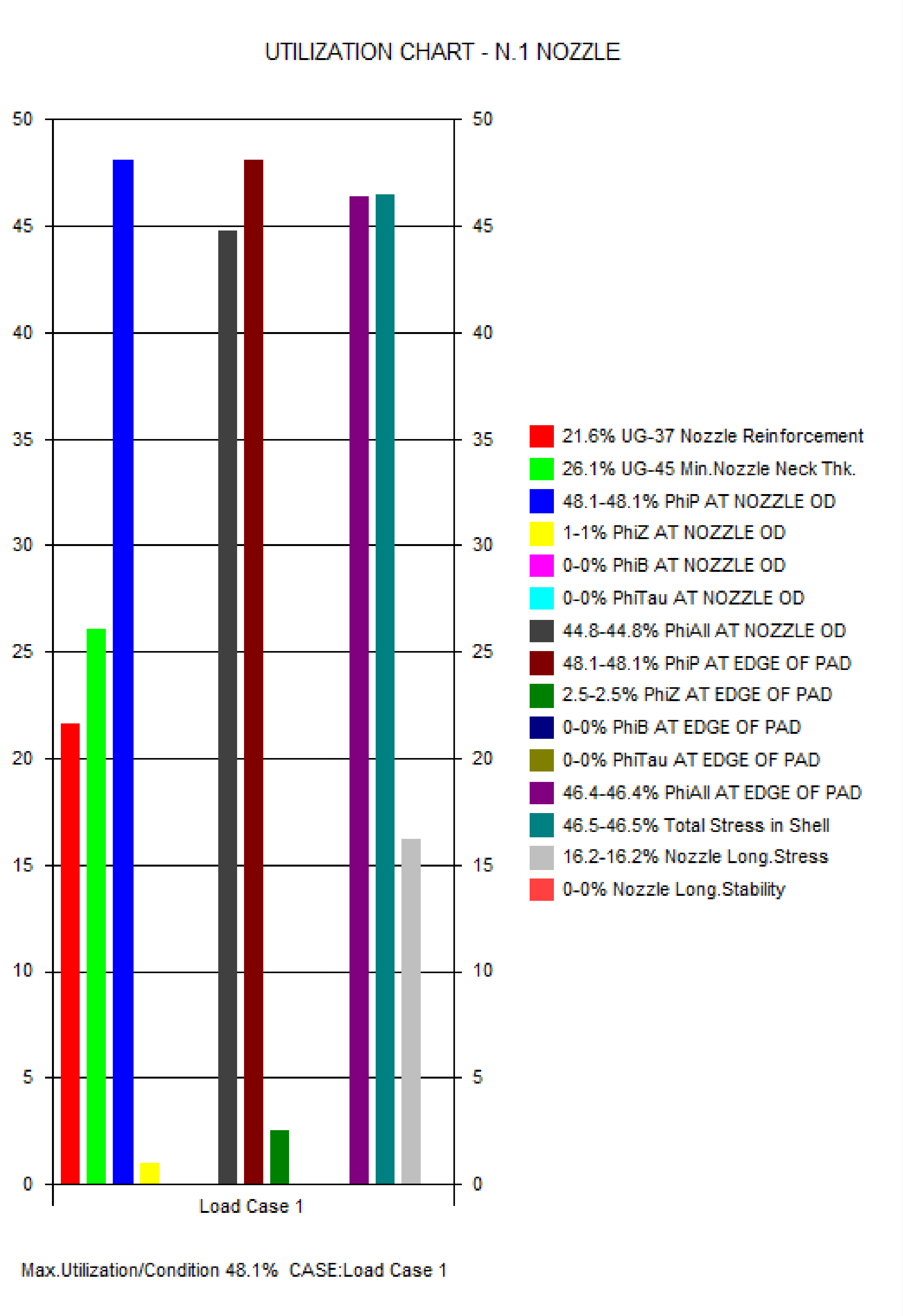
16.5.8 NOZZLE LONGITUDINAL STRESSES

»Nozzle Long.Stress SigLong=22.46 <= fb=138[N/mm2] « » (U= 16.2%) OK«
»Nozzle Long.Stability LongStab=0 <= 1.0=1« » (U= 0%) OK«

ERROR:a) ; ea/D <= 0.1 Outside Valid Range

Volume:0 m3 Weight:45.3 kg (SG= 7.85)

ASME VIII Div.1:2007 A09 - UG-37 REINFORCEMENT REQUIRED FOR OPENINGS IN SHELLS
N.1 Nozzle 17 Apr. 2012 17:27 ConnID:S1.1

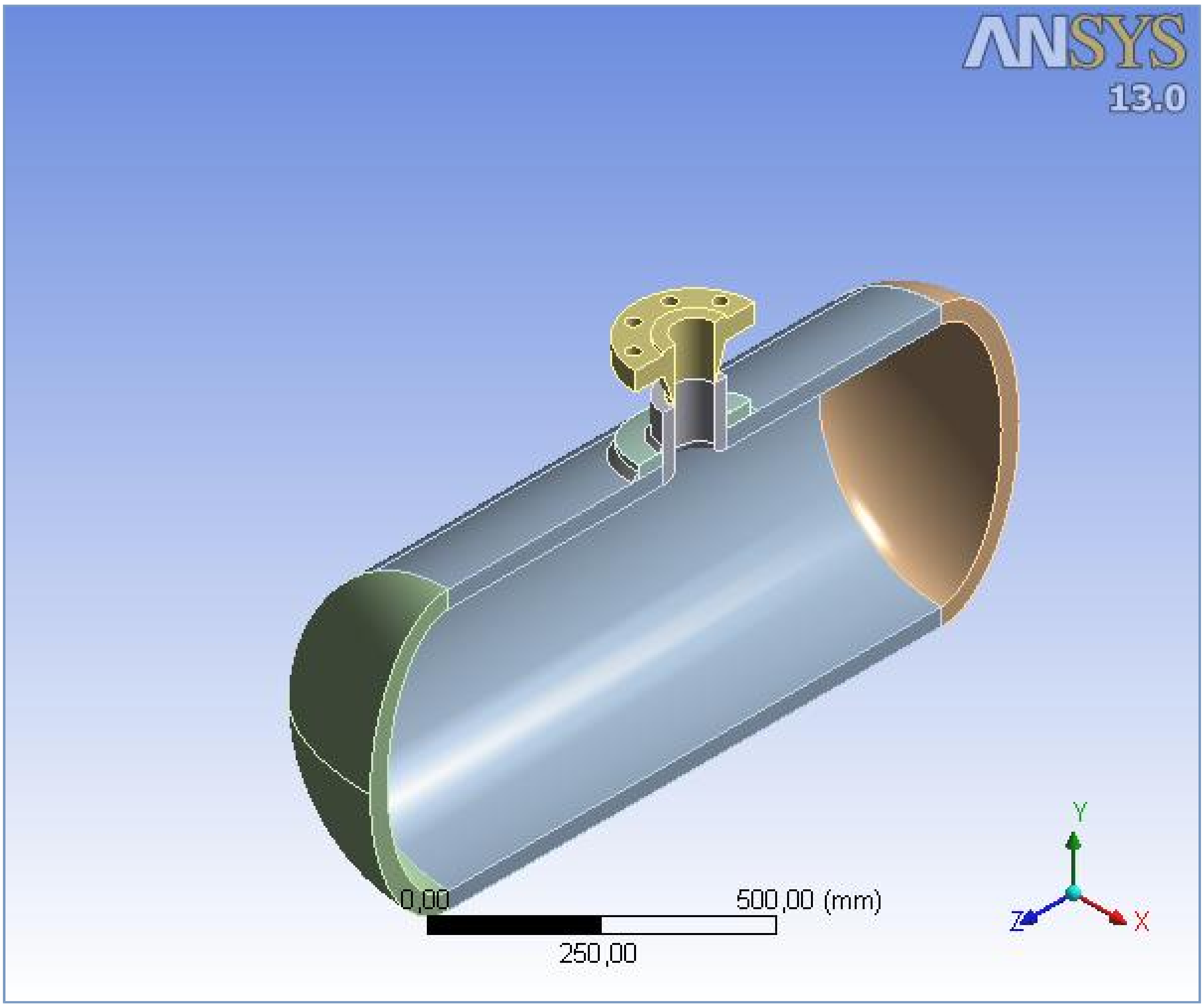


Max.Utilization/Condition 48.1% CASE:Load Case 1

APPENDIX D: Calculation report from ANSYS, Direct Route - NS-EN 13445; 2009, Gross plastic deformation (35 mm @ 100 bar)

Project

First Saved	Monday, April 16, 2012
Last Saved	Thursday, April 19, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thin wall\Assembly\weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	283,27 mm
Length Y	766,01 mm
Length Z	1317, mm
Properties	
Volume	4,1686e+007 mm³
Mass	327,23 kg
Scale Factor Value	1,

Statistics	
Bodies	7
Active Bodies	7
Nodes	29812
Elements	14640
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	Yes
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell:1*	*End cap:1*	*End cap:2*	*Reinforcement pad2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	SA-516 grade 70				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127,52 mm	283, mm	283,27 mm		114,53 mm
Length Y	125,14 mm	566, mm	566,01 mm		49,228 mm
Length Z	254,79 mm	1000, mm	158,52 mm		229,01 mm
Properties					
Volume	40570 mm³	2,8961e+007 mm³	5,4737e+006 mm³	5,4736e+006 mm³	3,6082e+005 mm³
Mass	0,31847 kg	227,34 kg	42,969 kg	42,968 kg	2,8325 kg
Centroid X	-69,697 mm	-170,08 mm	-125,46 mm	-125,47 mm	-59,042 mm

Centroid Y	281,88 mm	-2,0967 mm	-8,7545e-003 mm	5,5528e-002 mm	287,9 mm
Centroid Z	-0,11401 mm	-3,1226e-007 mm	578,79 mm	-578,79 mm	-3,779e-005 mm
Moment of Inertia Ip1	1952,7 kg·mm²	2,7057e+007 kg·mm²	9,7267e+005 kg·mm²	9,7333e+005 kg·mm²	12232 kg·mm²
Moment of Inertia Ip2	2395,9 kg·mm²	2,0592e+007 kg·mm²	3,1819e+005 kg·mm²	3,1829e+005 kg·mm²	14729 kg·mm²
Moment of Inertia Ip3	548,99 kg·mm²	9,473e+006 kg·mm²	1,1875e+006 kg·mm²	1,1882e+006 kg·mm²	2811, kg·mm²
Statistics					
Nodes	590	10793	4472	4903	973
Elements	143	5373	2483	2759	144
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 900 RF:1*	*Nozzle2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	SA-516 grade 70	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	120,5 mm	64,497 mm
Length Y	108, mm	135,53 mm
Length Z	241, mm	128,97 mm
Properties		
Volume	8,49e+005 mm³	5,2735e+005 mm³
Mass	6,6646 kg	4,1397 kg
Centroid X	-50,292 mm	-33,99 mm
Centroid Y	452,57 mm	311,08 mm
Centroid Z	-2,5746e-007 mm	2,3667e-004 mm
Moment of Inertia Ip1	25412 kg·mm²	11450 kg·mm²
Moment of Inertia Ip2	29464 kg·mm²	6997,5 kg·mm²
Moment of Inertia Ip3	8702,5 kg·mm²	6841,3 kg·mm²
Statistics		
Nodes	5902	2179
Elements	3374	364
Mesh Metric	None	

Coordinate Systems

TABLE 5
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian

Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	3,8742 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 8
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region*	*Contact Region 2*	*Contact Region 3*	*Contact Region 4*	*Contact Region 5*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face	2 Faces	1 Face
Target	1 Face	2 Faces	1 Face	2 Faces	1 Face
Contact Bodies	Welds				Main shell:1
Target Bodies	Main shell:1	Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1	Nozzle2:1	End cap:1
Definition					
Type	Bonded				

Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

TABLE 9
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region 6*	*Bonded - Main shell:1 To Reinforcement pad2:1*	*Contact Region 8*	*Contact Region 9*	*Contact Region 10*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face				
Target	1 Face				
Contact Bodies	Main shell:1			Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1
Target Bodies	End cap:2	Reinforcement pad2:1	Nozzle2:1		
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

Mesh

TABLE 10
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	2,26280 mm

Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	29812
Elements	14640
Mesh Metric	None

Static Structural (A5)

TABLE 11
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	20, °C
Generate Input Only	No

TABLE 12
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart	

Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thin wall configuration - Direct Route - NS-EN 13445; 2009_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 13
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Acceleration*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0, mm/s² (ramped)
Y Component	-11767 mm/s² (ramped)
Z Component	0, mm/s² (ramped)
Suppressed	No

FIGURE 1
Model (A4) > Static Structural (A5) > Acceleration

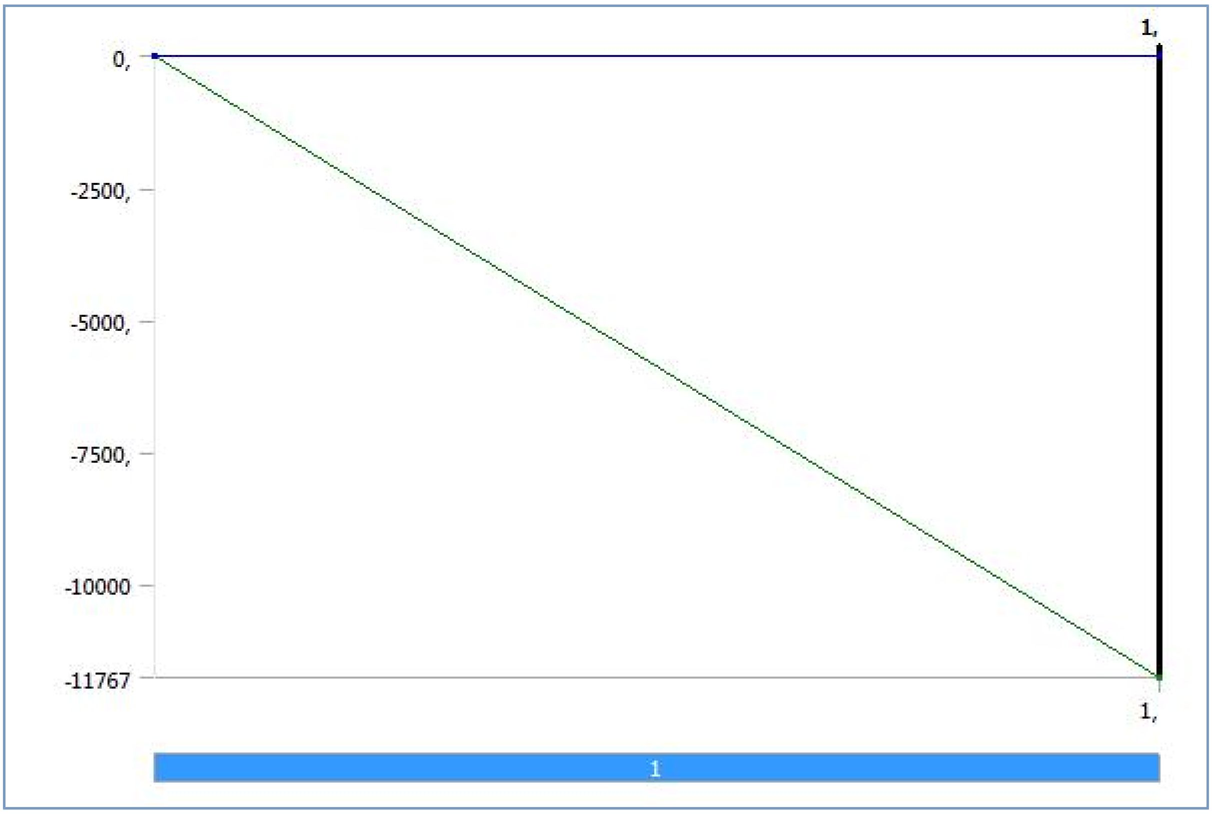


TABLE 14
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	15 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components
Magnitude	12, MPa (ramped)				
Suppressed	No				
Coordinate System			Global Coordinate System		Global Coordinate System
X Component			0, mm (ramped)		0, N (ramped)
Y Component			Free		18000 N (ramped)
Z Component			0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

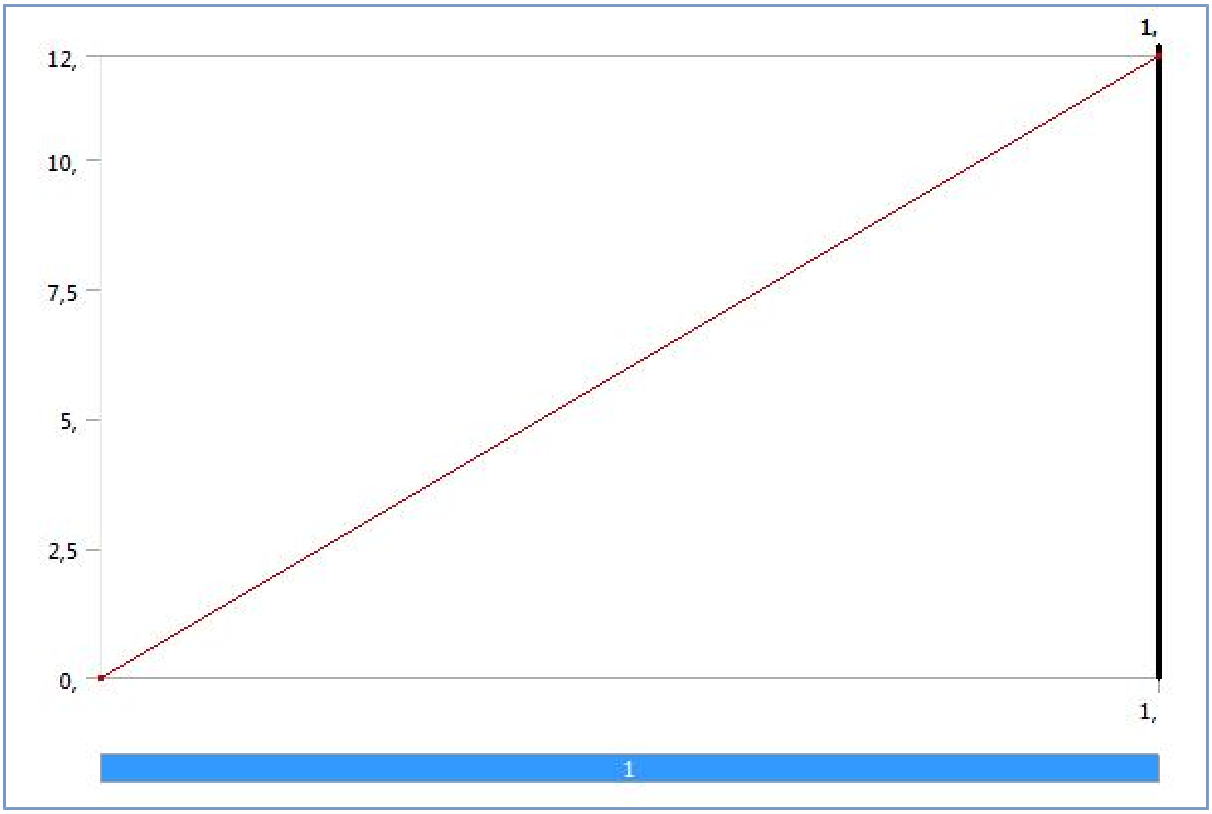


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

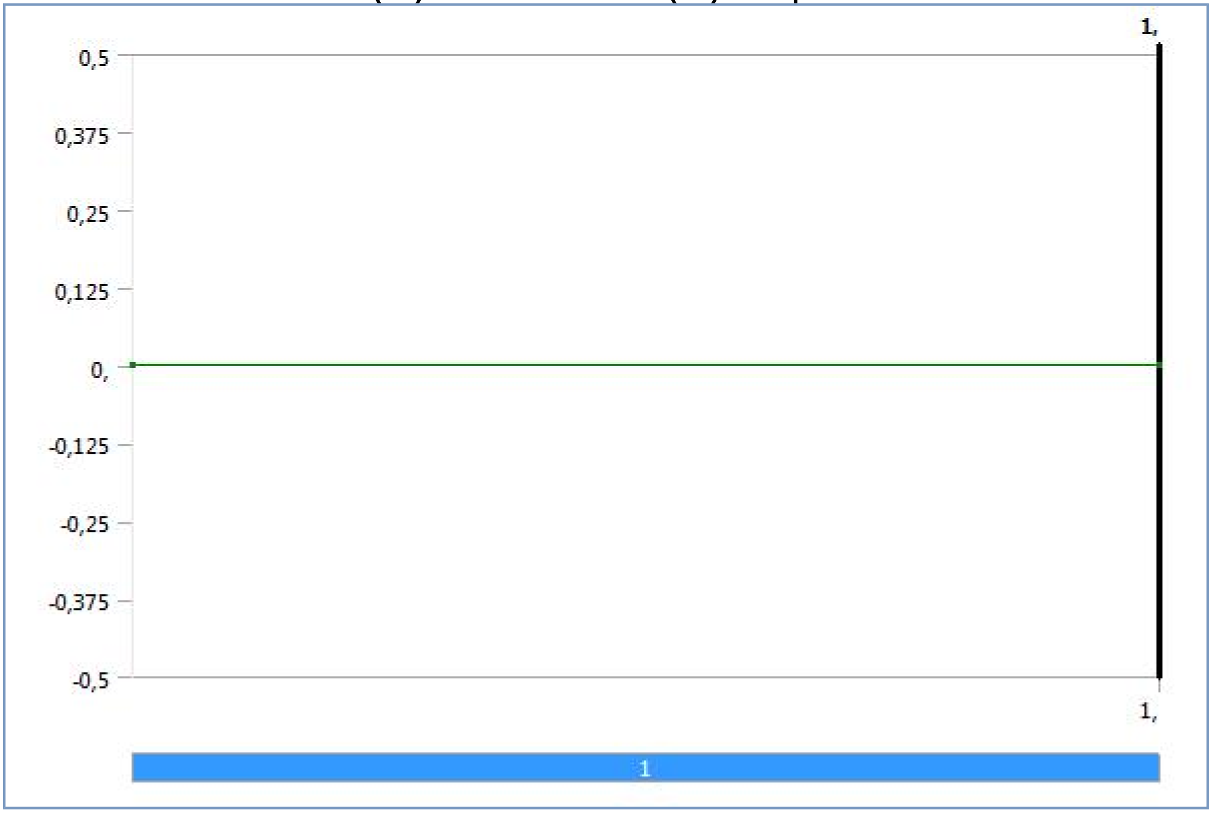
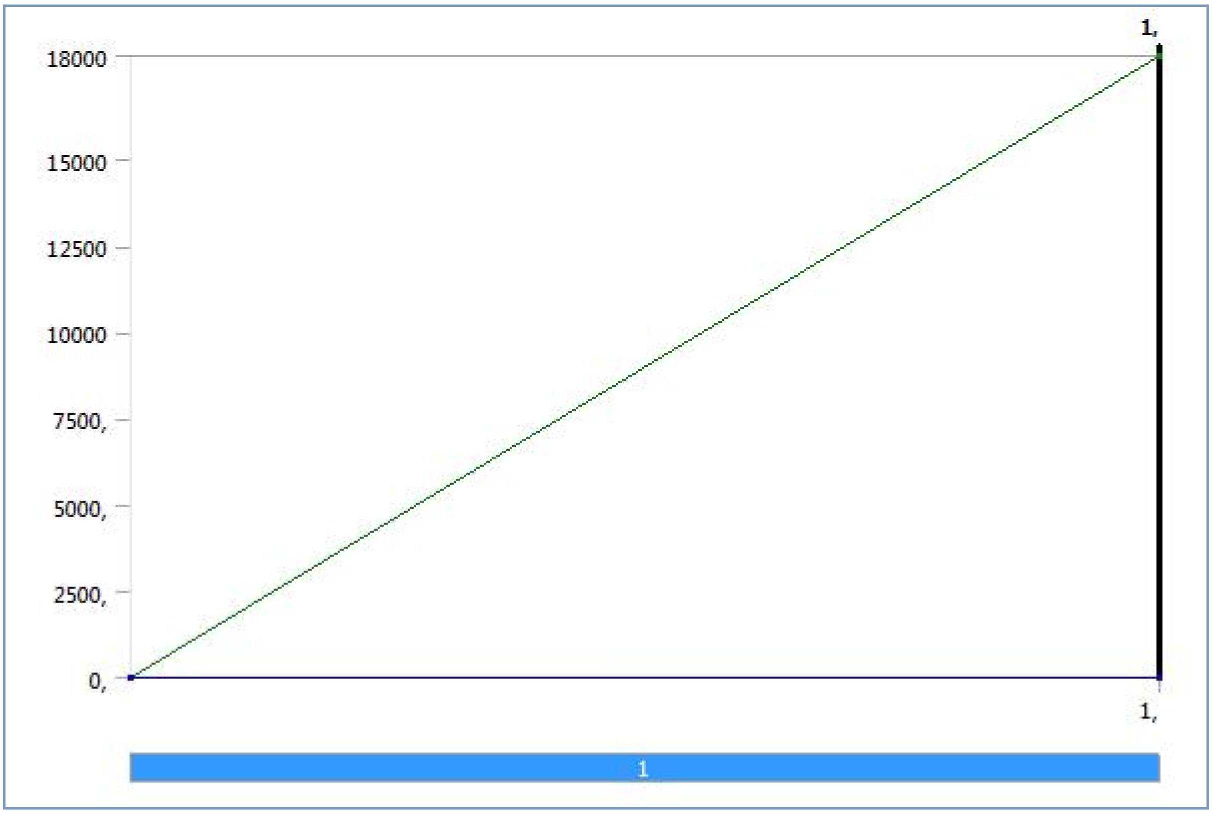


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 15
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Stress Intensity*	*Maximum Principal Elastic Strain*	*Total Deformation*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Stress Intensity	Maximum Principal Elastic Strain	Total Deformation
By	Time		

Display Time	Last		
Calculate Time History	Yes		
Identifier			
Integration Point Results			
Display Option	Averaged		
Results			
Minimum	0,17385 MPa	3,0166e-006 mm/mm	2,0702e-004 mm
Maximum	258,54 MPa	1,2245e-003 mm/mm	0,24472 mm
Minimum Occurs On	3 inch Weldneck Flange 900 RF:1		Main shell:1
Maximum Occurs On	Nozzle2:1		End cap:1
Information			
Time	1, s		
Load Step	1		
Substep	1		
Iteration Number	1		

Material Data

SA-516 grade 70

TABLE 18
SA-516 grade 70 > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 19
SA-516 grade 70 > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 20
SA-516 grade 70 > Compressive Yield Strength

Compressive Yield Strength MPa
260

TABLE 21
SA-516 grade 70 > Tensile Yield Strength

Tensile Yield Strength MPa
260

TABLE 22
SA-516 grade 70 > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
485

TABLE 23
SA-516 grade 70 > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 24
SA-516 grade 70 > Strain-Life Parameters

Strength	Strength	Ductility	Ductility	Cyclic Strength	Cyclic Strain

Coefficient MPa	Exponent	Coefficient	Exponent	Coefficient MPa	Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 25
SA-516 grade 70 > Isotropic Elasticity

Temperature C	Young’s Modulus MPa	Poisson’s Ratio	Bulk Modulus MPa	Shear Modulus MPa
20	2.e+005	0.3	1.6667e+005	76923

TABLE 26
SA-516 grade 70 > Isotropic Relative Permeability

Relative Permeability
10000

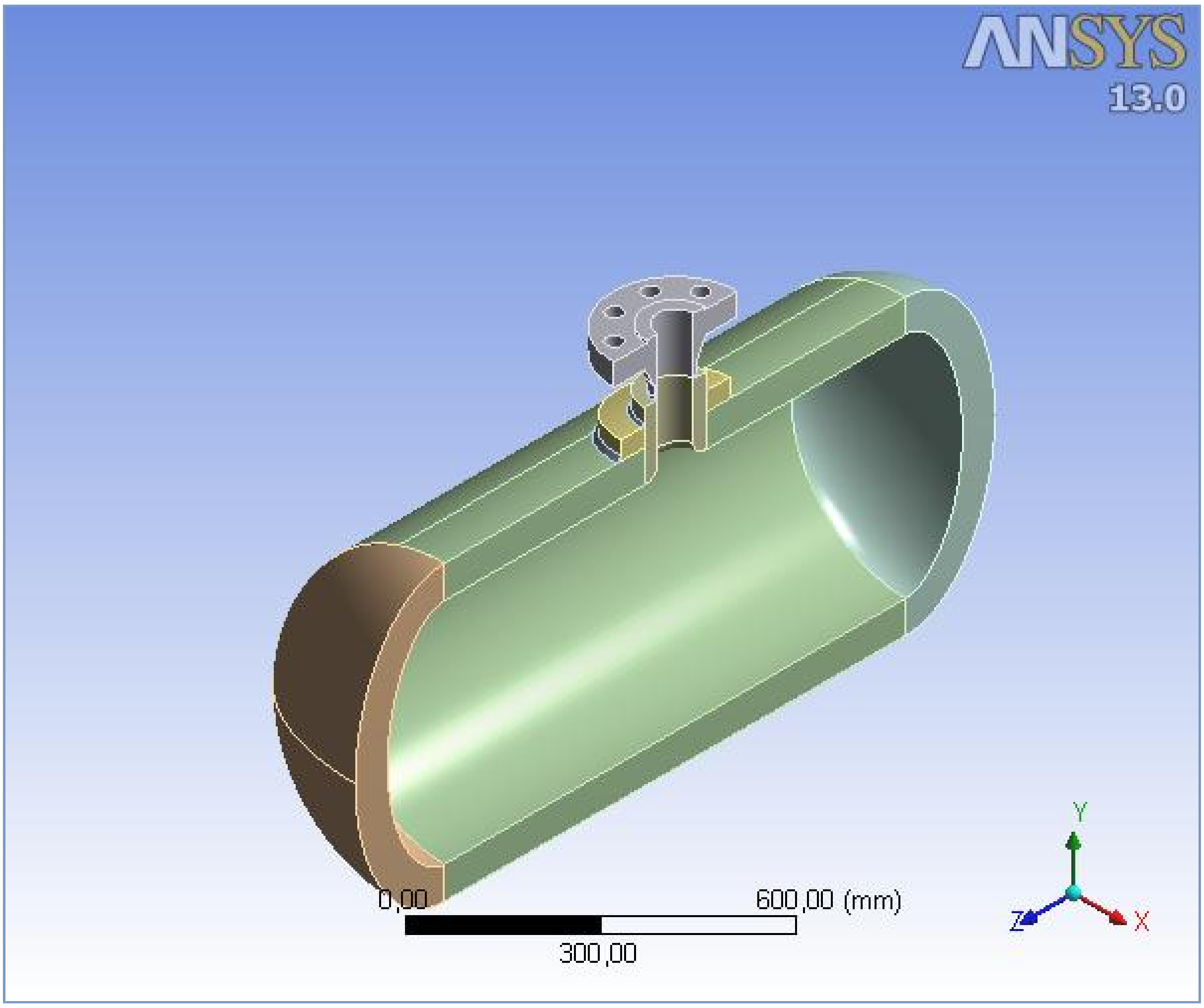
TABLE 27
SA-516 grade 70 > Uniaxial Test Data

Strain mm mm^-1	Stress MPa	Temperature C
0	0	20
2.e-003	260	20
5.e-002	485	20

APPENDIX E: Calculation report from ANSYS, Direct Route - NS-EN 13445; 2009 Gross plastic deformation (70 mm @ 200 bar)

Project

First Saved	Tuesday, April 10, 2012
Last Saved	Monday, April 23, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thick wall\Assembly\Weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	318,32 mm
Length Y	836, mm
Length Z	1387, mm
Properties	
Volume	8,98e+007 mm³
Mass	704,93 kg

Scale Factor Value	1,
Statistics	
Bodies	7
Active Bodies	7
Nodes	38820
Elements	19469
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell thick:1*	*End cap thick:1*	*End cap thick:2*	*Reinforcement pad thick2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127, mm	318, mm	318,32 mm		117,68 mm
Length Y	110,06 mm	636, mm	636,01 mm	636, mm	62,595 mm
Length Z	253,91 mm	1000, mm	193,51 mm		235,31 mm
Properties					
Volume	22864 mm^3	6,1723e+007 mm^3	1,277e+007 mm^3		5,9435e+005 mm^3
Mass	0,17948 kg	484,52 kg	100,24 kg		4,6657 kg
Centroid X	-67,2 mm	-182,3 mm	-135,52 mm	-135,53 mm	-60,897 mm

Centroid Y	325,62 mm	-2,3053 mm	-1,8244e-002 mm	4,1893e-002 mm	330,95 mm
Centroid Z	1,9269e-003 mm	2,5059e-006 mm	588,86 mm	-588,86 mm	-9,7601e-007 mm
Moment of Inertia Ip1	1106,8 kg·mm²	6,024e+007 kg·mm²	2,6829e+006 kg·mm²	2,684e+006 kg·mm²	21904 kg·mm²
Moment of Inertia Ip2	1392,8 kg·mm²	4,4447e+007 kg·mm²	9,059e+005 kg·mm²	9,061e+005 kg·mm²	25881 kg·mm²
Moment of Inertia Ip3	419,67 kg·mm²	2,3283e+007 kg·mm²	3,2488e+006 kg·mm²	3,25e+006 kg·mm²	5250,6 kg·mm²
Statistics					
Nodes	598	16787	5073	4260	1329
Elements	145	9824	2970	789	216
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 1500 RF:1*	*Nozzle thick2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	Structural Steel	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	133,5 mm	67,677 mm
Length Y	124, mm	155,41 mm
Length Z	267, mm	135,33 mm
Properties		
Volume	1,2214e+006 mm³	6,998e+005 mm³
Mass	9,5877 kg	5,4935 kg
Centroid X	-56,09 mm	-35,249 mm
Centroid Y	483,94 mm	320,86 mm
Centroid Z	2,2844e-005 mm	-5,6781e-004 mm
Moment of Inertia Ip1	45407 kg·mm²	18168 kg·mm²
Moment of Inertia Ip2	53311 kg·mm²	10053 kg·mm²
Moment of Inertia Ip3	15378 kg·mm²	11576 kg·mm²
Statistics		
Nodes	8481	2292
Elements	5135	390
Mesh Metric	None	

Coordinate Systems

TABLE 5
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian

Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	4,1262 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 8
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Welds To 3 inch Weldneck Flange 1500 RF:1*	*Bonded - Welds To Nozzle thick2:1*	*Bonded - Main shell thick:1 To End cap thick:1*	*Bonded - Main shell thick:1 To End cap thick:2*	*Bonded - Main shell thick:1 To Reinforcement pad thick2:1*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face		2 Faces
Target	1 Face	2 Faces	1 Face		
Contact Bodies	Welds		Main shell thick:1		
Target Bodies	3 inch Weldneck Flange 1500 RF:1	Nozzle thick2:1	End cap thick:1	End cap thick:2	Reinforcement pad thick2:1

Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

TABLE 9
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Main shell thick:1 To Nozzle thick2:1*	*Bonded - Reinforcement pad thick2:1 To Nozzle thick2:1*	*Bonded - 3 inch Weldneck Flange 1500 RF:1 To Nozzle thick2:1*	*Contact Region 11*	*Contact Region 13*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face			3 Faces	2 Faces
Target	1 Face			2 Faces	
Contact Bodies	Main shell thick:1	Reinforcement pad thick2:1	3 inch Weldneck Flange 1500 RF:1	Welds	
Target Bodies	Nozzle thick2:1			Main shell thick:1	Reinforcement pad thick2:1
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

Mesh

TABLE 10
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly

Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	1,24280 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	38820
Elements	19469
Mesh Metric	None

TABLE 11
Model (A4) > Mesh > Mesh Controls

Object Name	*Automatic Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	7 Bodies
Definition	
Suppressed	No
Method	Automatic
Element Midside Nodes	Use Global Setting

Static Structural (A5)

TABLE 12
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 13
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thick wall configuration - Direct Route - NS-EN 13445; 2009_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 14
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Acceleration*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0, mm/s² (ramped)
Y Component	-11767 mm/s² (ramped)

Z Component	0, mm/s² (ramped)
Suppressed	No

FIGURE 1
Model (A4) > Static Structural (A5) > Acceleration

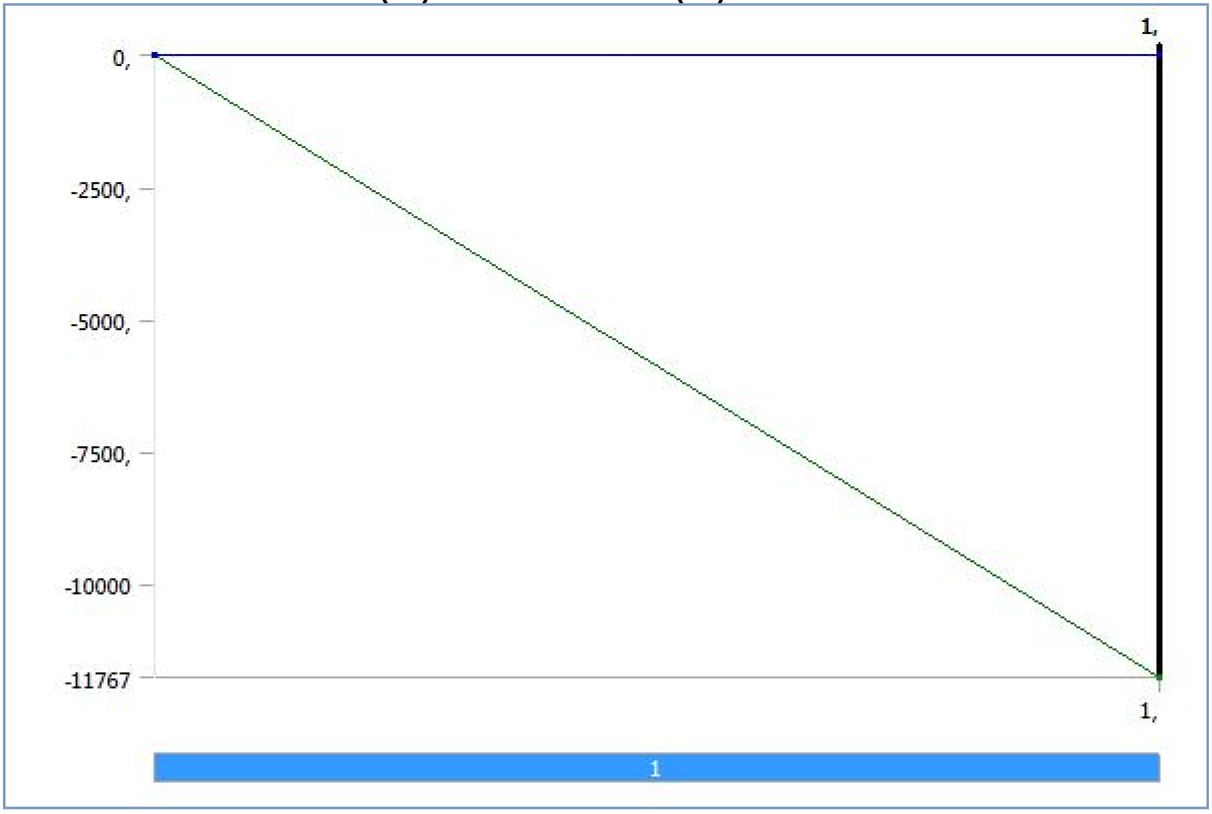


TABLE 15
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	16 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components
Magnitude	24, MPa (ramped)				
Suppressed	No				
Coordinate System			Global Coordinate System		Global Coordinate System
X Component			0, mm (ramped)		0, N (ramped)
Y Component			Free		18000 N (ramped)
Z Component			0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

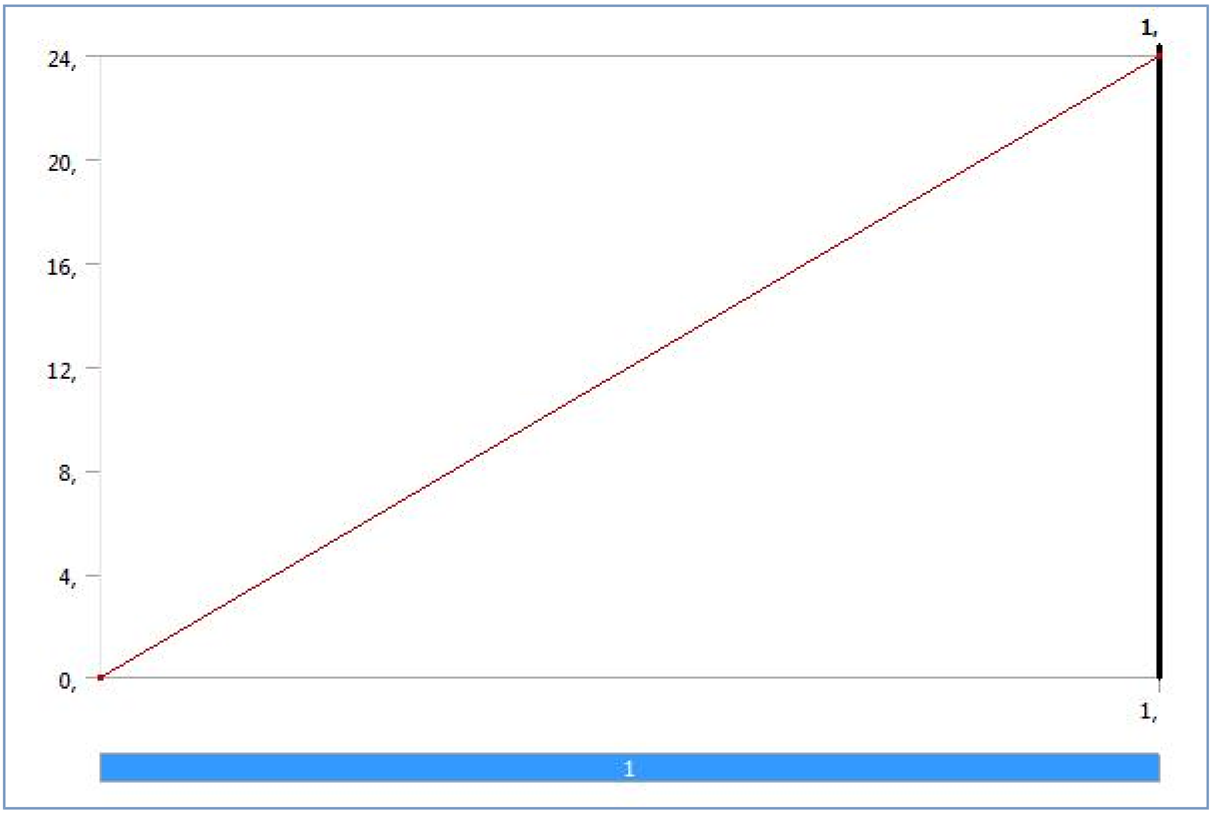


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

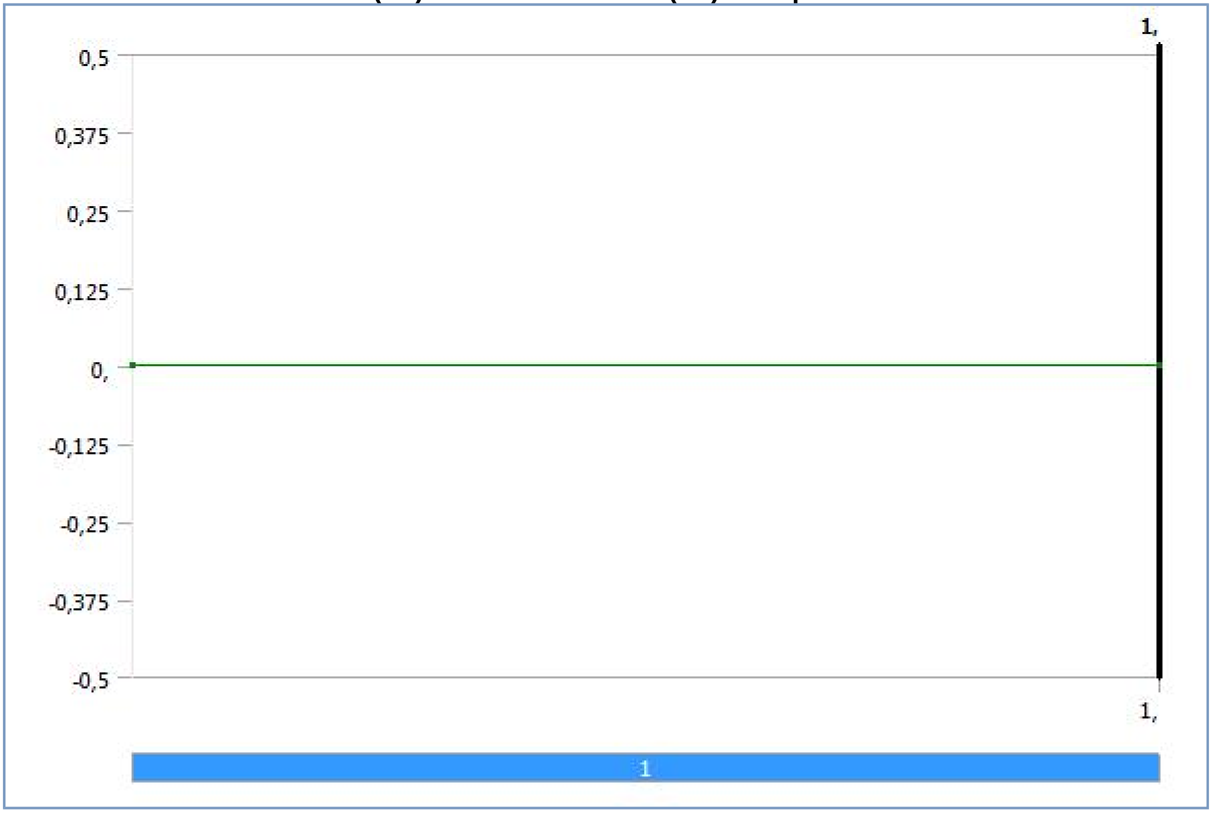
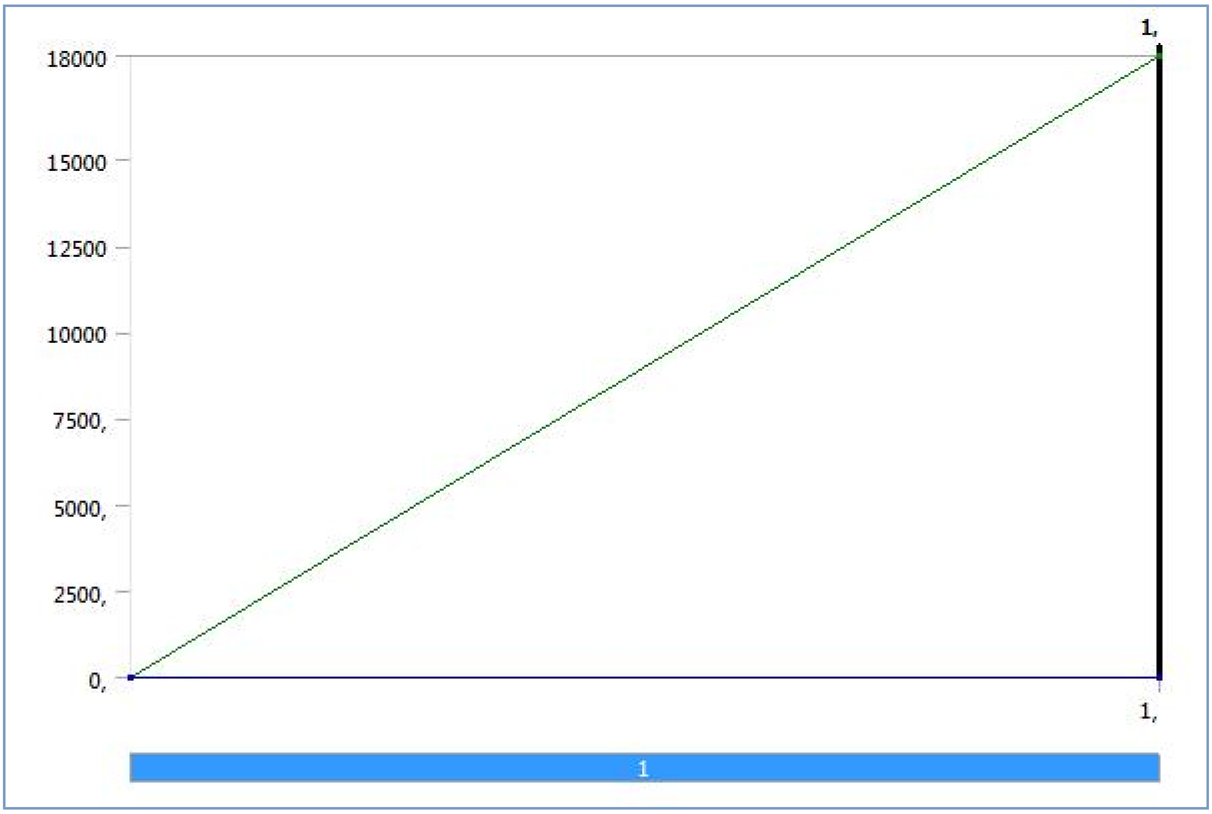


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 16
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 17
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 18
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Maximum Principal Elastic Strain*	*Total Deformation*	*Stress Intensity*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Maximum Principal Elastic	Total	Stress Intensity

	Strain	Deformation	
By	Time		
Display Time	Last		
Calculate Time History	Yes		
Identifier			
Integration Point Results			
Display Option	Averaged		Averaged
Results			
Minimum	2,6187e-006 mm/mm	1,6229e-004 mm	0,44648 MPa
Maximum	1,5268e-003 mm/mm	0,18728 mm	326,02 MPa
Minimum Occurs On	3 inch Weldneck Flange 1500 RF:1	Main shell thick:1	3 inch Weldneck Flange 1500 RF:1
Maximum Occurs On	Nozzle thick2:1	Main shell thick:1	Nozzle thick2:1
Information			
Time	1, s		
Load Step	1		
Substep	1		
Iteration Number	1		

Material Data

Structural Steel

TABLE 19
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 20
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 21
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
165

TABLE 22
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
165

TABLE 23
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
310

TABLE 24
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 25
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 26
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 27
Structural Steel > Isotropic Elasticity

Temperature C	Young’s Modulus MPa	Poisson’s Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

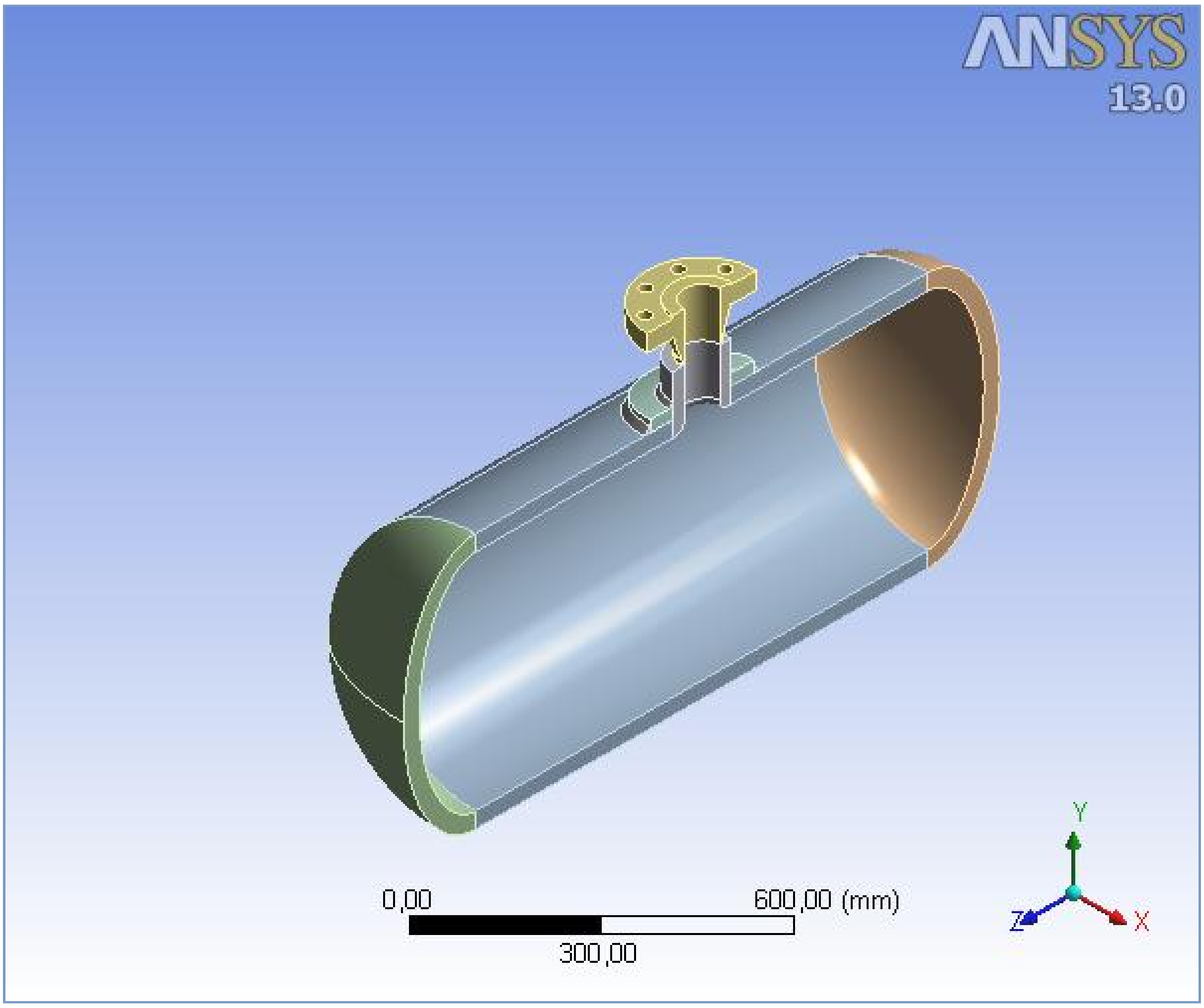
TABLE 28
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

APPENDIX F: Calculation report from ANSYS, Elastic Stress Analysis - ASME VIII div. 2; 2010, Protection against plastic collapse (35 mm @ 100 bar)

Project

First Saved	Monday, April 16, 2012
Last Saved	Wednesday, April 25, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thin wall\Assembly\weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	283,27 mm
Length Y	766,01 mm
Length Z	1317, mm
Properties	
Volume	4,1686e+007 mm³

Mass	327,23 kg
Scale Factor Value	1,
Statistics	
Bodies	7
Active Bodies	7
Nodes	29812
Elements	14640
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	Yes
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell:1*	*End cap:1*	*End cap:2*	*Reinforcement pad2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	SA-516 grade 70				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127,52 mm	283, mm	283,27 mm		114,53 mm
Length Y	125,14 mm	566, mm	566,01 mm		49,228 mm
Length Z	254,79 mm	1000, mm	158,52 mm		229,01 mm
Properties					
Volume	40570 mm^3	2,8961e+007 mm^3	5,4737e+006 mm^3	5,4736e+006 mm^3	3,6082e+005 mm^3
Mass	0,31847 kg	227,34 kg	42,969 kg	42,968 kg	2,8325 kg

Centroid X	-69,697 mm	-170,08 mm	-125,46 mm	-125,47 mm	-59,042 mm
Centroid Y	281,88 mm	-2,0967 mm	-8,7545e-003 mm	5,5528e-002 mm	287,9 mm
Centroid Z	-0,11401 mm	-3,1226e-007 mm	578,79 mm	-578,79 mm	-3,779e-005 mm
Moment of Inertia Ip1	1952,7 kg·mm^2	2,7057e+007 kg·mm^2	9,7267e+005 kg·mm^2	9,7333e+005 kg·mm^2	12232 kg·mm^2
Moment of Inertia Ip2	2395,9 kg·mm^2	2,0592e+007 kg·mm^2	3,1819e+005 kg·mm^2	3,1829e+005 kg·mm^2	14729 kg·mm^2
Moment of Inertia Ip3	548,99 kg·mm^2	9,473e+006 kg·mm^2	1,1875e+006 kg·mm^2	1,1882e+006 kg·mm^2	2811, kg·mm^2
Statistics					
Nodes	590	10793	4472	4903	973
Elements	143	5373	2483	2759	144
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 900 RF:1*	*Nozzle2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	SA-516 grade 70	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	120,5 mm	64,497 mm
Length Y	108, mm	135,53 mm
Length Z	241, mm	128,97 mm
Properties		
Volume	8,49e+005 mm^3	5,2735e+005 mm^3
Mass	6,6646 kg	4,1397 kg
Centroid X	-50,292 mm	-33,99 mm
Centroid Y	452,57 mm	311,08 mm
Centroid Z	-2,5746e-007 mm	2,3667e-004 mm
Moment of Inertia Ip1	25412 kg·mm^2	11450 kg·mm^2
Moment of Inertia Ip2	29464 kg·mm^2	6997,5 kg·mm^2
Moment of Inertia Ip3	8702,5 kg·mm^2	6841,3 kg·mm^2
Statistics		
Nodes	5902	2179
Elements	3374	364
Mesh Metric	None	

TABLE 5
Model (A4) > Construction Geometry

Object Name	*Construction Geometry*
State	Fully Defined
Display	
Show Mesh	No

TABLE 6
Model (A4) > Construction Geometry > Paths

Object Name	*Path*	*Path 2*	*Path 3*	*Path 4*
State	Fully Defined			
Definition				
Path Type	Two Points			
Path Coordinate System	Global Coordinate System			
Number of Sampling Points	47,			
Suppressed	No			
Start				
Coordinate System	Global Coordinate System			
Start X Coordinate	8,0745e-014 mm	2,7271e-013 mm	0, mm	-4,0022e-013 mm
Start Y Coordinate	283, mm	258,26 mm	0, mm	308, mm
Start Z Coordinate	164,99 mm	564,81 mm	658,5 mm	-114,5 mm
Location	Defined			
End				
Coordinate System	Global Coordinate System			
End X Coordinate	7,5194e-014 mm	2,437e-013 mm	1,6128e-014 mm	5,984e-014 mm
End Y Coordinate	248, mm	229,24 mm	-1,9752e-030 mm	250,69 mm
End Z Coordinate	164,99 mm	545,01 mm	623,5 mm	-49,079 mm
Location	Defined			

Coordinate Systems

TABLE 7
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 8
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 9
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	

Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	3,8742 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 10
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region*	*Contact Region 2*	*Contact Region 3*	*Contact Region 4*	*Contact Region 5*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face	2 Faces	1 Face
Target	1 Face	2 Faces	1 Face	2 Faces	1 Face
Contact Bodies	Welds				Main shell:1
Target Bodies	Main shell:1	Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1	Nozzle2:1	End cap:1
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

TABLE 11
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region 6*	*Bonded - Main shell:1 To Reinforcement pad2:1*	*Contact Region 8*	*Contact Region 9*	*Contact Region 10*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face				
Target	1 Face				
Contact Bodies	Main shell:1			Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1
Target Bodies	End cap:2	Reinforcement pad2:1	Nozzle2:1		
Definition					
Type	Bonded				

Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Mesh

TABLE 12
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	2,26280 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	29812
Elements	14640
Mesh Metric	None

Static Structural (A5)

TABLE 13
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	20, °C
Generate Input Only	No

TABLE 14
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thin wall configuration - Elastic Stress Analysis - ASME VIII div.2; 2010_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes

Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 15
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Standard Earth Gravity*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	-0, mm/s² (ramped)
Y Component	-9806,6 mm/s² (ramped)
Z Component	-0, mm/s² (ramped)
Suppressed	No
Direction	-Y Direction

FIGURE 1
Model (A4) > Static Structural (A5) > Standard Earth Gravity

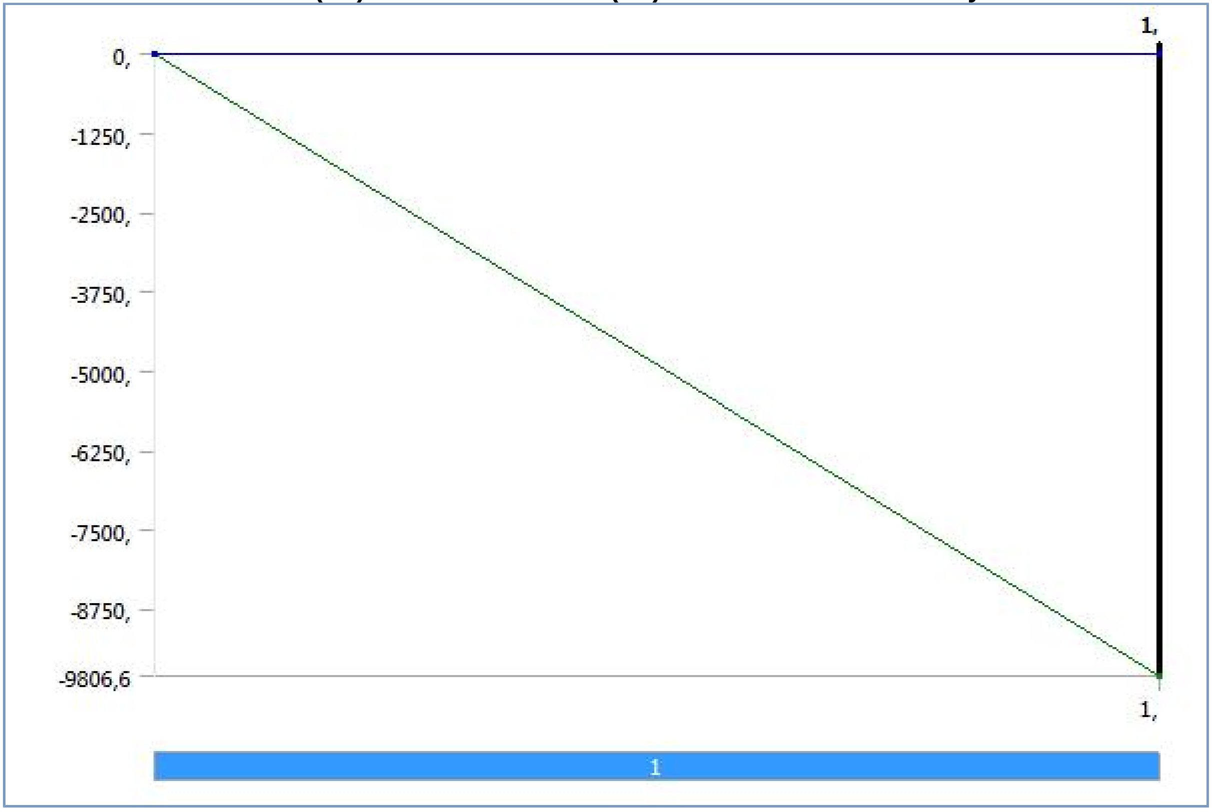


TABLE 16
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	15 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components

Magnitude	10, MPa (ramped)			
Suppressed	No			
Coordinate System		Global Coordinate System		Global Coordinate System
X Component		0, mm (ramped)		0, N (ramped)
Y Component		Free		15000 N (ramped)
Z Component		0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

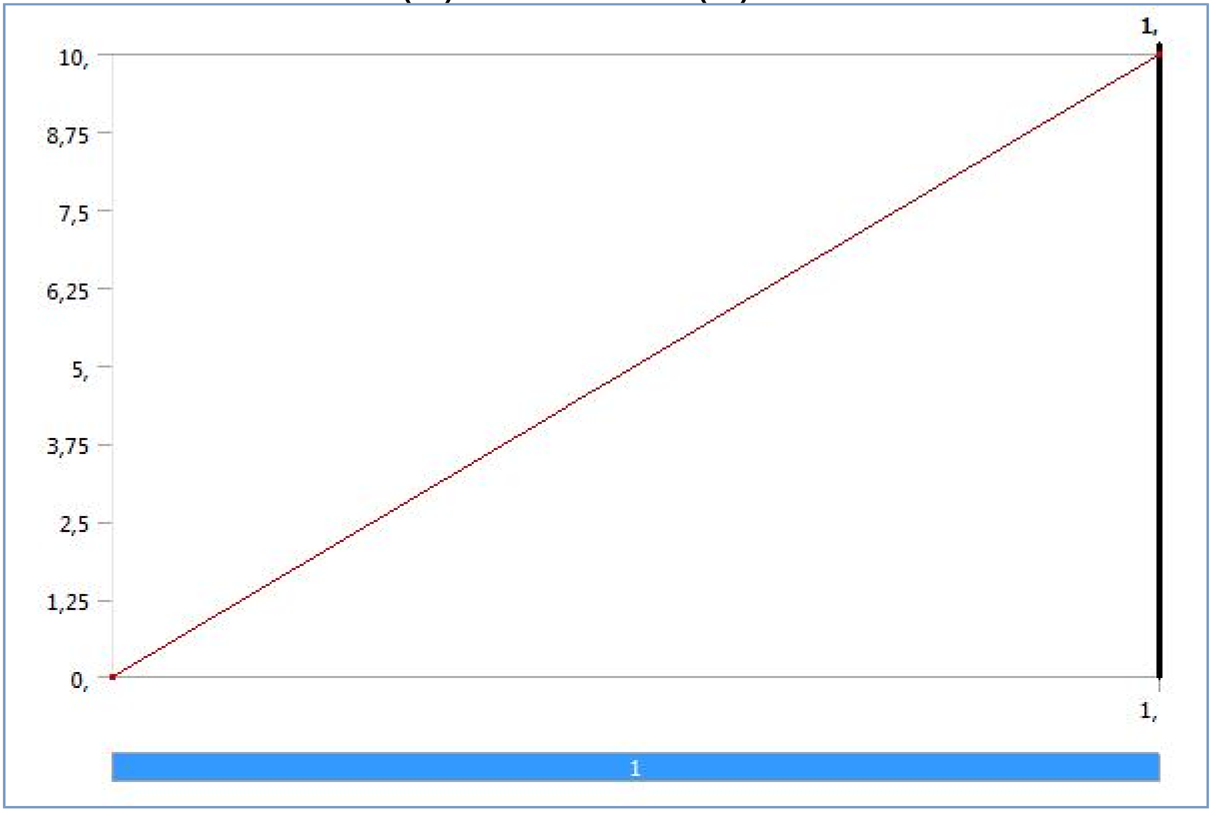


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

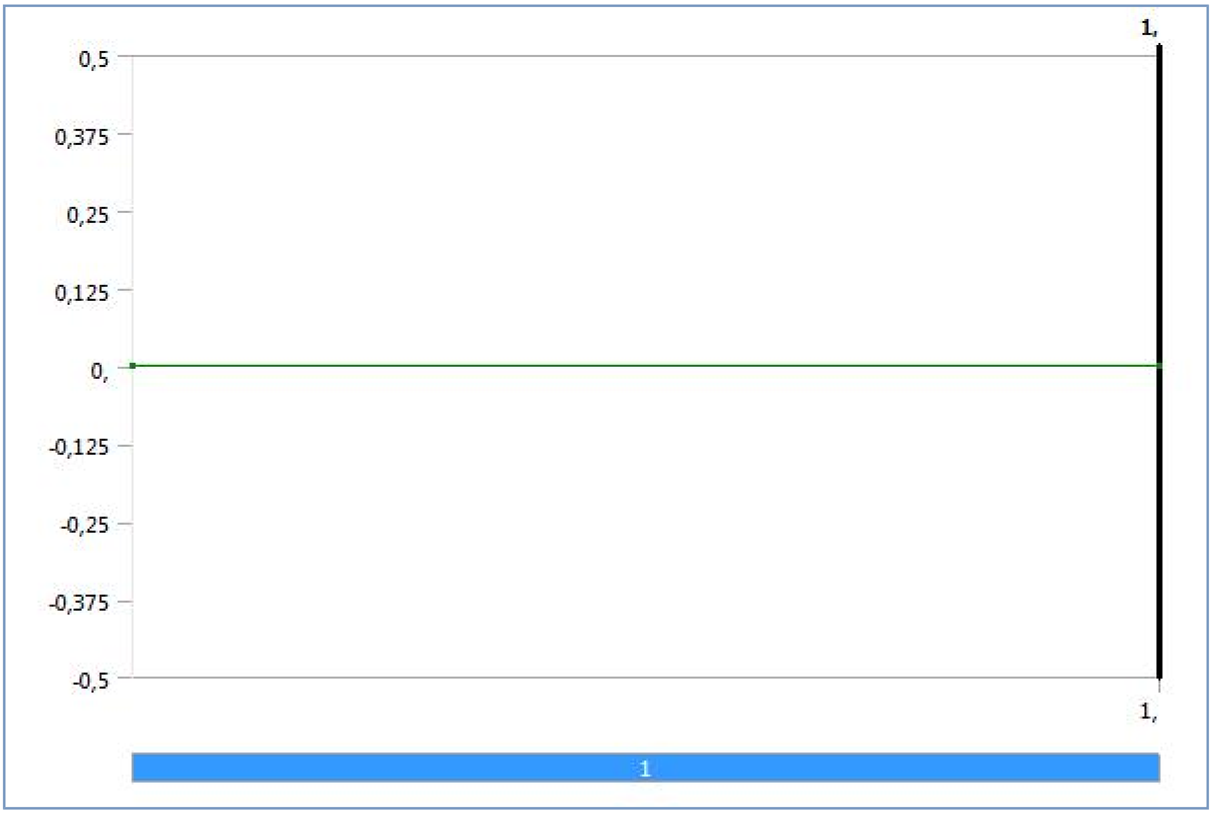
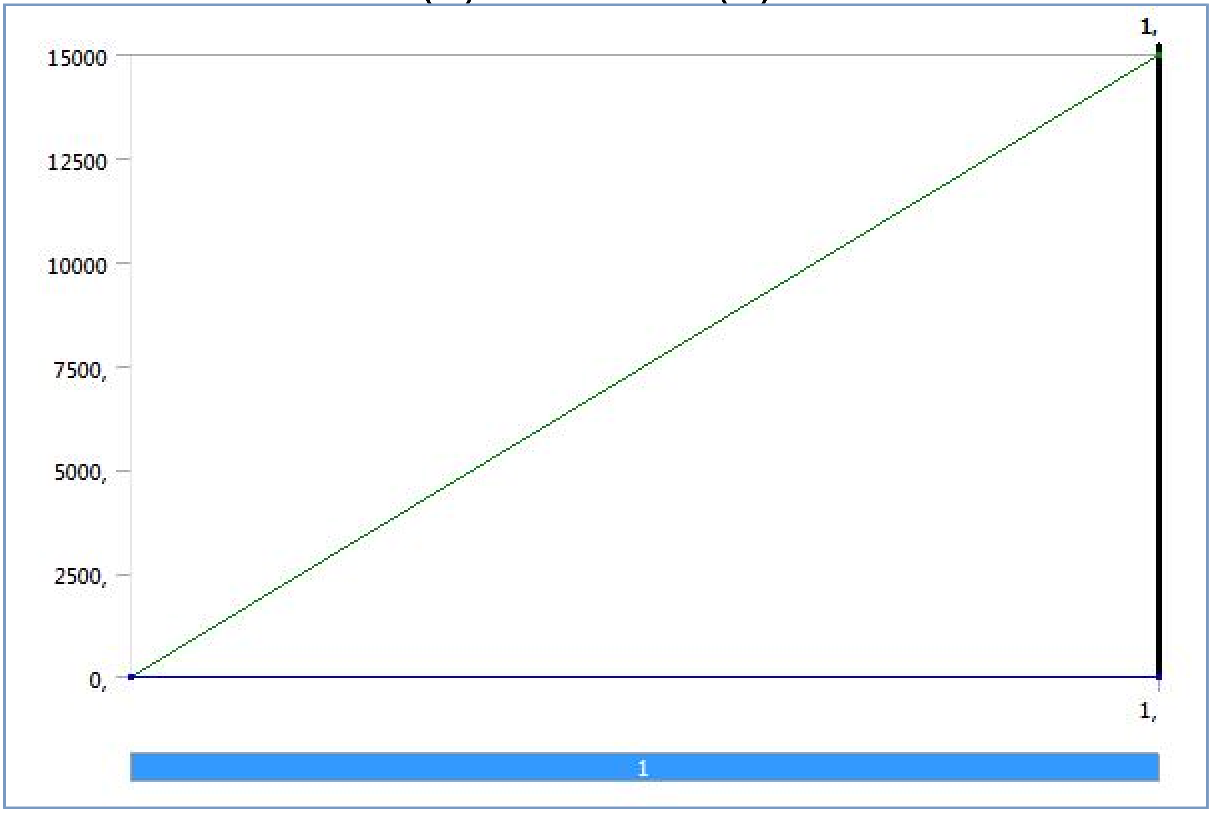


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 17
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved

Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 18
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 19
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Total Deformation*	*Linearized Equivalent Stress*	*Linearized Equivalent Stress 2*	*Linearized Equivalent Stress 3*	*Linearized Equivalent Stress 4*
State	Solved				
Scope					
Scoping Method	Geometry Selection	Path			
Geometry	All Bodies				
Path		Path	Path 2	Path 3	Path 4
Definition					
Type	Total Deformation	Linearized Equivalent Stress			
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Subtype		All			
Coordinate System		Global Coordinate System			
2D Behavior		Planar			
Results					
Minimum	1,701e-004 mm				
Maximum	0,20403 mm				
Minimum Occurs On	Main shell:1				
Maximum Occurs On	End cap:1				
Membrane		68,062 MPa	41,483 MPa	61,936 MPa	57,628 MPa
Bending (Inside)		26,567 MPa	50,299 MPa	31,7 MPa	81,688 MPa
Bending (Outside)		26,567 MPa	50,299 MPa	31,7 MPa	81,688 MPa
Membrane+Bending (Inside)		42,287 MPa	13,035 MPa	93,633 MPa	24,652 MPa
Membrane+Bending (Center)		68,062 MPa	41,483 MPa	61,936 MPa	57,628 MPa
Membrane+Bending (Outside)		94,278 MPa	91,279 MPa	30,247 MPa	139,21 MPa
Peak (Inside)		1,1536e-002 MPa	2,1841e-002 MPa	1,3542 MPa	16,274 MPa
Peak (Center)		3,1192e-011 MPa	1,5562e-010 MPa	1,378 MPa	10,024 MPa
Peak (Outside)		1,1536e-002 MPa	2,1841e-002 MPa	1,4278 MPa	55,146 MPa

Total (Inside)		42,276 MPa	13,051 MPa	92,308 MPa	8,4686 MPa
Total (Center)		68,062 MPa	41,483 MPa	63,284 MPa	49,862 MPa
Total (Outside)		94,29 MPa	91,3 MPa	28,847 MPa	187,71 MPa
Information					
Time	1, s				
Load Step	1				
Substep	1				
Iteration Number	1				

FIGURE 5
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress

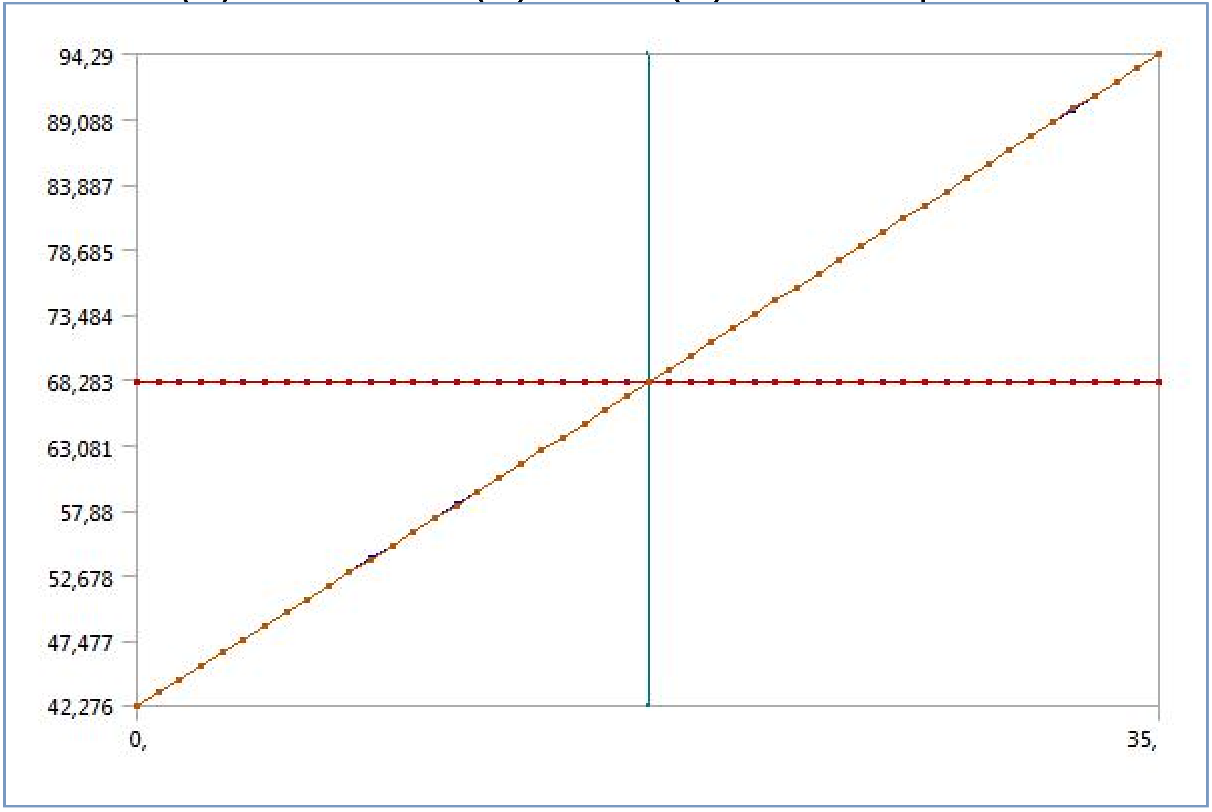


TABLE 20
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		26,567	42,287	1,1536e-002	42,276
0,72917		25,46	43,342	1,1055e-002	43,331
1,4583		24,353	44,399	1,0575e-002	44,389
2,1875		23,246	45,459	1,0094e-002	45,449
2,9167		22,139	46,521	9,6133e-003	46,511
3,6458		21,032	47,585	9,1326e-003	47,576
4,375		19,926	48,651	8,652e-003	48,642
5,1042		18,819	49,718	8,1713e-003	49,711
5,8333		17,712	50,788	7,6906e-003	50,78
6,5625		16,605	51,859	7,21e-003	51,852
7,2917		15,498	52,931	6,7293e-003	52,925
8,0208		14,391	54,005	6,2487e-003	53,999
8,75		13,284	55,08	5,768e-003	55,075
9,4792		12,177	56,157	5,2873e-003	56,152
10,208		11,07	57,234	4,8067e-003	57,23
10,937		9,9628	58,313	4,326e-003	58,309
11,667		8,8558	59,393	3,8453e-003	59,389
12,396		7,7488	60,473	3,3647e-003	60,47

13,125	68,062	6,6418	61,555	2,884e-003	61,552
13,854		5,5349	62,638	2,4033e-003	62,635
14,583		4,4279	63,721	1,9227e-003	63,719
15,313		3,3209	64,805	1,442e-003	64,804
16,042		2,2139	65,89	9,6133e-004	65,889
16,771		1,107	66,976	4,8067e-004	66,975
17,5		2,3098e-014	68,062	2,7888e-011	68,062
18,229		1,107	69,149	4,8067e-004	69,15
18,958		2,2139	70,237	9,6133e-004	70,238
19,688		3,3209	71,325	1,442e-003	71,326
20,417		4,4279	72,414	1,9227e-003	72,416
21,146		5,5349	73,503	2,4033e-003	73,505
21,875		6,6418	74,593	2,884e-003	74,596
22,604		7,7488	75,683	3,3647e-003	75,686
23,333		8,8558	76,774	3,8453e-003	76,778
24,063		9,9628	77,865	4,326e-003	77,869
24,792		11,07	78,957	4,8067e-003	78,962
25,521		12,177	80,049	5,2873e-003	80,054
26,25		13,284	81,142	5,768e-003	81,147
26,979		14,391	82,234	6,2487e-003	82,241
27,708		15,498	83,328	6,7293e-003	83,334
28,438		16,605	84,421	7,21e-003	84,429
29,167		17,712	85,515	7,6906e-003	85,523
29,896		18,819	86,61	8,1713e-003	86,618
30,625		19,926	87,704	8,652e-003	87,713
31,354		21,032	88,799	9,1326e-003	88,808
32,083		22,139	89,894	9,6133e-003	89,904
32,813		23,246	90,99	1,0094e-002	91,
33,542		24,353	92,086	1,0575e-002	92,096
34,271		25,46	93,182	1,1055e-002	93,193
35,		26,567	94,278	1,1536e-002	94,29

FIGURE 6
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 2

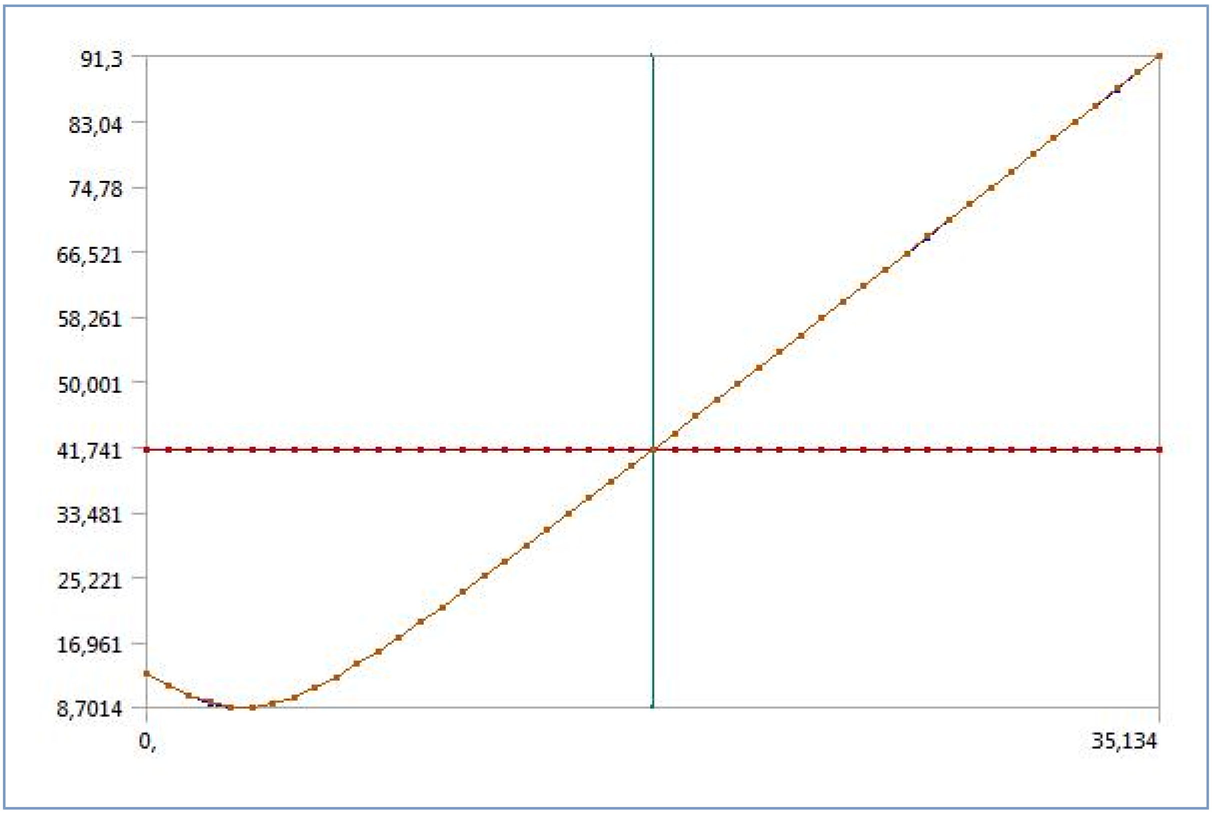


TABLE 21
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 2

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	41,483	50,299	13,035	2,1841e-002	13,051
0,73196		48,204	11,555	2,0931e-002	11,568
1,4639		46,108	10,29	2,0021e-002	10,301
2,1959		44,012	9,3303	1,9111e-002	9,3373
2,9278		41,916	8,7752	1,8201e-002	8,778
3,6598		39,82	8,7028	1,7291e-002	8,7014
4,3918		37,725	9,1246	1,6381e-002	9,1195
5,1237		35,629	9,9782	1,5471e-002	9,9705
5,8557		33,533	11,165	1,4561e-002	11,156
6,5876		31,437	12,591	1,365e-002	12,581
7,3196		29,341	14,184	1,274e-002	14,174
8,0515		27,245	15,894	1,183e-002	15,884
8,7835		25,15	17,688	1,092e-002	17,678
9,5155		23,054	19,542	1,001e-002	19,533
10,247		20,958	21,44	9,1003e-003	21,432
10,979		18,862	23,372	8,1903e-003	23,364
11,711		16,766	25,33	7,2803e-003	25,323
12,443		14,671	27,309	6,3702e-003	27,303
13,175		12,575	29,304	5,4602e-003	29,299
13,907		10,479	31,312	4,5502e-003	31,308
14,639		8,3832	33,331	3,6401e-003	33,327
15,371		6,2874	35,359	2,7301e-003	35,356
16,103		4,1916	37,394	1,8201e-003	37,392
16,835		2,0958	39,436	9,1003e-004	39,435
17,567		2,0746e-014	41,483	9,4981e-010	41,483
18,299		2,0958	43,534	9,1003e-004	43,535
19,031		4,1916	45,59	1,8201e-003	45,592
19,763		6,2874	47,649	2,7301e-003	47,652
20,495		8,3832	49,712	3,6401e-003	49,715

21,227		10,479	51,777	4,5502e-003	51,781
21,959		12,575	53,844	5,4602e-003	53,849
22,691		14,671	55,913	6,3702e-003	55,92
23,423		16,766	57,985	7,2803e-003	57,992
24,155		18,862	60,058	8,1903e-003	60,066
24,887		20,958	62,132	9,1003e-003	62,141
25,619		23,054	64,208	1,001e-002	64,218
26,351		25,15	66,286	1,092e-002	66,296
27,082		27,245	68,364	1,183e-002	68,376
27,814		29,341	70,443	1,274e-002	70,456
28,546		31,437	72,524	1,365e-002	72,537
29,278		33,533	74,605	1,4561e-002	74,619
30,01		35,629	76,687	1,5471e-002	76,702
30,742		37,725	78,77	1,6381e-002	78,786
31,474		39,82	80,853	1,7291e-002	80,87
32,206		41,916	82,937	1,8201e-002	82,955
32,938		44,012	85,022	1,9111e-002	85,041
33,67		46,108	87,107	2,0021e-002	87,127
34,402		48,204	89,192	2,0931e-002	89,213
35,134		50,299	91,279	2,1841e-002	91,3

FIGURE 7
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 3

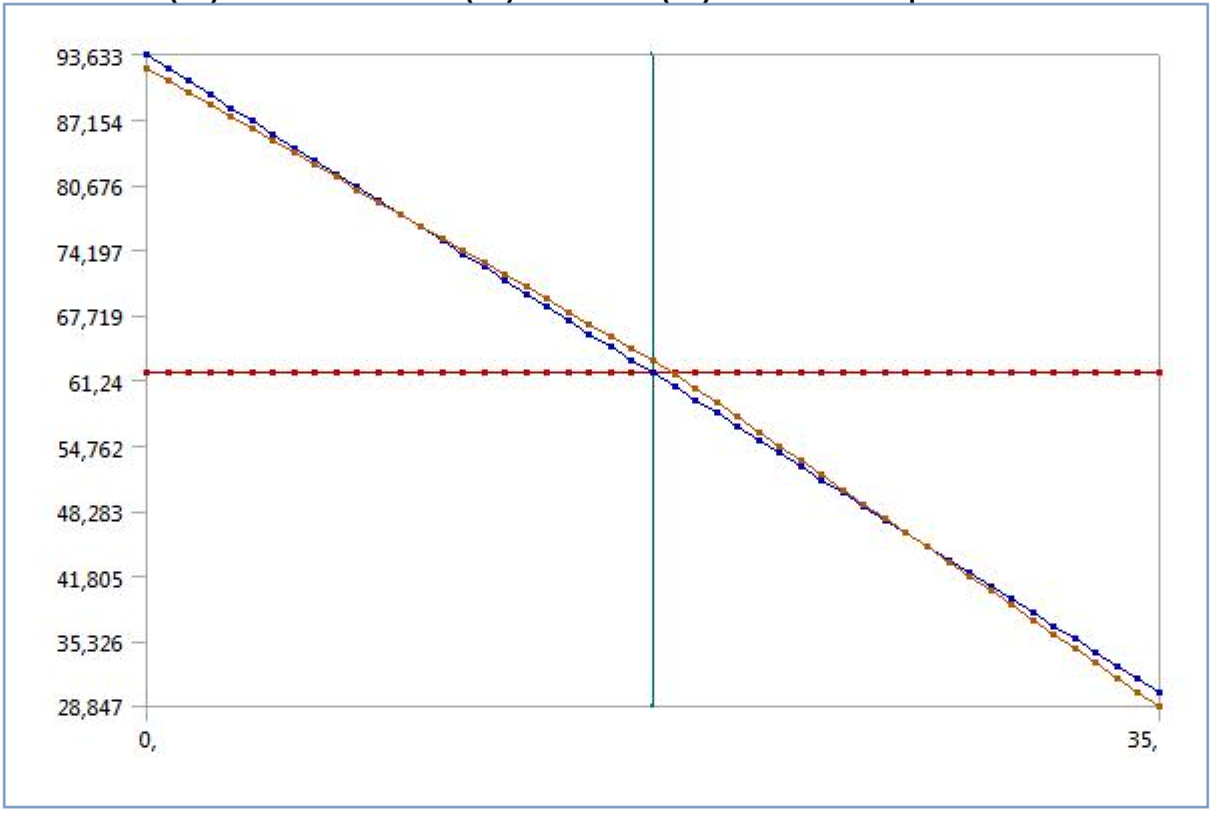


TABLE 22
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 3

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		31,7	93,633	1,3542	92,308
0,72917		30,38	92,312	1,3558	91,098
1,4583		29,059	90,991	1,3573	89,887
2,1875		27,738	89,671	1,3588	88,676
2,9167		26,417	88,35	1,3604	87,465
3,6458		25,096	87,029	1,3619	86,255

4,375	61,936	23,775	85,709	1,3634	85,044
5,1042		22,455	84,388	1,365	83,833
5,8333		21,134	83,067	1,3665	82,623
6,5625		19,813	81,746	1,368	81,412
7,2917		18,492	80,426	1,3696	80,201
8,0208		17,171	79,105	1,3711	78,991
8,75		15,85	77,784	1,3726	77,78
9,4792		14,529	76,464	1,3742	76,569
10,208		13,209	75,143	1,3757	75,359
10,938		11,888	73,822	1,3772	74,148
11,667		10,567	72,501	1,3788	72,938
12,396		9,246	71,181	1,3803	71,727
13,125		7,9251	69,86	1,3818	70,517
13,854		6,6043	68,539	1,3834	69,306
14,583		5,2834	67,219	1,3849	68,096
15,313		3,9626	65,898	1,3864	66,885
16,042		2,6417	64,577	1,388	65,675
16,771		1,3209	63,257	1,3895	64,464
17,5		2,0623e-013	61,936	1,391	63,284
18,229		1,3209	60,616	1,3925	61,908
18,958		2,6417	59,295	1,3941	60,47
19,687		3,9626	57,974	1,3956	59,033
20,417		5,2834	56,654	1,3971	57,595
21,146		6,6043	55,333	1,3987	56,157
21,875		7,9251	54,013	1,4002	54,72
22,604		9,246	52,692	1,4017	53,282
23,333		10,567	51,372	1,4033	51,845
24,062		11,888	50,051	1,4048	50,407
24,792		13,209	48,731	1,4063	48,97
25,521		14,529	47,41	1,4079	47,532
26,25		15,85	46,09	1,4094	46,095
26,979		17,171	44,769	1,4109	44,657
27,708		18,492	43,449	1,4125	43,22
28,437		19,813	42,129	1,414	41,782
29,167		21,134	40,808	1,4155	40,345
29,896		22,455	39,488	1,4171	38,908
30,625		23,775	38,168	1,4186	37,47
31,354		25,096	36,847	1,4201	36,033
32,083		26,417	35,527	1,4217	34,596
32,812		27,738	34,207	1,4232	33,159
33,542		29,059	32,887	1,4247	31,721
34,271		30,38	31,567	1,4263	30,284
35,		31,7	30,247	1,4278	28,847

FIGURE 8
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 4

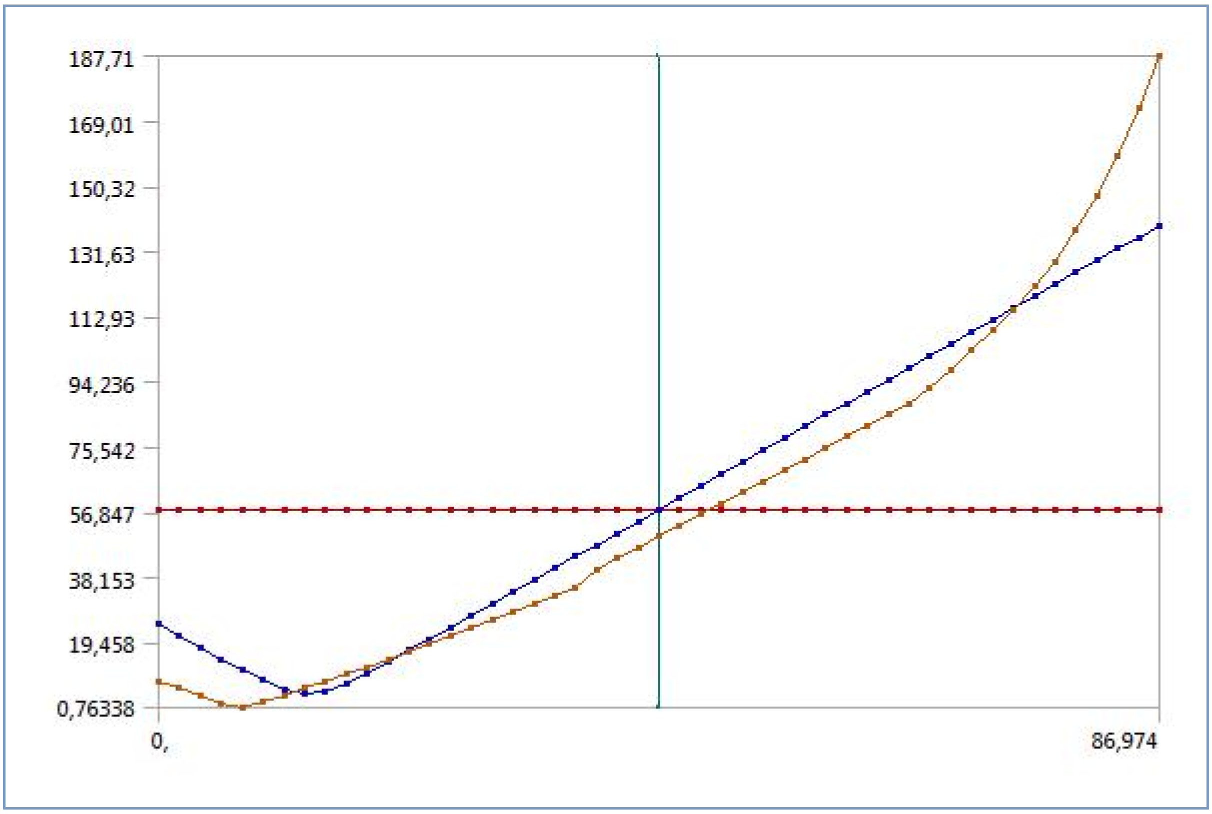


TABLE 23
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 4

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	57,628	81,688	24,652	16,274	8,4686
1,812		78,284	21,315	16,996	6,2434
3,6239		74,88	18,003	17,729	4,0828
5,4359		71,477	14,732	18,472	2,0183
7,2478		68,073	11,539	19,224	0,76338
9,0598		64,669	8,5104	19,984	2,3848
10,872		61,266	5,9057	20,75	4,3563
12,684		57,862	4,5275	21,523	6,3359
14,496		54,458	5,4119	22,301	8,3053
16,308		51,055	7,8261	23,084	10,265
18,12		47,651	10,788	23,871	12,358
19,932		44,247	13,953	24,662	14,582
21,744		40,844	17,209	25,458	16,887
23,555		37,44	20,513	26,256	19,19
25,367		34,037	23,845	27,058	21,475
27,179		30,633	27,195	27,862	23,735
28,991		27,229	30,557	28,669	25,965
30,803		23,826	33,927	29,478	28,161
32,615		20,422	37,303	30,289	30,321
34,427		17,018	40,684	31,102	32,755
36,239		13,615	44,068	31,917	35,371
38,051		10,211	47,455	32,734	40,407
39,863		6,8073	50,845	33,552	43,55
41,675		3,4037	54,236	34,372	46,703
43,487		0,	57,628	35,193	49,862
45,299		3,4037	61,022	36,015	53,028
47,111		6,8073	64,417	36,839	56,198
48,923		10,211	67,813	37,664	59,372
50,735		13,615	71,209	38,489	62,55

52,547		17,018	74,606	39,316	65,731
54,359		20,422	78,004	40,143	68,914
56,171		23,826	81,402	40,972	72,1
57,983		27,229	84,801	41,801	75,287
59,795		30,633	88,2	42,631	78,582
61,607		34,037	91,599	43,461	81,664
63,419		37,44	94,999	44,293	84,828
65,231		40,844	98,399	45,125	87,99
67,043		44,247	101,8	45,957	92,737
68,855		47,651	105,2	46,79	97,926
70,666		51,055	108,6	47,624	103,26
72,478		54,458	112,	48,458	108,93
74,29		57,862	115,4	49,293	115,07
76,102		61,266	118,8	50,128	121,58
77,914		64,669	122,2	50,963	128,44
79,726		68,073	125,61	51,799	138,02
81,538		71,477	129,01	52,635	147,79
83,35		74,88	132,41	53,472	159,32
85,162		78,284	135,81	54,309	172,79
86,974		81,688	139,21	55,146	187,71

TABLE 24
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Linearized Maximum Principal Stress*	*Linearized Middle Principal Stress*	*Linearized Minimum Principal Stress*
State	Solved		
Scope			
Scoping Method	Path		
Path	Path 4		
Geometry	All Bodies		
Definition			
Type	Linearized Maximum Principal Stress	Linearized Middle Principal Stress	Linearized Minimum Principal Stress
Subtype	All		
By	Time		
Display Time	Last		
Coordinate System	Global Coordinate System		
2D Behavior	Planar		
Results			
Membrane	62,01 MPa	14,6 MPa	-2,1323 MPa
Bending (Inside)	3,6573 MPa	-13,72 MPa	-85,321 MPa
Bending (Outside)	85,321 MPa	13,72 MPa	-3,6573 MPa
Membrane+Bending (Inside)	2,698 MPa	-0,29133 MPa	-23,312 MPa
Membrane+Bending (Center)	62,01 MPa	14,6 MPa	-2,1323 MPa
Membrane+Bending (Outside)	147,33 MPa	28,258 MPa	-5,7269 MPa
Peak (Inside)	15,202 MPa	0,4948 MPa	-2,2816 MPa
Peak (Center)	4,2374 MPa	9,3201e-002 MPa	-7,1938 MPa
Peak (Outside)	38,539 MPa	0,13115 MPa	-24,651 MPa
Total (Inside)	0,77505 MPa	-0,14804 MPa	-8,1173 MPa
Total (Center)	54,852 MPa	18,819 MPa	-2,0554 MPa
Total (Outside)	185,85 MPa	9,4237 MPa	-11,397 MPa
Information			
Time	1, s		
Load Step	1		

Substep	1
Iteration Number	1

FIGURE 9

Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Maximum Principal Stress

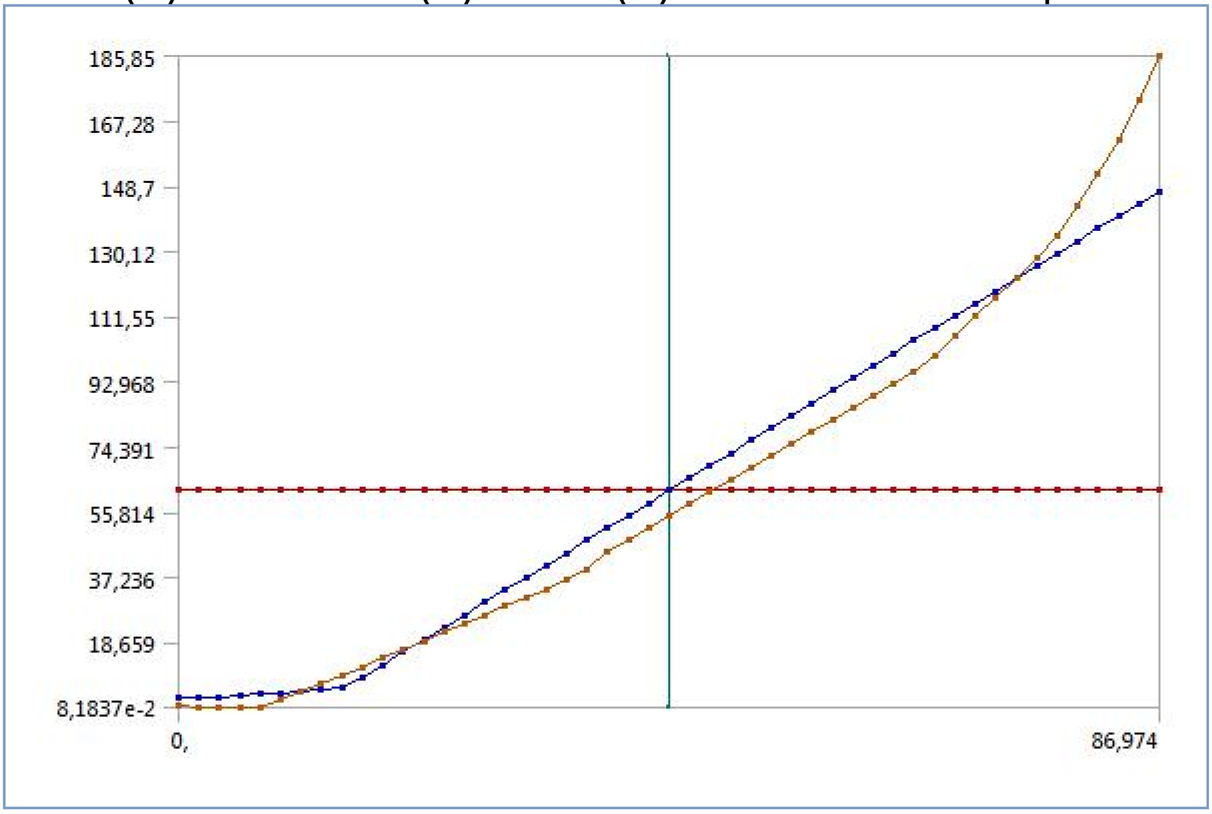


TABLE 25

Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Maximum Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		3,6573	2,698	15,202	0,77505
1,812		3,5049	2,8423	15,688	0,31863
3,6239		3,3525	3,0766	16,175	0,11437
5,4359		3,2002	3,3968	16,661	8,1837e-002
7,2478		3,0478	3,7865	17,147	0,15935
9,0598		2,8954	4,2271	17,633	2,1227
10,872		2,743	4,7031	18,119	4,402
12,684		2,5906	5,2044	18,605	6,7594
14,496		2,4382	5,7427	19,092	9,1951
16,308		2,2858	8,6929	19,578	11,709
18,12		2,1334	12,244	20,064	14,132
19,932		1,981	15,797	20,55	16,589
21,744		1,8287	19,352	21,036	19,086
23,555		1,6763	22,906	21,522	21,574
25,367		1,5239	26,461	22,009	24,052
27,179		1,3715	30,016	22,495	26,521
28,991		1,2191	33,571	22,981	28,979
30,803		1,0667	37,126	23,467	31,428
32,615		0,91433	40,681	23,953	33,866
34,427		0,76194	44,236	24,44	36,582
36,239		0,60955	47,79	24,926	39,501
38,051		0,45717	51,345	25,412	44,666
39,863		0,30478	54,9	25,898	48,06
41,675		0,15239	58,455	26,384	51,455

43,487	62,01	0,	62,01	26,87	54,852
45,299		3,555	65,565	27,357	58,249
47,111		7,1101	69,12	27,843	61,647
48,923		10,665	72,675	28,329	65,046
50,735		14,22	76,23	28,815	68,446
52,547		17,775	79,785	29,301	71,846
54,359		21,33	83,34	29,788	75,246
56,171		24,885	86,895	30,274	78,647
57,983		28,44	90,45	30,76	82,047
59,795		31,995	94,006	31,246	85,502
61,607		35,55	97,561	31,732	88,848
63,419		39,105	101,12	32,219	92,277
65,231		42,66	104,67	32,705	95,703
67,043		46,215	108,23	33,191	100,48
68,855		49,77	111,78	33,677	106,09
70,666		53,325	115,34	34,163	111,69
72,478		56,881	118,89	34,65	117,04
74,29		60,436	122,45	35,136	122,47
76,102		63,991	126,	35,622	128,23
77,914		67,546	129,56	36,108	134,34
79,726		71,101	133,11	36,594	143,3
81,538		74,656	136,67	37,081	152,27
83,35		78,211	140,22	37,567	162,2
85,162		81,766	143,78	38,053	173,53
86,974		85,321	147,33	38,539	185,85

FIGURE 10
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Middle Principal Stress

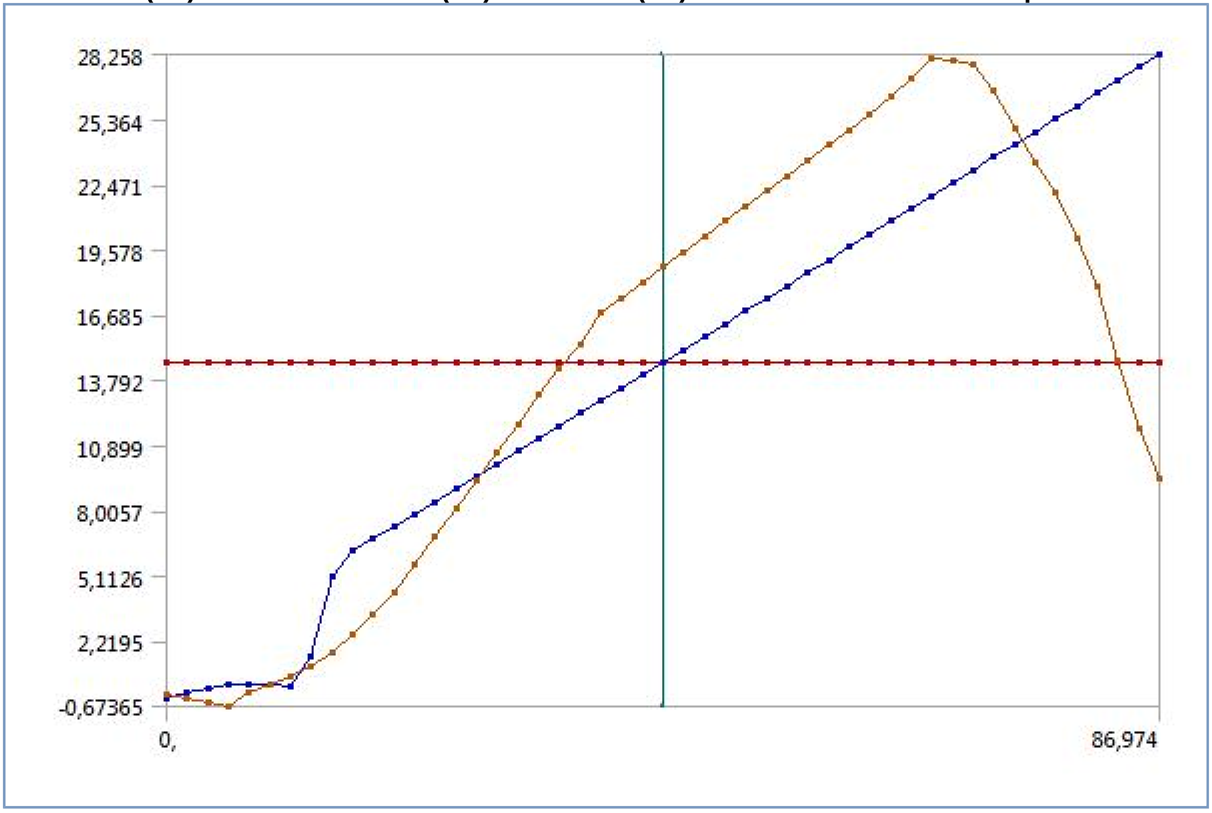


TABLE 26
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Middle Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		-13,72	-0,29133	0,4948	-0,14804

1,812	14,6	-13,148	-1,6142e-002	0,28501	-0,32833
3,6239		-12,577	0,1691	0,11951	-0,53031
5,4359		-12,005	0,26868	-7,6122e-003	-0,67365
7,2478		-11,433	0,29891	-0,10362	-8,2437e-002
9,0598		-10,862	0,27909	-0,17541	0,327
10,872		-10,29	0,22768	-0,22872	0,61593
12,684		-9,7184	1,581	-0,268	1,0627
14,496		-9,1468	5,1057	-0,29659	1,7054
16,308		-8,5751	6,237	-0,31695	2,5565
18,12		-8,0034	6,777	-0,33091	3,3765
19,932		-7,4317	7,3209	-0,33982	4,4074
21,744		-6,8601	7,869	-0,3447	5,6289
23,555		-6,2884	8,4206	-0,34632	6,8638
25,367		-5,7167	8,9751	-0,34527	8,1068
27,179		-5,1451	9,5319	-0,34203	9,3556
28,991		-4,5734	10,091	-0,33696	10,61
30,803		-4,0017	10,651	-0,33034	11,869
32,615		-3,43	11,212	-0,3224	13,132
34,427		-2,8584	11,775	-0,31335	14,295
36,239		-2,2867	12,339	-0,30333	15,35
38,051		-1,715	12,903	-0,29247	16,778
39,863		-1,1433	13,468	-0,28088	17,459
41,675		-0,57167	14,034	-0,26865	18,139
43,487		0,	14,6	-0,25586	18,819
45,299		0,57167	15,166	-0,24256	19,497
47,111		1,1433	15,733	-0,22882	20,175
48,923		1,715	16,301	-0,21468	20,852
50,735		2,2867	16,868	-0,20018	21,53
52,547		2,8584	17,436	-0,18536	22,206
54,359		3,43	18,005	-0,17025	22,883
56,171		4,0017	18,573	-0,15487	23,559
57,983		4,5734	19,142	-0,13926	24,235
59,795		5,1451	19,711	-0,12343	24,844
61,607		5,7167	20,28	-0,1074	25,589
63,419		6,2884	20,849	-9,1181e-002	26,382
65,231		6,8601	21,418	-7,4798e-002	27,172
67,043		7,4317	21,988	-5,8261e-002	28,071
68,855		8,0034	22,557	-4,158e-002	27,983
70,666		8,5751	23,127	-2,4767e-002	27,807
72,478		9,1468	23,697	-7,8319e-003	26,666
74,29		9,7184	24,266	9,2169e-003	24,962
76,102		10,29	24,836	2,6371e-002	23,451
77,914		10,862	25,406	4,3624e-002	22,129
79,726		11,433	25,976	6,0969e-002	20,066
81,538		12,005	26,547	7,8399e-002	17,932
83,35		12,577	27,117	9,591e-002	14,714
85,162		13,148	27,687	0,1135	11,706
86,974		13,72	28,258	0,13115	9,4237

FIGURE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Minimum Principal Stress

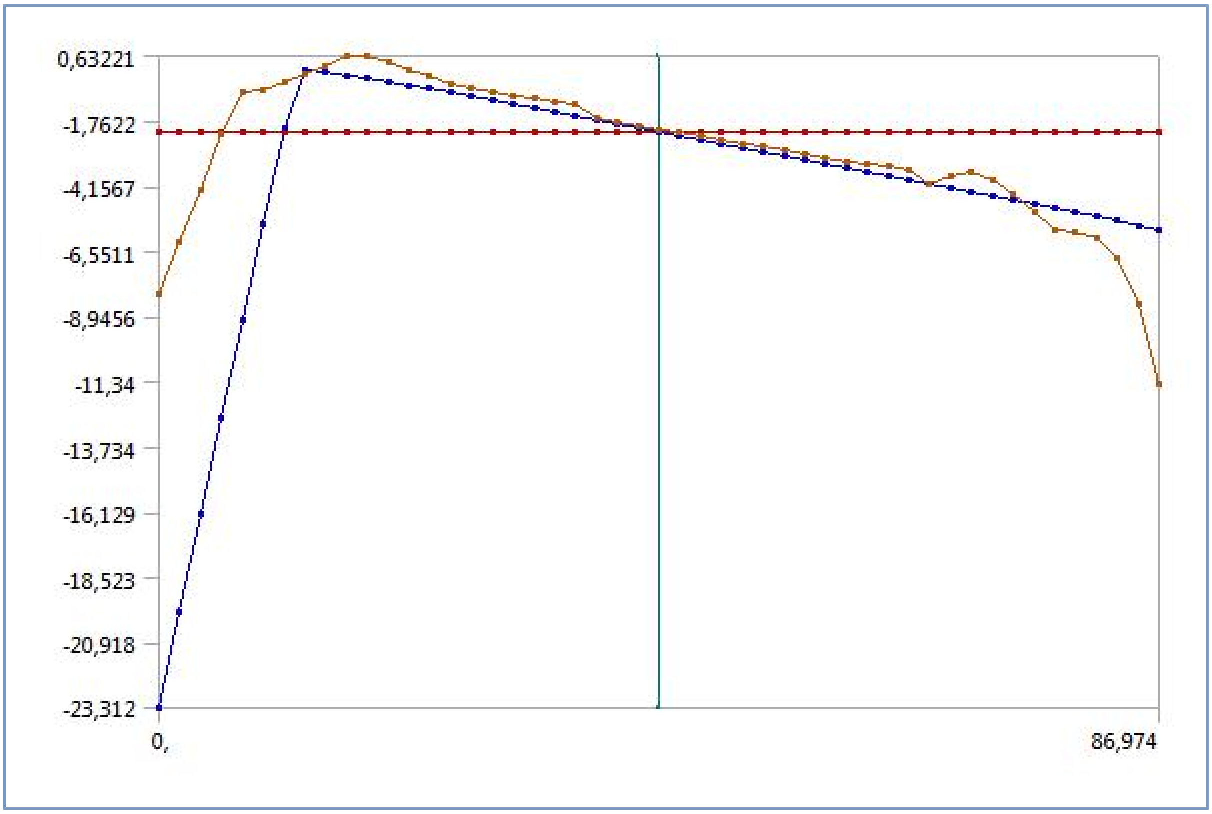


TABLE 27
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Minimum Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	-2,1323	-85,321	-23,312	-2,2816	-8,1173
1,812		-81,766	-19,757	-2,5454	-6,2231
3,6239		-78,211	-16,203	-2,8535	-4,2524
5,4359		-74,656	-12,648	-3,1999	-2,2052
7,2478		-71,101	-9,0938	-3,5775	-0,69564
9,0598		-67,546	-5,5403	-3,9793	-0,58317
10,872		-63,991	-1,9905	-4,3995	-0,35925
12,684		-60,436	0,12915	-4,8338	-6,4398e-002
14,496		-56,881	4,0526e-002	-5,2788	0,26324
16,308		-53,325	-6,6639e-002	-5,7321	0,61083
18,12		-49,77	-0,18318	-6,1917	0,63221
19,932		-46,215	-0,30634	-6,6564	0,43249
21,744		-42,66	-0,43447	-7,1251	0,13628
23,555		-39,105	-0,5664	-7,5971	-0,13239
25,367		-35,55	-0,70132	-8,0717	-0,36707
27,179		-31,995	-0,83864	-8,5485	-0,56531
28,991		-28,44	-0,97789	-9,0272	-0,72608
30,803		-24,885	-1,1187	-9,5074	-0,84899
32,615		-21,33	-1,2609	-9,989	-0,93396
34,427		-17,775	-1,4042	-10,472	-1,0257
36,239		-14,22	-1,5484	-10,955	-1,0997
38,051		-10,665	-1,6934	-11,44	-1,6721
39,863		-7,1101	-1,8391	-11,925	-1,7996
41,675		-3,555	-1,9854	-12,411	-1,9274
43,487		0,	-2,1323	-12,897	-2,0554
45,299		-0,15239	-2,2796	-13,384	-2,1837
47,111		-0,30478	-2,4272	-13,871	-2,3121
48,923		-0,45717	-2,5753	-14,359	-2,4407
50,735		-0,60955	-2,7237	-14,847	-2,5695

52,547		-0,76194	-2,8723	-15,336	-2,6985
54,359		-0,91433	-3,0212	-15,824	-2,8275
56,171		-1,0667	-3,1704	-16,313	-2,9567
57,983		-1,2191	-3,3197	-16,803	-3,0859
59,795		-1,3715	-3,4693	-17,292	-3,2708
61,607		-1,5239	-3,619	-17,782	-3,343
63,419		-1,6763	-3,7688	-18,271	-3,4332
65,231		-1,8287	-3,9189	-18,761	-3,523
67,043		-1,981	-4,069	-19,252	-4,0406
68,855		-2,1334	-4,2193	-19,742	-3,7731
70,666		-2,2858	-4,3696	-20,232	-3,6325
72,478		-2,4382	-4,5201	-20,723	-3,9023
74,29		-2,5906	-4,6707	-21,213	-4,4625
76,102		-2,743	-4,8214	-21,704	-5,0792
77,914		-2,8954	-4,9721	-22,195	-5,7527
79,726		-3,0478	-5,1229	-22,686	-5,8393
81,538		-3,2002	-5,2738	-23,177	-6,0547
83,35		-3,3525	-5,4248	-23,668	-6,7618
85,162		-3,5049	-5,5758	-24,159	-8,4467
86,974		-3,6573	-5,7269	-24,651	-11,397

Material Data

SA-516 grade 70

TABLE 28
SA-516 grade 70 > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 29
SA-516 grade 70 > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 30
SA-516 grade 70 > Compressive Yield Strength

Compressive Yield Strength MPa
260

TABLE 31
SA-516 grade 70 > Tensile Yield Strength

Tensile Yield Strength MPa
260

TABLE 32
SA-516 grade 70 > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
485

TABLE 33
SA-516 grade 70 > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C

22

TABLE 34
SA-516 grade 70 > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 35
SA-516 grade 70 > Isotropic Elasticity

Temperature C	Young’s Modulus MPa	Poisson’s Ratio	Bulk Modulus MPa	Shear Modulus MPa
20	2.e+005	0.3	1.6667e+005	76923

TABLE 36
SA-516 grade 70 > Isotropic Relative Permeability

Relative Permeability
10000

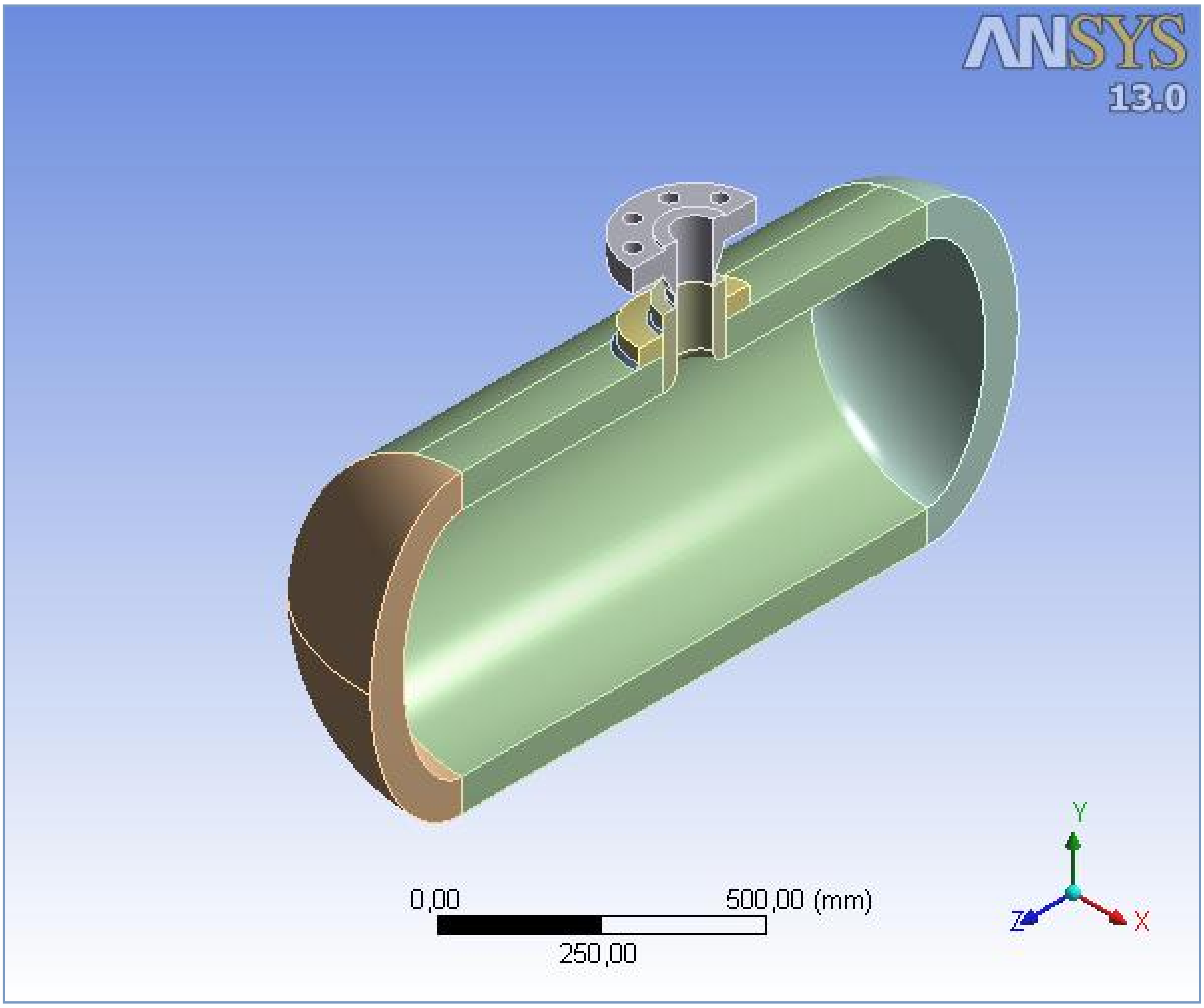
TABLE 37
SA-516 grade 70 > Uniaxial Test Data

Strain mm mm^-1	Stress MPa	Temperature C
0	0	20
2.e-003	260	20
5.e-002	485	20

APPENDIX G: Calculation report from ANSYS Elastic Stress Analysis, ASME VIII div. 2; 2010, Protection against plastic collapse (70 mm @ 200 bar)

Project

First Saved	Tuesday, April 10, 2012
Last Saved	Monday, April 30, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thick wall\Assembly\Weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	318,32 mm
Length Y	836, mm
Length Z	1387, mm
Properties	

Volume	8,98e+007 mm^3
Mass	704,93 kg
Scale Factor Value	1,
Statistics	
Bodies	7
Active Bodies	7
Nodes	38820
Elements	19469
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell thick:1*	*End cap thick:1*	*End cap thick:2*	*Reinforcement pad thick2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127, mm	318, mm	318,32 mm		117,68 mm
Length Y	110,06 mm	636, mm	636,01 mm	636, mm	62,595 mm
Length Z	253,91 mm	1000, mm	193,51 mm		235,31 mm
Properties					
Volume	22864 mm^3	6,1723e+007 mm^3	1,277e+007 mm^3		5,9435e+005 mm^3

Mass	0,17948 kg	484,52 kg	100,24 kg		4,6657 kg
Centroid X	-67,2 mm	-182,3 mm	-135,52 mm	-135,53 mm	-60,897 mm
Centroid Y	325,62 mm	-2,3053 mm	-1,8244e-002 mm	4,1893e-002 mm	330,95 mm
Centroid Z	1,9269e-003 mm	2,5059e-006 mm	588,86 mm	-588,86 mm	-9,7601e-007 mm
Moment of Inertia Ip1	1106,8 kg·mm²	6,024e+007 kg·mm²	2,6829e+006 kg·mm²	2,684e+006 kg·mm²	21904 kg·mm²
Moment of Inertia Ip2	1392,8 kg·mm²	4,4447e+007 kg·mm²	9,059e+005 kg·mm²	9,061e+005 kg·mm²	25881 kg·mm²
Moment of Inertia Ip3	419,67 kg·mm²	2,3283e+007 kg·mm²	3,2488e+006 kg·mm²	3,25e+006 kg·mm²	5250,6 kg·mm²
Statistics					
Nodes	598	16787	5073	4260	1329
Elements	145	9824	2970	789	216
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 1500 RF:1*	*Nozzle thick2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	Structural Steel	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	133,5 mm	67,677 mm
Length Y	124, mm	155,41 mm
Length Z	267, mm	135,33 mm
Properties		
Volume	1,2214e+006 mm³	6,998e+005 mm³
Mass	9,5877 kg	5,4935 kg
Centroid X	-56,09 mm	-35,249 mm
Centroid Y	483,94 mm	320,86 mm
Centroid Z	2,2844e-005 mm	-5,6781e-004 mm
Moment of Inertia Ip1	45407 kg·mm²	18168 kg·mm²
Moment of Inertia Ip2	53311 kg·mm²	10053 kg·mm²
Moment of Inertia Ip3	15378 kg·mm²	11576 kg·mm²
Statistics		
Nodes	8481	2292
Elements	5135	390
Mesh Metric	None	

TABLE 5
Model (A4) > Construction Geometry

Object Name	*Construction Geometry*
State	Fully Defined
Display	
Show Mesh	No

TABLE 6
Model (A4) > Construction Geometry > Paths

Object Name	*Path*	*Path 2*	*Path 3*	*Path 4*
State	Fully Defined			
Definition				
Path Type	Two Points			
Path Coordinate System	Global Coordinate System			
Number of Sampling Points	47,			
Suppressed	No			
Start				
Coordinate System	Global Coordinate System			
Start X Coordinate	-8,8969e-014 mm	5,4757e-014 mm	0, mm	-3,7559e-013 mm
Start Y Coordinate	318, mm	275,14 mm	0, mm	358, mm
Start Z Coordinate	200,69 mm	597,02 mm	693,5 mm	-105,15 mm
Location	Defined			
End				
Coordinate System	Global Coordinate System			
End X Coordinate	-9,6655e-014 mm	-34,138 mm	1,6864e-014 mm	-4,6654e-013 mm
End Y Coordinate	248, mm	235,74 mm	-2,0652e-030 mm	260,52 mm
End Z Coordinate	200,69 mm	531,16 mm	623,5 mm	-42,433 mm
Location	Defined			

Coordinate Systems

TABLE 7
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 8
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 9
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	

Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	4,1262 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 10
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Welds To 3 inch Weldneck Flange 1500 RF:1*	*Bonded - Welds To Nozzle thick2:1*	*Bonded - Main shell thick:1 To End cap thick:1*	*Bonded - Main shell thick:1 To End cap thick:2*	*Bonded - Main shell thick:1 To Reinforcement pad thick2:1*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face		2 Faces
Target	1 Face	2 Faces	1 Face		
Contact Bodies	Welds		Main shell thick:1		
Target Bodies	3 inch Weldneck Flange 1500 RF:1	Nozzle thick2:1	End cap thick:1	End cap thick:2	Reinforcement pad thick2:1
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

TABLE 11
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Main shell thick:1 To Nozzle thick2:1*	*Bonded - Reinforcement pad thick2:1 To Nozzle thick2:1*	*Bonded - 3 inch Weldneck Flange 1500 RF:1 To Nozzle thick2:1*	*Contact Region 11*	*Contact Region 13*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face			3 Faces	2 Faces
Target	1 Face			2 Faces	
Contact	Main shell	Reinforcement pad	3 inch Weldneck	Welds	

Bodies	thick:1	thick2:1	Flange 1500 RF:1		
Target Bodies	Nozzle thick2:1			Main shell thick:1	Reinforcement pad thick2:1
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

Mesh

TABLE 12
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	1,24280 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	

Nodes	38820
Elements	19469
Mesh Metric	None

TABLE 13
Model (A4) > Mesh > Mesh Controls

Object Name	*Automatic Method*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	7 Bodies
Definition	
Suppressed	No
Method	Automatic
Element Midside Nodes	Use Global Setting

Static Structural (A5)

TABLE 14
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 15
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation	

Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thick wall configuration - Elastic Stress Analysis - ASME VIII div.2; 2010_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

TABLE 16
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Standard Earth Gravity*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	-0, mm/s² (ramped)
Y Component	-9806,6 mm/s² (ramped)
Z Component	-0, mm/s² (ramped)
Suppressed	No
Direction	-Y Direction

FIGURE 1
Model (A4) > Static Structural (A5) > Standard Earth Gravity

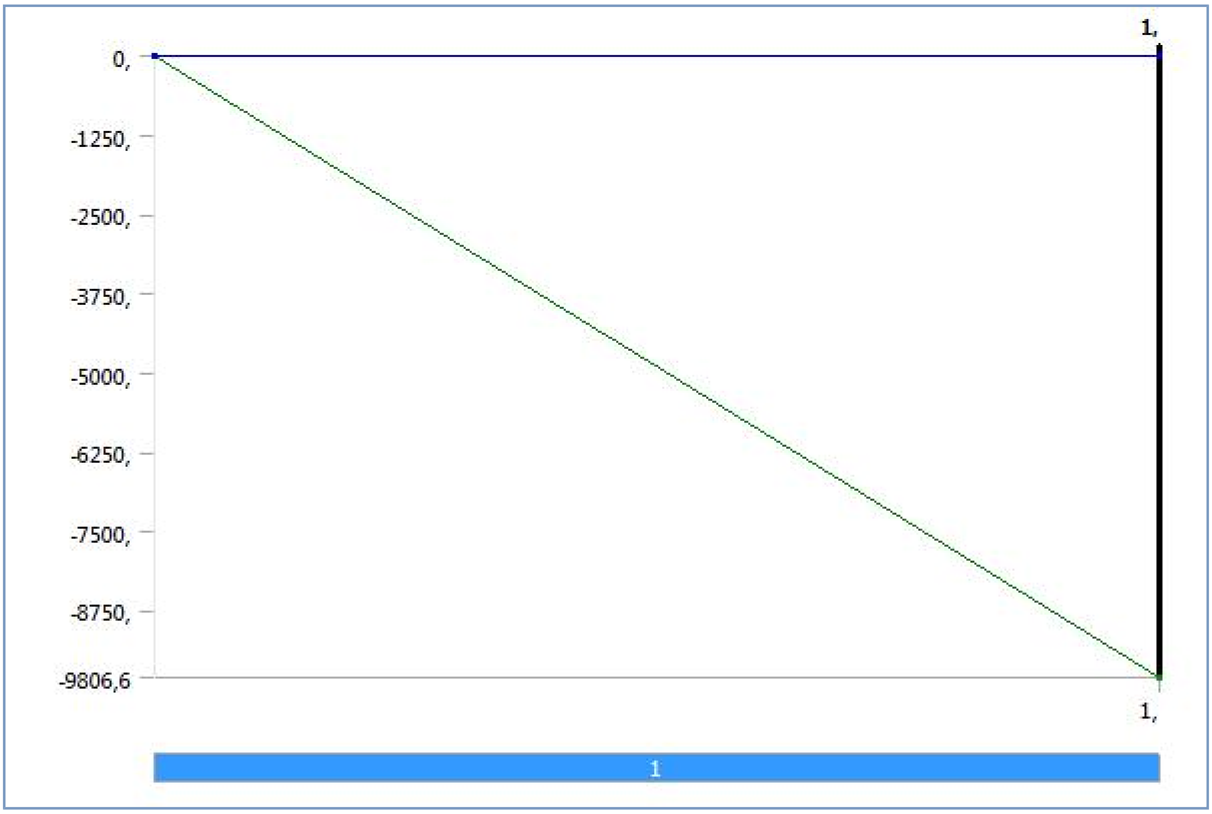


TABLE 17
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	16 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components
Magnitude	20, MPa (ramped)				
Suppressed	No				
Coordinate System			Global Coordinate System		Global Coordinate System
X Component			0, mm (ramped)		0, N (ramped)
Y Component			Free		15000 N (ramped)
Z Component			0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

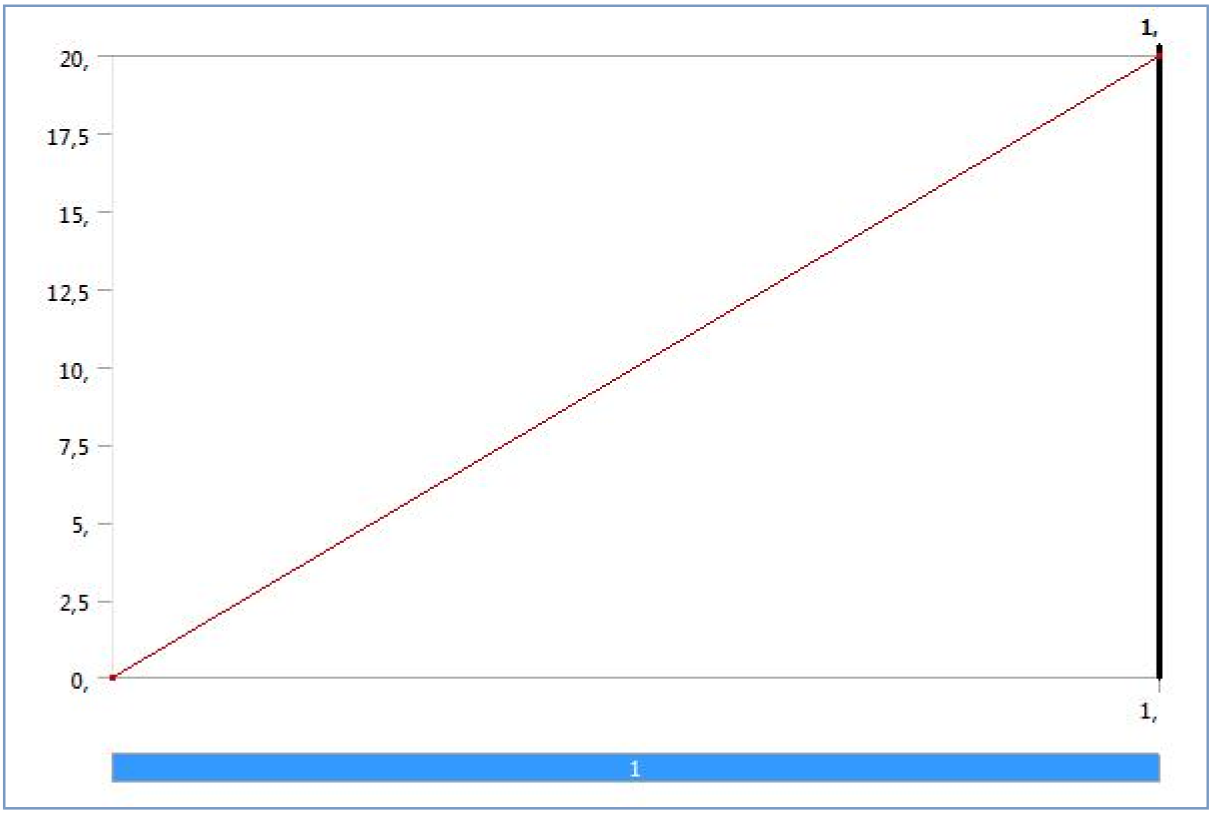


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

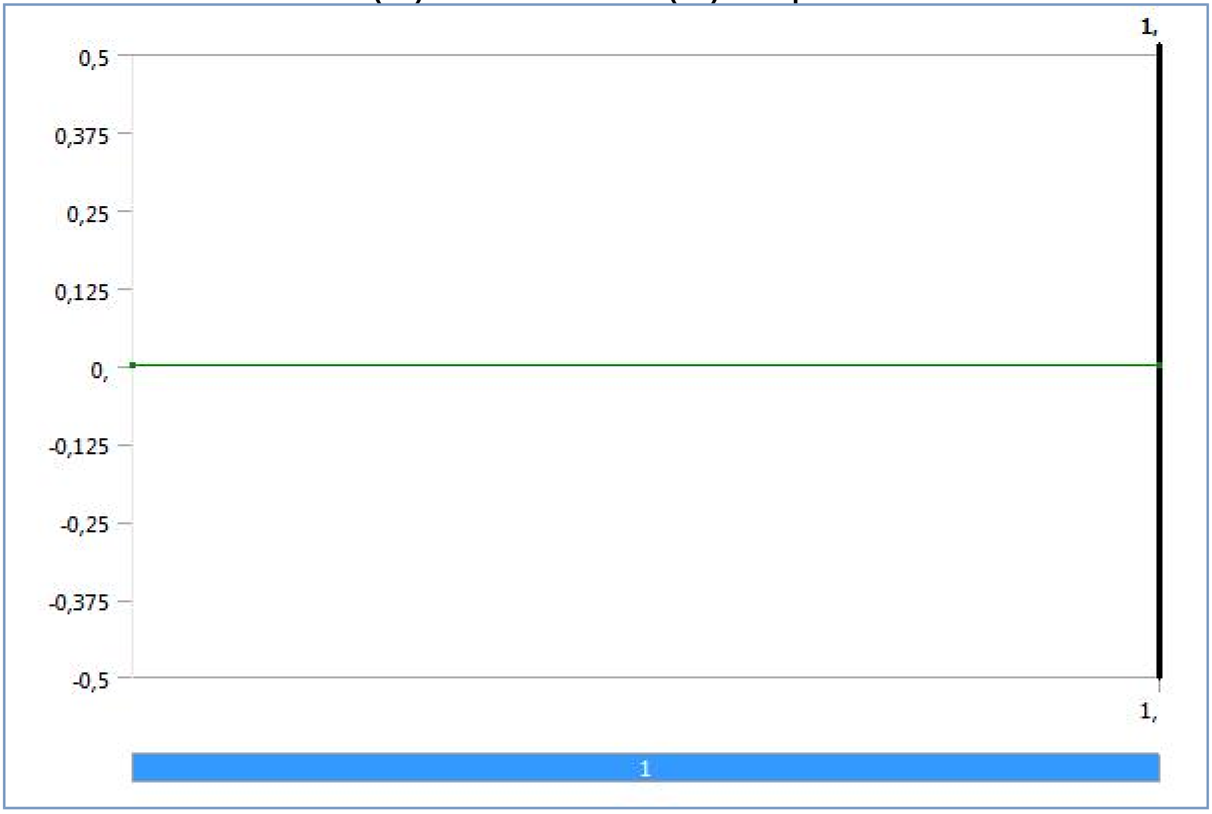
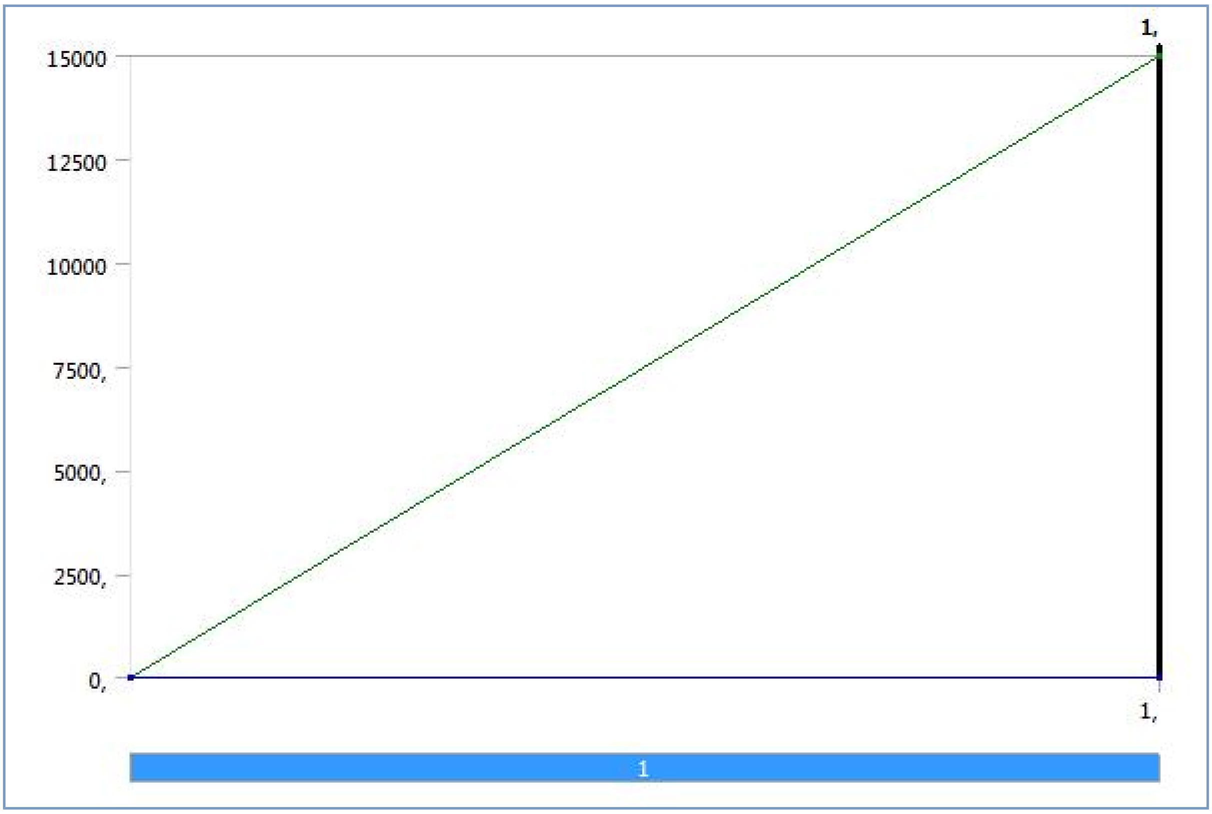


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 18
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 19
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 20
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Total Deformation*	*Linearized Equivalent Stress*	*Linearized Equivalent Stress 2*	*Linearized Equivalent Stress 3*	*Linearized Equivalent Stress 4*
State	Solved				
Scope					
Scoping Method	Geometry Selection	Path			
Geometry	All Bodies				

Path		Path	Path 2	Path 3	Path 4
Definition					
Type	Total Deformation	Linearized Equivalent Stress			
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Subtype		All			
Coordinate System		Global Coordinate System			
2D Behavior		Planar			
Results					
Minimum	1,3797e-004 mm				
Maximum	0,15589 mm				
Minimum Occurs On	Main shell thick:1				
Maximum Occurs On	Main shell thick:1				
Membrane		73,563 MPa	38,414 MPa	52,451 MPa	70,739 MPa
Bending (Inside)		30,617 MPa	52,457 MPa	21,825 MPa	98,758 MPa
Bending (Outside)		30,617 MPa	52,457 MPa	21,825 MPa	98,758 MPa
Membrane+Bending (Inside)		44,274 MPa	15,151 MPa	74,273 MPa	29,187 MPa
Membrane+Bending (Center)		73,563 MPa	38,414 MPa	52,451 MPa	70,739 MPa
Membrane+Bending (Outside)		103,62 MPa	90,693 MPa	30,633 MPa	169,3 MPa
Peak (Inside)		3,3441 MPa	10,689 MPa	1,5507 MPa	27,043 MPa
Peak (Center)		3,3543 MPa	7,3302 MPa	1,5296 MPa	18,322 MPa
Peak (Outside)		3,3645 MPa	18,802 MPa	1,6345 MPa	95,92 MPa
Total (Inside)		46,814 MPa	7,6146 MPa	72,786 MPa	2,738 MPa
Total (Center)		71,021 MPa	31,438 MPa	53,921 MPa	56,08 MPa
Total (Outside)		106,25 MPa	108,89 MPa	29,065 MPa	258,91 MPa
Information					
Time	1, s				
Load Step	1				
Substep	1				
Iteration Number	1				

FIGURE 5
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress

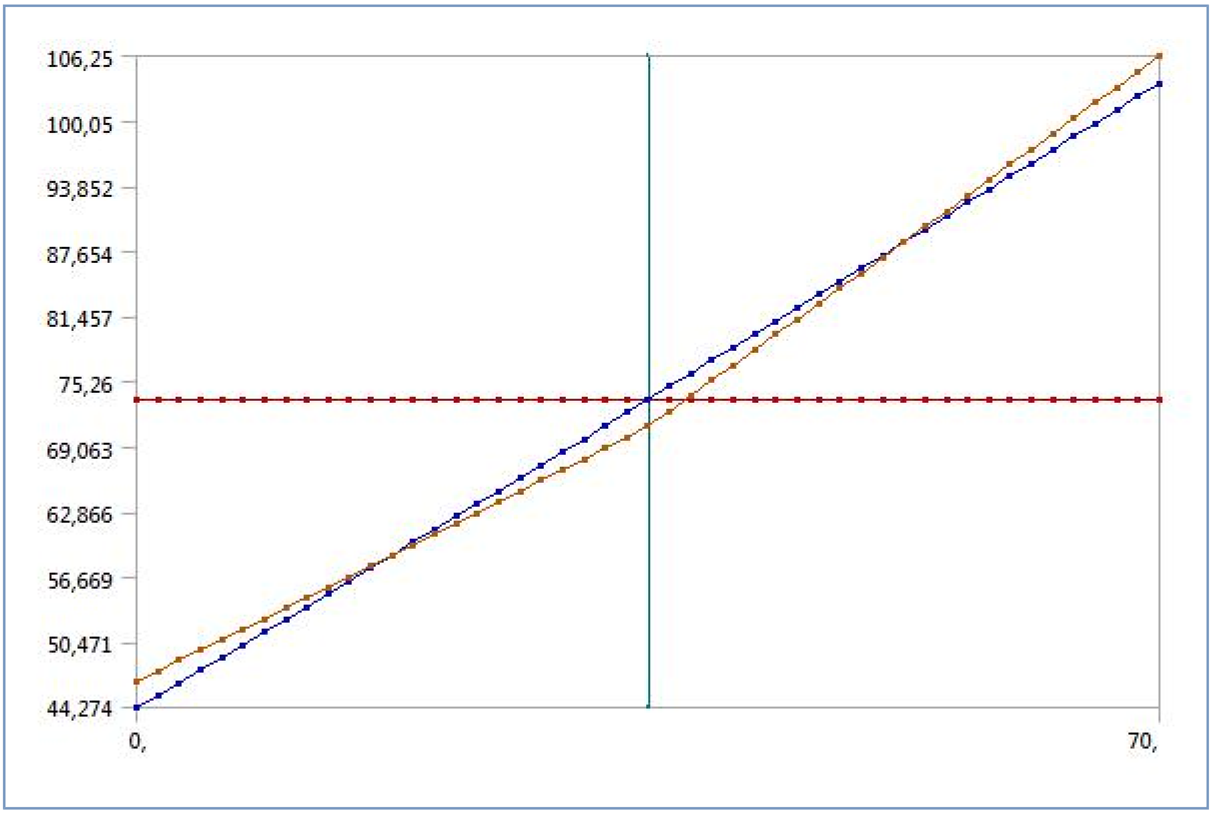


TABLE 21
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	73,563	30,617	44,274	3,3441	46,814
1,4583		29,342	45,46	3,3445	47,788
2,9167		28,066	46,65	3,3449	48,767
4,375		26,79	47,844	3,3454	49,75
5,8333		25,514	49,043	3,3458	50,736
7,2917		24,239	50,246	3,3462	51,726
8,75		22,963	51,452	3,3466	52,72
10,208		21,687	52,661	3,347	53,717
11,667		20,412	53,873	3,3475	54,717
13,125		19,136	55,089	3,3479	55,719
14,583		17,86	56,306	3,3483	56,725
16,042		16,584	57,527	3,3487	57,733
17,5		15,309	58,75	3,3492	58,743
18,958		14,033	59,975	3,3496	59,756
20,417		12,757	61,202	3,35	60,771
21,875		11,481	62,431	3,3504	61,788
23,333		10,206	63,662	3,3509	62,807
24,792		8,9301	64,894	3,3513	63,828
26,25		7,6543	66,128	3,3517	64,851
27,708		6,3786	67,364	3,3521	65,875
29,167		5,1029	68,601	3,3526	66,901
30,625		3,8272	69,84	3,353	67,929
32,083		2,5514	71,08	3,3534	68,958
33,542		1,2757	72,321	3,3538	69,989
35,		1,3496e-014	73,563	3,3543	71,021
36,458		1,2757	74,807	3,3547	72,473
37,917		2,5514	76,051	3,3551	73,927
39,375		3,8272	77,297	3,3555	75,382
40,833		5,1029	78,543	3,356	76,84

42,292		6,3786	79,791	3,3564	78,299
43,75		7,6543	81,039	3,3568	79,759
45,208		8,9301	82,288	3,3572	81,221
46,667		10,206	83,538	3,3577	82,685
48,125		11,481	84,789	3,3581	84,15
49,583		12,757	86,04	3,3585	85,616
51,042		14,033	87,293	3,3589	87,083
52,5		15,309	88,545	3,3594	88,552
53,958		16,584	89,799	3,3598	90,022
55,417		17,86	91,053	3,3602	91,492
56,875		19,136	92,308	3,3607	92,964
58,333		20,412	93,563	3,3611	94,436
59,792		21,687	94,819	3,3615	95,91
61,25		22,963	96,075	3,3619	97,384
62,708		24,239	97,332	3,3624	98,859
64,167		25,514	98,589	3,3628	100,34
65,625		26,79	99,847	3,3632	101,81
67,083		28,066	101,11	3,3636	103,29
68,542		29,342	102,36	3,3641	104,77
70,		30,617	103,62	3,3645	106,25

FIGURE 6
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 2

TABLE 22
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 2

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		52,457	15,151	10,689	7,6146
1,75		50,271	13,099	10,851	6,8507
3,5		48,085	11,099	11,014	6,313
5,2499		45,9	9,1837	11,177	6,0586
6,9999		43,714	7,4194	11,341	6,1197
8,7499		41,528	5,9422	11,505	6,4842

10,5	38,414	39,343	5,0126	11,67	7,1029
12,25		37,157	4,9494	11,835	7,9139
14,		34,971	5,7812	12,	8,8623
15,75		32,785	7,2043	12,166	9,9069
17,5		30,6	8,9407	12,332	11,019
19,25		28,414	10,841	12,498	12,177
21,		26,228	12,832	12,665	13,37
22,75		24,043	14,878	12,831	14,586
24,5		21,857	16,959	12,999	15,82
26,25		19,671	19,064	13,166	17,114
28,		17,486	21,185	13,334	18,704
29,75		15,3	23,317	13,502	20,321
31,5		13,114	25,459	13,67	21,961
33,25		10,928	27,608	13,839	23,618
35,		8,7428	29,762	14,008	25,289
36,75		6,5571	31,92	14,177	26,971
38,499		4,3714	34,082	14,346	28,585
40,249		2,1857	36,247	14,515	30,01
41,999		0,	38,414	14,685	31,438
43,749		2,1857	40,583	14,855	32,87
45,499		4,3714	42,754	15,025	34,305
47,249		6,5571	44,926	15,195	35,792
48,999		8,7428	47,1	15,365	37,447
50,749		10,928	49,274	15,536	39,105
52,499		13,114	51,45	15,707	40,764
54,249		15,3	53,626	15,878	42,939
55,999		17,486	55,803	16,049	45,445
57,749		19,671	57,981	16,22	47,956
59,499		21,857	60,159	16,391	52,187
61,249		24,043	62,338	16,562	56,232
62,999		26,228	64,517	16,734	60,281
64,749		28,414	66,697	16,906	64,334
66,499		30,6	68,877	17,078	68,388
68,249		32,785	71,057	17,25	72,443
69,999		34,971	73,238	17,422	76,498
71,749		37,157	75,419	17,594	80,552
73,499		39,343	77,601	17,766	84,606
75,249		41,528	79,782	17,939	88,658
76,999		43,714	81,964	18,111	92,708
78,749		45,9	84,146	18,284	96,756
80,499		48,085	86,328	18,456	100,8
82,249		50,271	88,51	18,629	104,84
83,999		52,457	90,693	18,802	108,89

FIGURE 7
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 3

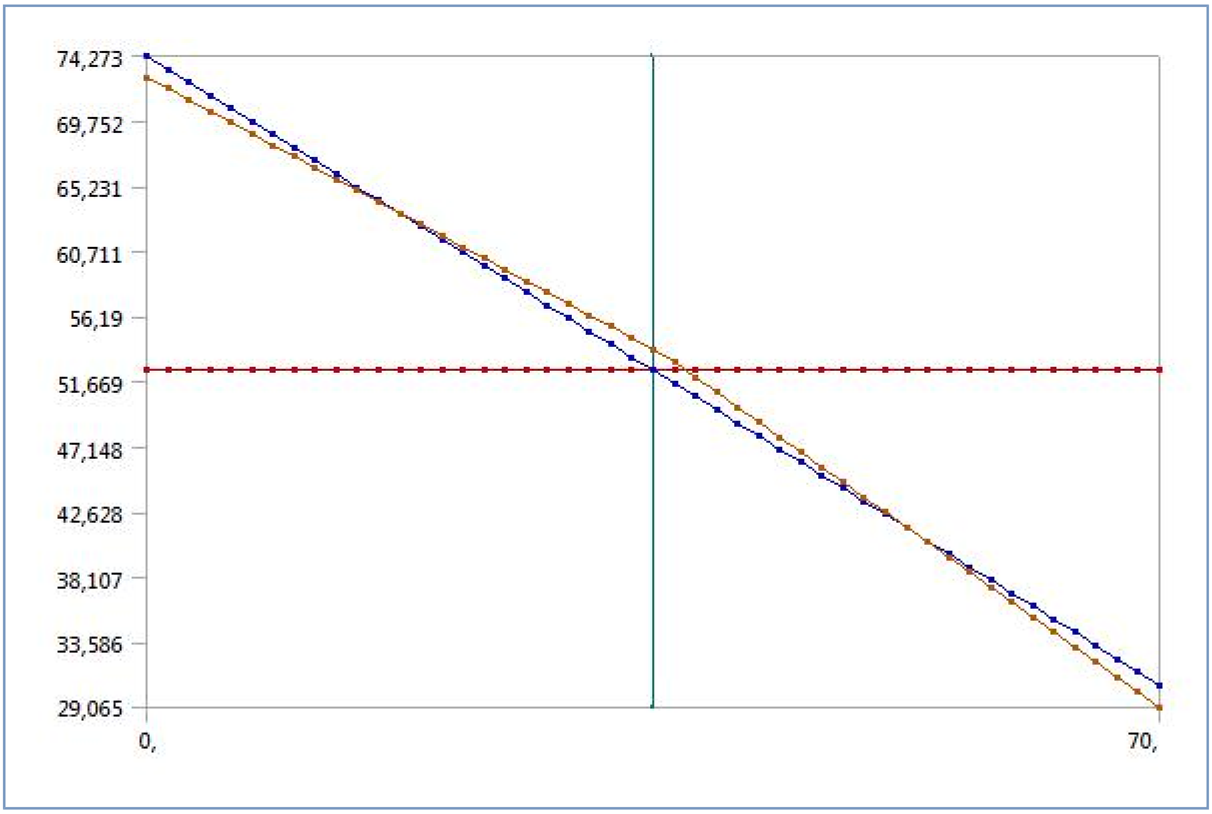


TABLE 23
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 3

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	52,451	21,825	74,273	1,5507	72,786
1,4583		20,916	73,364	1,5524	72,
2,9167		20,006	72,454	1,5542	71,213
4,375		19,097	71,545	1,5559	70,427
5,8333		18,188	70,636	1,5576	69,641
7,2917		17,278	69,726	1,5594	68,855
8,75		16,369	68,817	1,5611	68,069
10,208		15,459	67,908	1,5629	67,282
11,667		14,55	66,999	1,5646	66,496
13,125		13,641	66,089	1,5664	65,71
14,583		12,731	65,18	1,5681	64,924
16,042		11,822	64,271	1,5699	64,138
17,5		10,913	63,362	1,5716	63,352
18,958		10,003	62,452	1,5734	62,566
20,417		9,0938	61,543	1,5751	61,78
21,875		8,1844	60,634	1,5769	60,994
23,333		7,275	59,725	1,5786	60,208
24,792		6,3656	58,815	1,5804	59,422
26,25		5,4563	57,906	1,5821	58,636
27,708		4,5469	56,997	1,5839	57,85
29,167		3,6375	56,088	1,5856	57,064
30,625		2,7281	55,178	1,5873	56,279
32,083		1,8188	54,269	1,5891	55,493
33,542		0,90938	53,36	1,5908	54,707
35,		0,	52,451	1,5926	53,921
36,458		0,90938	51,542	1,5943	53,007
37,917		1,8188	50,632	1,5961	51,965
39,375		2,7281	49,723	1,5978	50,924
40,833		3,6375	48,814	1,5996	49,883

42,292		4,5469	47,905	1,6013	48,841
43,75		5,4563	46,996	1,6031	47,8
45,208		6,3656	46,087	1,6048	46,759
46,667		7,275	45,177	1,6066	45,718
48,125		8,1844	44,268	1,6083	44,677
49,583		9,0938	43,359	1,6101	43,635
51,042		10,003	42,45	1,6118	42,594
52,5		10,913	41,541	1,6136	41,553
53,958		11,822	40,632	1,6153	40,512
55,417		12,731	39,723	1,617	39,471
56,875		13,641	38,814	1,6188	38,43
58,333		14,55	37,905	1,6205	37,389
59,792		15,459	36,995	1,6223	36,349
61,25		16,369	36,086	1,624	35,308
62,708		17,278	35,177	1,6258	34,267
64,167		18,188	34,268	1,6275	33,227
65,625		19,097	33,36	1,6293	32,186
67,083		20,006	32,451	1,631	31,146
68,542		20,916	31,542	1,6328	30,105
70,		21,825	30,633	1,6345	29,065

FIGURE 8
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 4

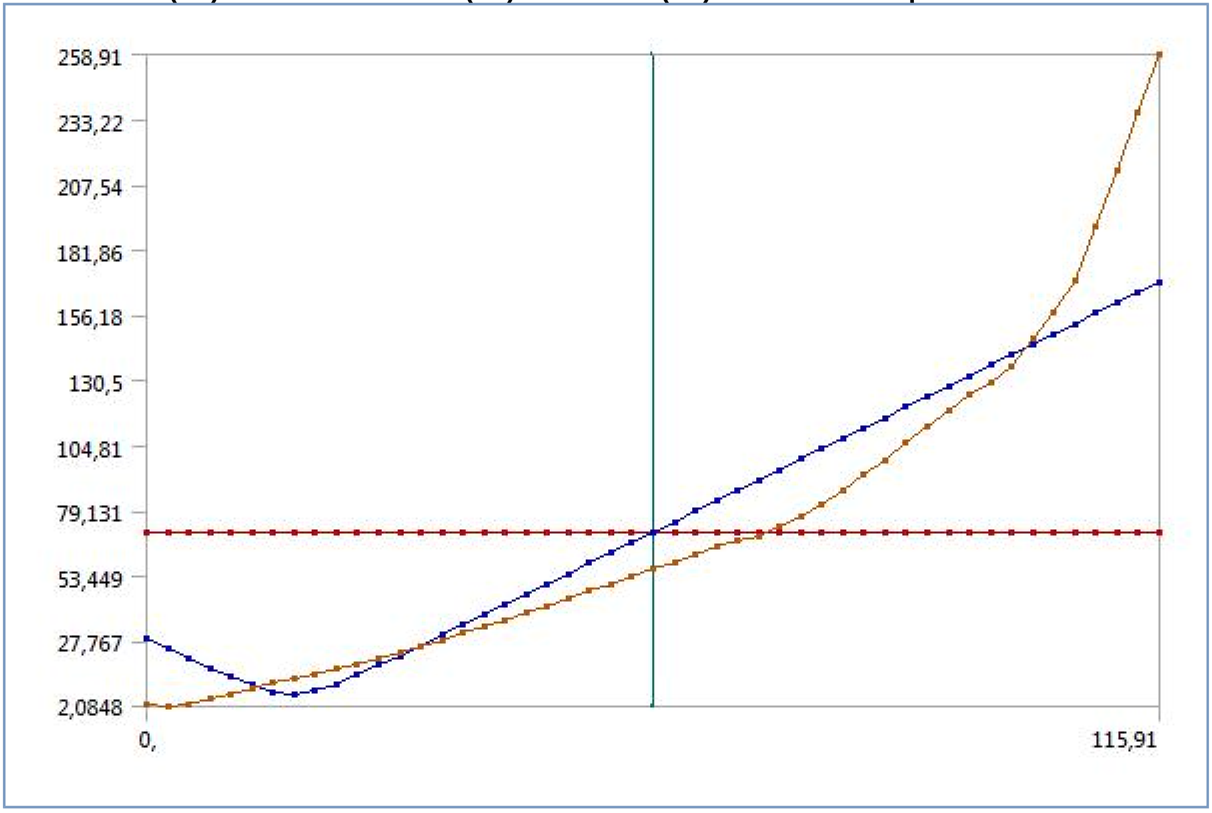


TABLE 24
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Equivalent Stress 4

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		98,758	29,187	27,043	2,738
2,4149		94,643	25,207	28,463	2,0848
4,8298		90,528	21,28	29,884	3,2561
7,2447		86,413	17,439	31,308	5,1251
9,6596		82,298	13,758	32,733	7,1687
12,074		78,183	10,407	34,159	9,2776

14,489	70,739	74,068	7,8221	35,587	11,422
16,904		69,953	6,9234	37,016	13,314
19,319		65,838	8,2793	38,445	15,019
21,734		61,723	11,092	39,876	16,911
24,149		57,609	14,538	41,307	19,004
26,564		53,494	18,264	42,739	21,26
28,979		49,379	22,128	44,172	23,554
31,394		45,264	26,069	45,605	25,858
33,808		41,149	30,056	47,038	28,443
36,223		37,034	34,074	48,472	31,067
38,638		32,919	38,113	49,907	33,675
41,053		28,804	42,166	51,342	36,258
43,468		24,689	46,23	52,777	38,813
45,883		20,574	50,303	54,213	41,339
48,298		16,46	54,382	55,648	44,507
50,713		12,345	58,465	57,085	47,397
53,128		8,2298	62,553	58,521	50,29
55,542		4,1149	66,645	59,958	53,184
57,957		4,293e-014	70,739	61,394	56,08
60,372		4,1149	74,835	62,832	58,977
62,787		8,2298	78,933	64,269	61,875
65,202		12,345	83,033	65,706	64,775
67,617		16,46	87,134	67,144	67,675
70,032		20,574	91,237	68,582	68,89
72,447		24,689	95,341	70,02	72,719
74,862		28,804	99,445	71,458	76,989
77,276		32,919	103,55	72,896	81,952
79,691		37,034	107,66	74,334	87,384
82,106		41,149	111,76	75,773	93,202
84,521		45,264	115,87	77,211	99,343
86,936		49,379	119,98	78,65	105,76
89,351		53,494	124,09	80,089	112,44
91,766		57,609	128,2	81,527	118,88
94,181		61,723	132,3	82,966	124,75
96,596		65,838	136,41	84,405	129,74
99,011		69,953	140,52	85,844	136,13
101,43		74,068	144,63	87,284	146,81
103,84		78,183	148,74	88,723	157,57
106,26		82,298	152,85	90,162	169,56
108,67		86,413	156,97	91,602	191,35
111,08		90,528	161,08	93,041	213,58
113,5		94,643	165,19	94,481	236,13
115,91		98,758	169,3	95,92	258,91

TABLE 25
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Linearized Maximum Principal Stress*	*Linearized Middle Principal Stress*	*Linearized Minimum Principal Stress*
State	Solved		
Scope			
Scoping Method	Path		
Path	Path 4		
Geometry	All Bodies		
Definition			
Type	Linearized Maximum Principal Stress	Linearized Middle Principal Stress	Linearized Minimum Principal Stress
Subtype	All		

By	Time		
Display Time	Last		
Coordinate System	Global Coordinate System		
2D Behavior	Planar		
Results			
Membrane	74,446 MPa	10,453 MPa	-1,5121 MPa
Bending (Inside)	0,18526 MPa	-7,8552 MPa	-102,35 MPa
Bending (Outside)	102,35 MPa	7,8552 MPa	-0,18526 MPa
Membrane+Bending (Inside)	4,1699 MPa	-2,8976 MPa	-27,901 MPa
Membrane+Bending (Center)	74,446 MPa	10,453 MPa	-1,5121 MPa
Membrane+Bending (Outside)	176,79 MPa	17,867 MPa	-1,256 MPa
Peak (Inside)	24,951 MPa	2,6051 MPa	-5,1124 MPa
Peak (Center)	7,1752 MPa	-1,3804 MPa	-13,859 MPa
Peak (Outside)	73,972 MPa	5,0757 MPa	-35,58 MPa
Total (Inside)	0,20597 MPa	-1,4369 MPa	-2,9548 MPa
Total (Center)	60,602 MPa	17,53 MPa	-2,8095 MPa
Total (Outside)	250,75 MPa	7,5188 MPa	-21,399 MPa
Information			
Time	1, s		
Load Step	1		
Substep	1		
Iteration Number	1		

FIGURE 9
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Maximum Principal Stress

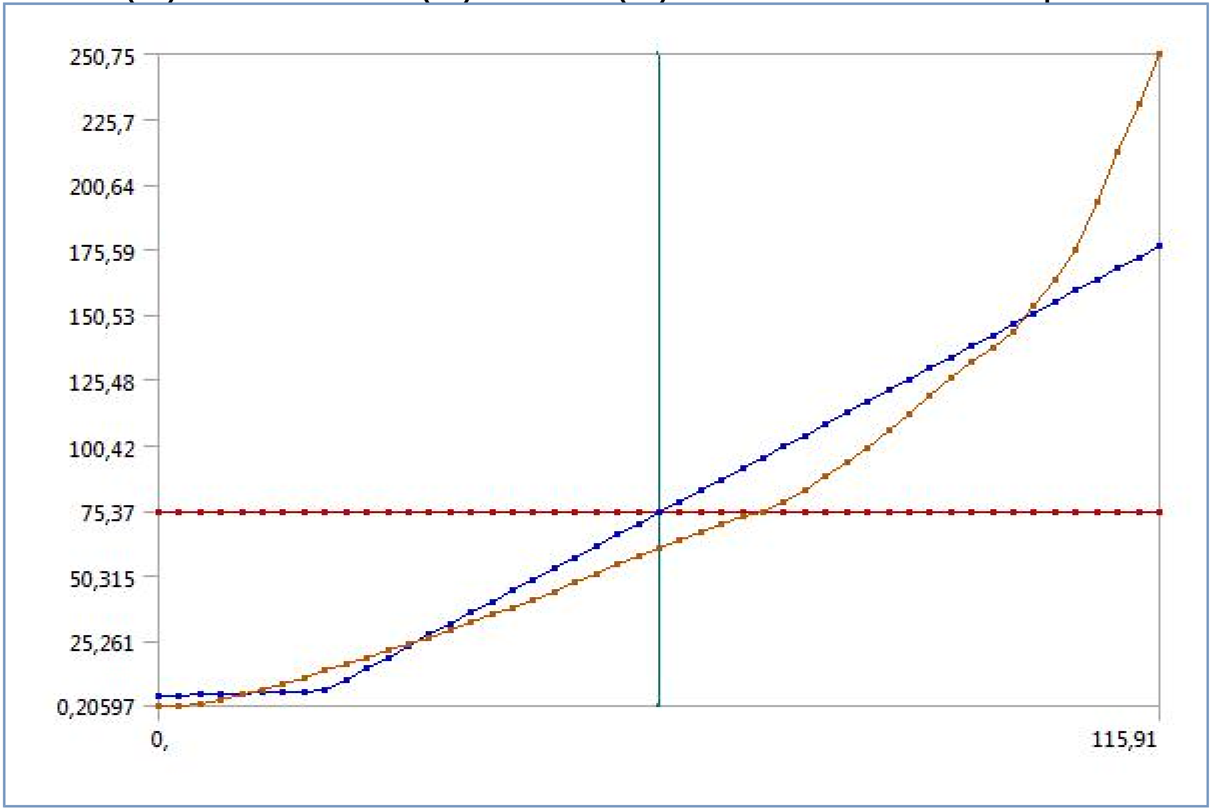


TABLE 26
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Maximum Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		0,18526	4,1699	24,951	0,20597
2,4149		0,17754	4,3755	25,972	0,29259

4,8298	74,446	0,16982	4,5884	26,993	0,65663
7,2447		0,1621	4,8081	28,014	2,5819
9,6596		0,15438	5,0344	29,035	4,5895
12,074		0,14667	5,2669	30,056	6,6791
14,489		0,13895	5,5057	31,077	8,8505
16,904		0,13123	5,7511	32,098	11,213
19,319		0,12351	6,2654	33,119	13,747
21,734		0,11579	10,483	34,141	16,287
24,149		0,10807	14,746	35,162	18,841
26,564		0,10035	19,01	36,183	21,421
28,979		9,2631e-002	23,274	37,204	24,023
31,394		8,4912e-002	27,538	38,226	26,647
33,808		7,7192e-002	31,802	39,247	29,51
36,223		6,9473e-002	36,067	40,268	32,413
38,638		6,1754e-002	40,331	41,289	35,321
41,053		5,4035e-002	44,595	42,311	38,236
43,468		4,6315e-002	48,86	43,332	41,156
45,883		3,8596e-002	53,124	44,353	44,082
48,298		3,0877e-002	57,389	45,375	48,206
50,713		2,3158e-002	61,653	46,396	51,305
53,128		1,5438e-002	65,917	47,417	54,404
55,542		7,7192e-003	70,182	48,438	57,503
57,957		4,4704e-014	74,446	49,46	60,602
60,372		4,2644	78,711	50,481	63,701
62,787		8,5289	82,975	51,502	66,8
65,202		12,793	87,24	52,524	69,899
67,617		17,058	91,504	53,545	72,998
70,032		21,322	95,768	54,566	74,685
72,447		25,587	100,03	55,588	78,972
74,862		29,851	104,3	56,609	83,386
77,276		34,116	108,56	57,63	88,353
79,691		38,38	112,83	58,652	93,827
82,106		42,644	117,09	59,673	99,697
84,521		46,909	121,36	60,694	105,88
86,936		51,173	125,62	61,716	112,33
89,351		55,438	129,88	62,737	119,42
91,766		59,702	134,15	63,759	126,39
94,181		63,967	138,41	64,78	132,62
96,596		68,231	142,68	65,801	137,66
99,011		72,496	146,94	66,823	143,69
101,43		76,76	151,21	67,844	153,94
103,84		81,024	155,47	68,865	164,19
106,26		85,289	159,74	69,887	175,37
108,67		89,553	164,	70,908	194,08
111,08		93,818	168,26	71,929	212,91
113,5		98,082	172,53	72,951	231,81
115,91		102,35	176,79	73,972	250,75

FIGURE 10
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Middle Principal Stress

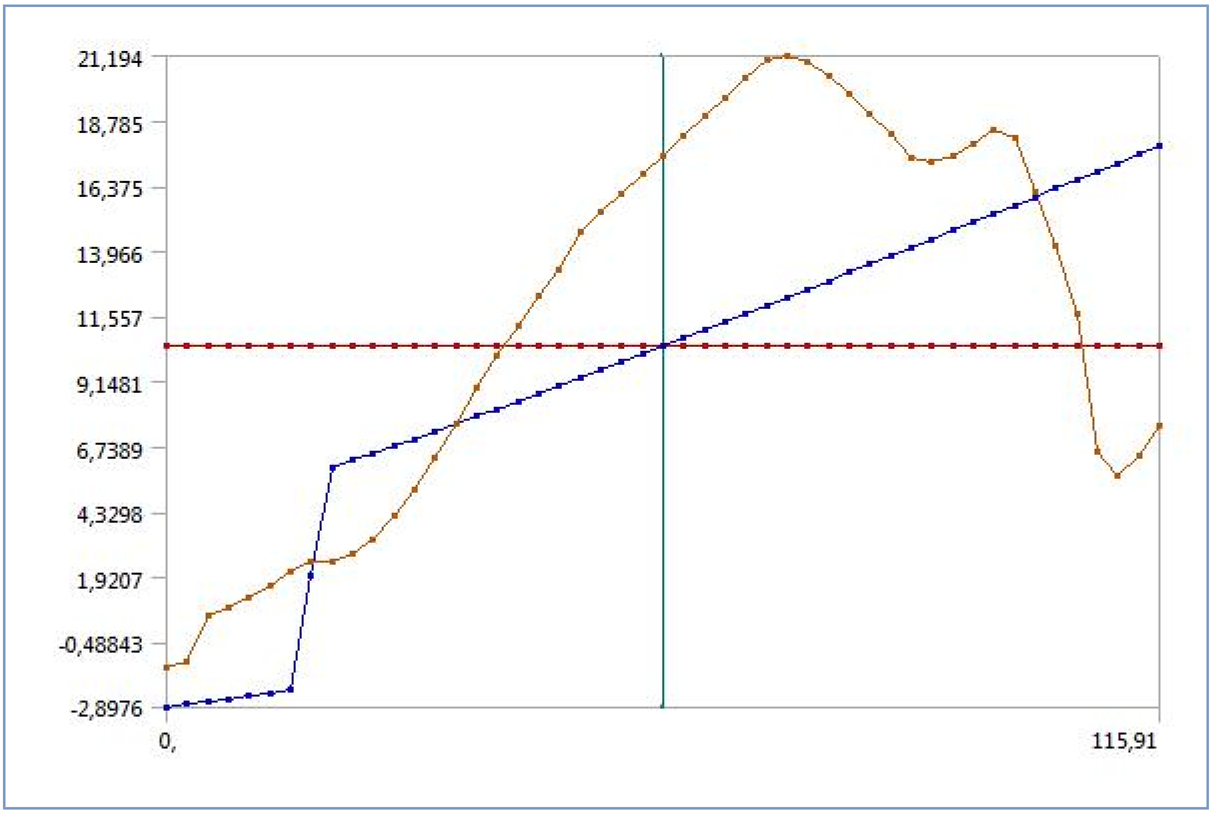


TABLE 27
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Middle Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,	10,453	-7,8552	-2,8976	2,6051	-1,4369
2,4149		-7,5279	-2,7835	2,6563	-1,1852
4,8298		-7,2006	-2,6766	2,7076	0,48411
7,2447		-6,8733	-2,5765	2,7589	0,76584
9,6596		-6,546	-2,4828	2,8103	1,1341
12,074		-6,2187	-2,3948	2,8616	1,591
14,489		-5,8914	-2,2388	2,913	2,14
16,904		-5,5641	1,9482	2,9644	2,4687
19,319		-5,2368	5,9464	3,0159	2,5128
21,734		-4,9095	6,2468	3,0673	2,7643
24,149		-4,5822	6,5064	3,1187	3,3152
26,564		-4,2549	6,7693	3,1702	4,1613
28,979		-3,9276	7,0358	3,2216	5,1896
31,394		-3,6003	7,3058	3,2731	6,327
33,808		-3,273	7,5792	3,3246	7,627
36,223		-2,9457	7,8557	3,376	8,899
38,638		-2,6184	8,1351	3,4275	10,11
41,053		-2,2911	8,4172	3,479	11,253
43,468		-1,9638	8,7018	3,5305	12,324
45,883		-1,6365	8,9888	3,582	13,321
48,298		-1,3092	9,2779	3,6334	14,705
50,713		-0,9819	9,5691	3,6849	15,409
53,128		-0,6546	9,8622	3,7364	16,115
55,542		-0,3273	10,157	3,7879	16,822
57,957		3,8547e-015	10,453	3,8394	17,53
60,372		0,3273	10,751	3,8909	18,24
62,787		0,6546	11,051	3,9424	18,95
65,202		0,9819	11,352	3,9939	19,662
67,617		1,3092	11,654	4,0454	20,374

70,032		1,6365	11,957	4,0969	21,059
72,447		1,9638	12,261	4,1484	21,194
74,862		2,2911	12,566	4,2	20,992
77,276		2,6184	12,873	4,2515	20,448
79,691		2,9457	13,18	4,303	19,802
82,106		3,273	13,488	4,3545	19,077
84,521		3,6003	13,797	4,406	18,291
86,936		3,9276	14,106	4,4575	17,455
89,351		4,2549	14,417	4,509	17,248
91,766		4,5822	14,728	4,5605	17,534
94,181		4,9095	15,039	4,612	17,924
96,596		5,2368	15,351	4,6636	18,481
99,011		5,5641	15,664	4,7151	18,197
101,43		5,8914	15,978	4,7666	16,177
103,84		6,2187	16,291	4,8181	14,161
106,26		6,546	16,606	4,8696	11,641
108,67		6,8733	16,921	4,9211	6,5504
111,08		7,2006	17,236	4,9726	5,6955
113,5		7,5279	17,551	5,0242	6,4396
115,91		7,8552	17,867	5,0757	7,5188

FIGURE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Minimum Principal Stress

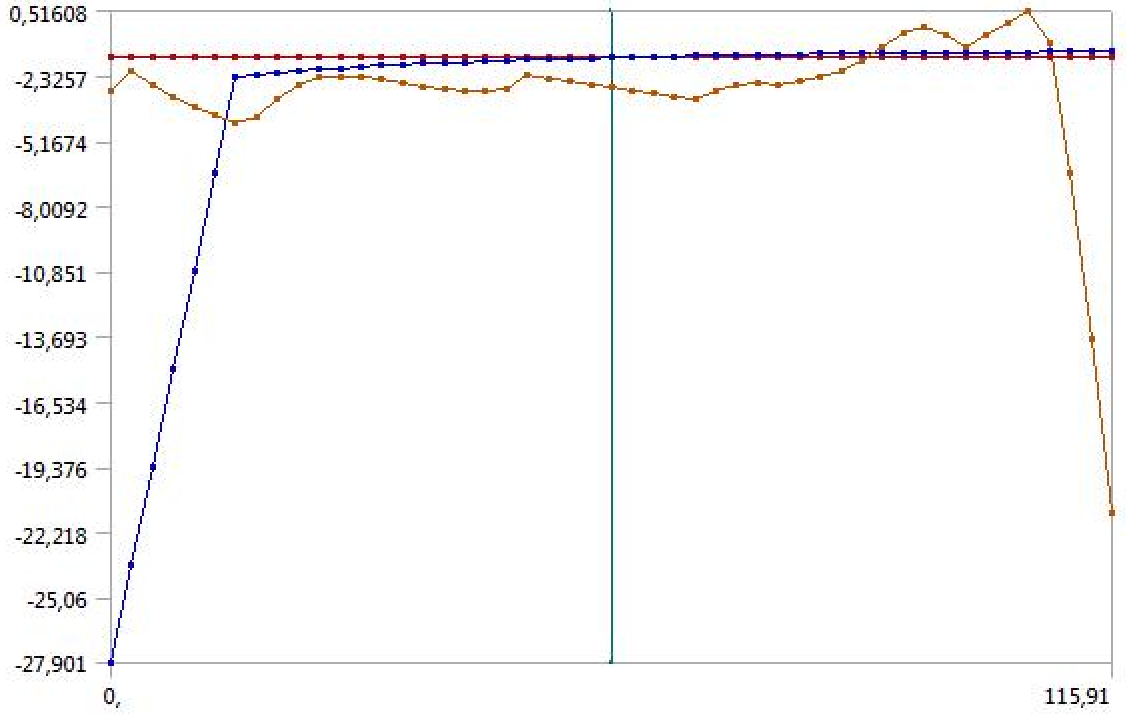


TABLE 28
Model (A4) > Static Structural (A5) > Solution (A6) > Linearized Minimum Principal Stress

Length [mm]	Membrane [MPa]	Bending [MPa]	Membrane+Bending [MPa]	Peak [MPa]	Total [MPa]
0,		-102,35	-27,901	-5,1124	-2,9548
2,4149		-98,082	-23,637	-5,7466	-2,0921
4,8298		-93,818	-19,373	-6,3809	-2,6823
7,2447		-89,553	-15,109	-7,0153	-3,204
9,6596		-85,289	-10,845	-7,6497	-3,6524
12,074		-81,024	-6,5811	-8,2842	-4,0293

14,489	-1,5121	-76,76	-2,3917	-8,9187	-4,3379
16,904		-72,496	-2,2402	-9,5533	-4,1092
19,319		-68,231	-2,1687	-10,188	-3,3121
21,734		-63,967	-2,1025	-10,823	-2,674
24,149		-59,702	-2,0409	-11,457	-2,3521
26,564		-55,438	-1,9836	-12,092	-2,3269
28,979		-51,173	-1,9302	-12,727	-2,387
31,394		-46,909	-1,8805	-13,361	-2,4594
33,808		-42,644	-1,8341	-13,996	-2,6422
36,223		-38,38	-1,7909	-14,631	-2,8073
38,638		-34,116	-1,7507	-15,265	-2,9218
41,053		-29,851	-1,7131	-15,9	-2,9791
43,468		-25,587	-1,6781	-16,535	-2,9753
45,883		-21,322	-1,6455	-17,17	-2,9081
48,298		-17,058	-1,615	-17,804	-2,2945
50,713		-12,793	-1,5866	-18,439	-2,4209
53,128		-8,5289	-1,56	-19,074	-2,549
55,542		-4,2644	-1,5352	-19,709	-2,6786
57,957		-4,2253e-017	-1,5121	-20,344	-2,8095
60,372		-7,7192e-003	-1,4905	-20,979	-2,9414
62,787		-1,5438e-002	-1,4703	-21,613	-3,0744
65,202		-2,3158e-002	-1,4515	-22,248	-3,2083
67,617		-3,0877e-002	-1,4339	-22,883	-3,3429
70,032		-3,8596e-002	-1,4175	-23,518	-3,0097
72,447		-4,6315e-002	-1,4022	-24,153	-2,6837
74,862		-5,4035e-002	-1,3879	-24,787	-2,651
77,276		-6,1754e-002	-1,3746	-25,422	-2,6766
79,691		-6,9473e-002	-1,3622	-26,057	-2,5676
82,106		-7,7192e-002	-1,3506	-26,692	-2,3541
84,521		-8,4912e-002	-1,3399	-27,327	-2,0581
86,936		-9,2631e-002	-1,3299	-27,962	-1,6965
89,351		-0,10035	-1,3206	-28,597	-1,054
91,766		-0,10807	-1,312	-29,231	-0,45802
94,181		-0,11579	-1,3041	-29,866	-0,21707
96,596		-0,12351	-1,2967	-30,501	-0,54329
99,011		-0,13123	-1,2899	-31,136	-1,0276
101,43		-0,13895	-1,2836	-31,771	-0,49581
103,84		-0,14667	-1,2779	-32,406	3,132e-002
106,26		-0,15438	-1,2726	-33,041	0,51608
108,67		-0,1621	-1,2678	-33,676	-0,86731
111,08		-0,16982	-1,2635	-34,31	-6,5161
113,5		-0,17754	-1,2595	-34,945	-13,784
115,91		-0,18526	-1,256	-35,58	-21,399

Material Data

Structural Steel

TABLE 29
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 30
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

TABLE 31
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
165

TABLE 32
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
165

TABLE 33
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
310

TABLE 34
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 35
Structural Steel > Alternating Stress Mean Stress

Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

TABLE 36
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 37
Structural Steel > Isotropic Elasticity

Temperature C	Young’s Modulus MPa	Poisson’s Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

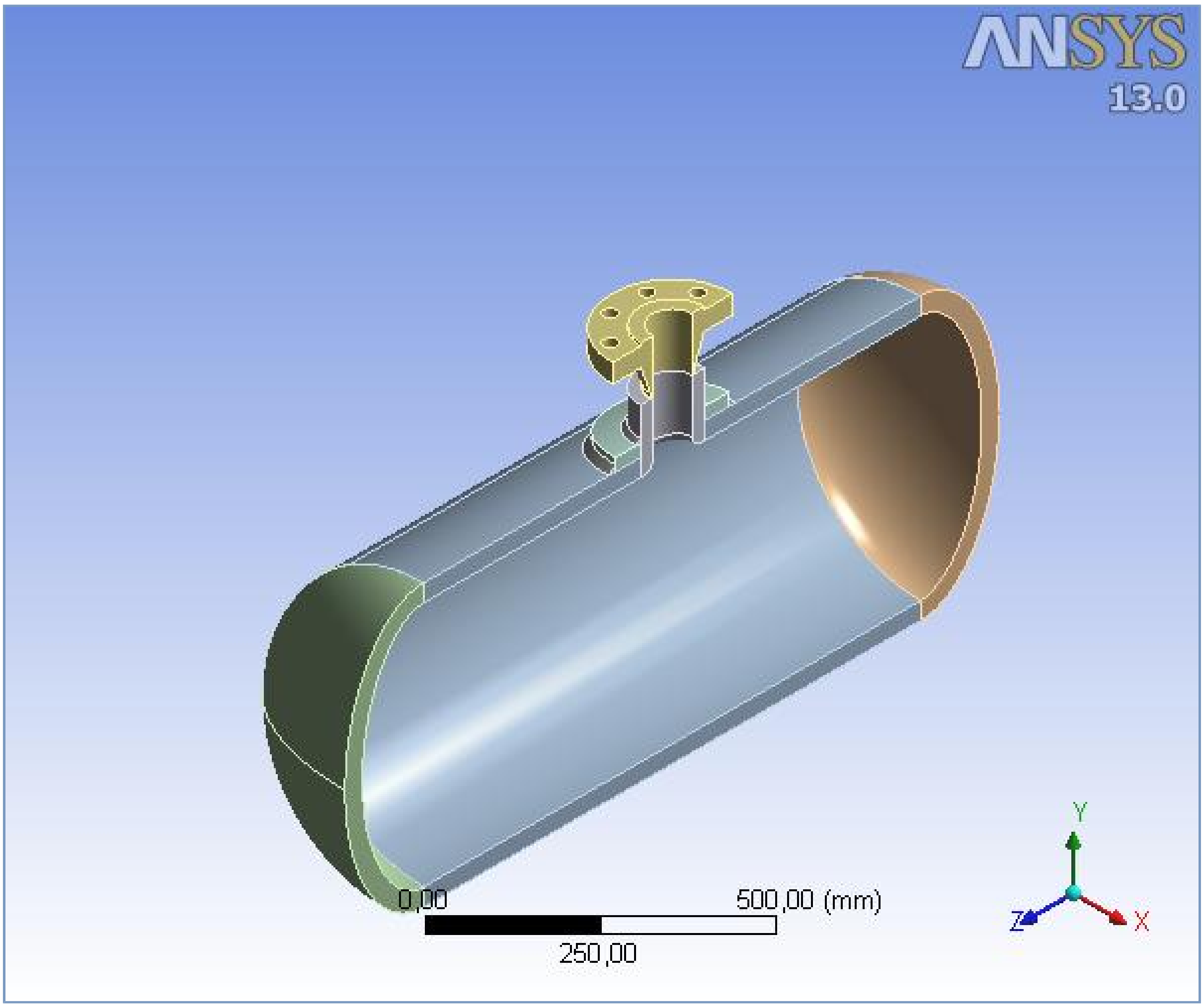
TABLE 38
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

APPENDIX H: Calculation report from ANSYS, Elastic - plastic Stress Analysis, ASME VIII div. 2; 2010, Protection against plastic collapse (35 mm @ 100 bar)

Project

First Saved	Monday, April 16, 2012
Last Saved	Tuesday, May 22, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thin wall\Assembly\weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	283,27 mm
Length Y	766,01 mm
Length Z	1317, mm
Properties	
Volume	4,1686e+007 mm³
Mass	327,23 kg
Scale Factor Value	1,

Statistics	
Bodies	7
Active Bodies	7
Nodes	29812
Elements	14640
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	Yes
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell:1*	*End cap:1*	*End cap:2*	*Reinforcement pad2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	SA-516 grade 70				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127,52 mm	283, mm	283,27 mm		114,53 mm
Length Y	125,14 mm	566, mm	566,01 mm		49,228 mm
Length Z	254,79 mm	1000, mm	158,52 mm		229,01 mm
Properties					
Volume	40570 mm³	2,8961e+007 mm³	5,4737e+006 mm³	5,4736e+006 mm³	3,6082e+005 mm³
Mass	0,31847 kg	227,34 kg	42,969 kg	42,968 kg	2,8325 kg
Centroid X	-69,697 mm	-170,08 mm	-125,46 mm	-125,47 mm	-59,042 mm

Centroid Y	281,88 mm	-2,0967 mm	-8,7545e-003 mm	5,5528e-002 mm	287,9 mm
Centroid Z	-0,11401 mm	-3,1226e-007 mm	578,79 mm	-578,79 mm	-3,779e-005 mm
Moment of Inertia Ip1	1952,7 kg·mm²	2,7057e+007 kg·mm²	9,7267e+005 kg·mm²	9,7333e+005 kg·mm²	12232 kg·mm²
Moment of Inertia Ip2	2395,9 kg·mm²	2,0592e+007 kg·mm²	3,1819e+005 kg·mm²	3,1829e+005 kg·mm²	14729 kg·mm²
Moment of Inertia Ip3	548,99 kg·mm²	9,473e+006 kg·mm²	1,1875e+006 kg·mm²	1,1882e+006 kg·mm²	2811, kg·mm²
Statistics					
Nodes	590	10793	4472	4903	973
Elements	143	5373	2483	2759	144
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 900 RF:1*	*Nozzle2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	SA-516 grade 70	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	120,5 mm	64,497 mm
Length Y	108, mm	135,53 mm
Length Z	241, mm	128,97 mm
Properties		
Volume	8,49e+005 mm³	5,2735e+005 mm³
Mass	6,6646 kg	4,1397 kg
Centroid X	-50,292 mm	-33,99 mm
Centroid Y	452,57 mm	311,08 mm
Centroid Z	-2,5746e-007 mm	2,3667e-004 mm
Moment of Inertia Ip1	25412 kg·mm²	11450 kg·mm²
Moment of Inertia Ip2	29464 kg·mm²	6997,5 kg·mm²
Moment of Inertia Ip3	8702,5 kg·mm²	6841,3 kg·mm²
Statistics		
Nodes	5902	2179
Elements	3374	364
Mesh Metric	None	

Coordinate Systems

TABLE 5
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian

Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	3,8742 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 8
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region*	*Contact Region 2*	*Contact Region 3*	*Contact Region 4*	*Contact Region 5*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face	2 Faces	1 Face
Target	1 Face	2 Faces	1 Face	2 Faces	1 Face
Contact Bodies	Welds				Main shell:1
Target Bodies	Main shell:1	Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1	Nozzle2:1	End cap:1
Definition					
Type	Bonded				

Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

TABLE 9
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Contact Region 6*	*Bonded - Main shell:1 To Reinforcement pad2:1*	*Contact Region 8*	*Contact Region 9*	*Contact Region 10*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face				
Target	1 Face				
Contact Bodies	Main shell:1			Reinforcement pad2:1	3 inch Weldneck Flange 900 RF:1
Target Bodies	End cap:2	Reinforcement pad2:1	Nozzle2:1		
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

Mesh

TABLE 10
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	2,26280 mm

Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	29812
Elements	14640
Mesh Metric	None

Static Structural (A5)

TABLE 11
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	20, °C
Generate Input Only	No

TABLE 12
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	

Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thin wall configuration - Elastic-plastic Stress Analysis - ASME VIII div.2; 2010_files\dp0 \SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	nmm

TABLE 13
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Acceleration*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0, mm/s² (ramped)
Y Component	-20594 mm/s² (ramped)
Z Component	0, mm/s² (ramped)
Suppressed	No

FIGURE 1
Model (A4) > Static Structural (A5) > Acceleration

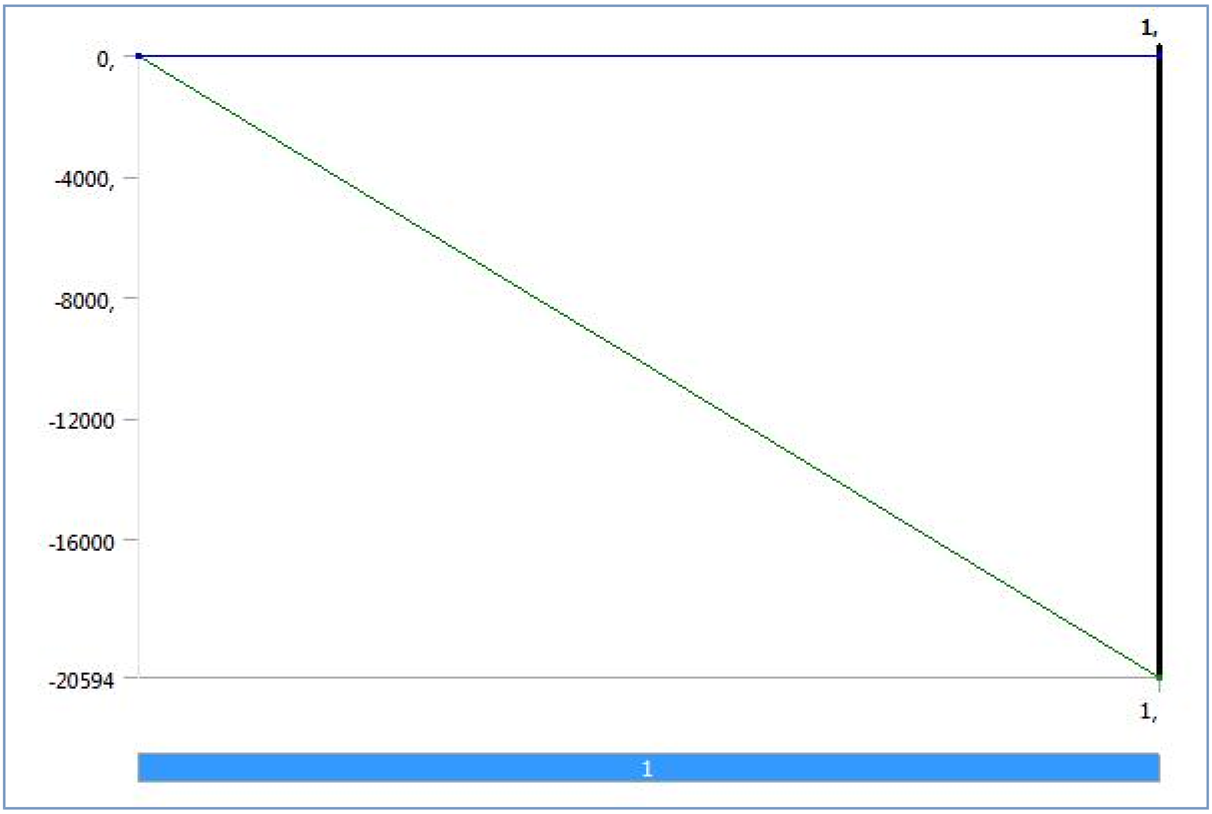


TABLE 14
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	15 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components
Magnitude	21, MPa (ramped)				
Suppressed	No				
Coordinate System			Global Coordinate System		Global Coordinate System
X Component			0, mm (ramped)		0, N (ramped)
Y Component			Free		40500 N (ramped)
Z Component			0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

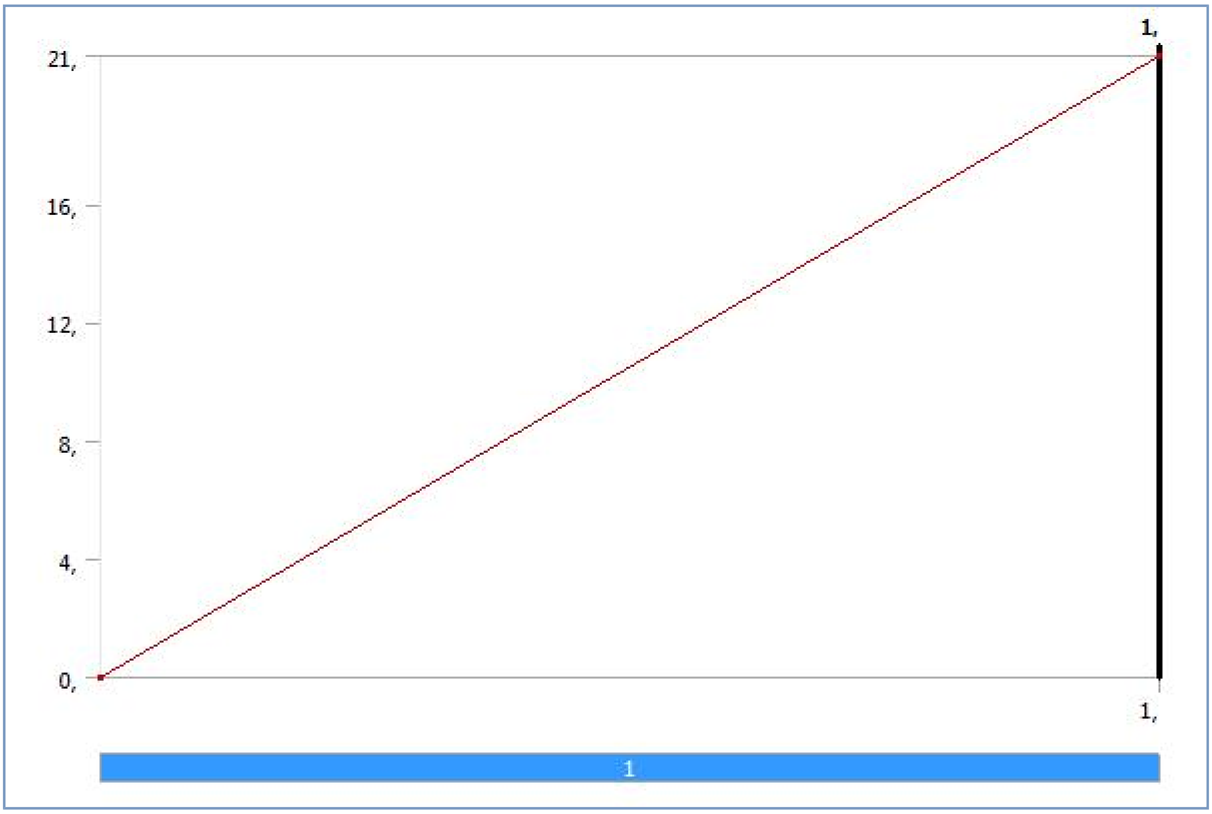


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

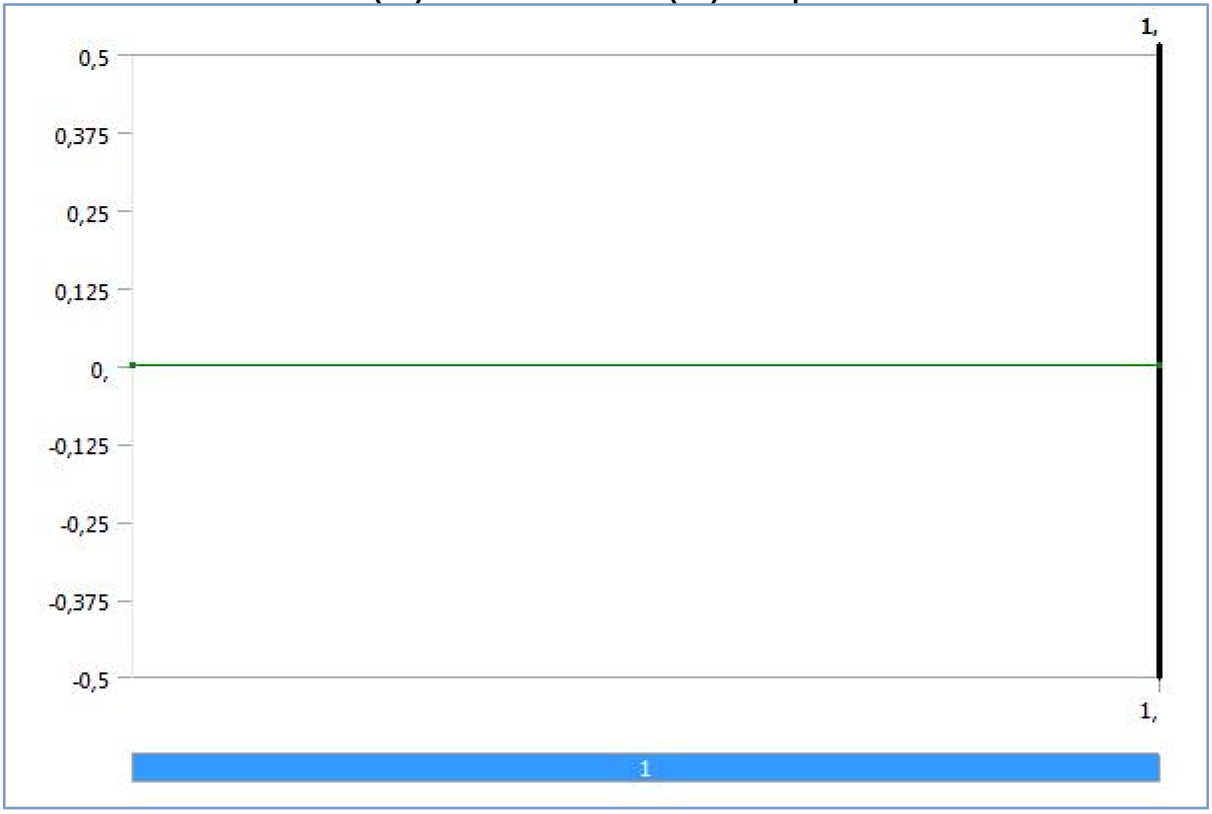
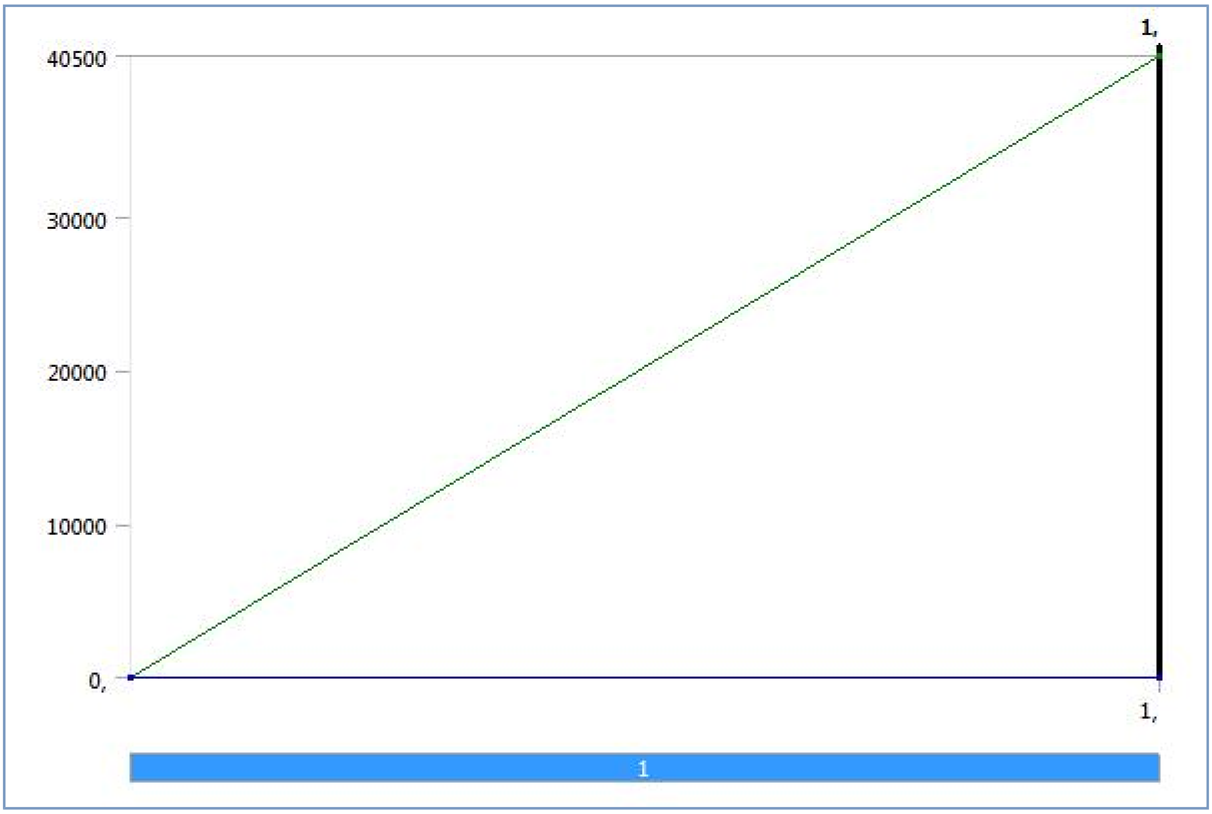


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 15
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 16
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 17
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Equivalent Elastic Strain*	*Equivalent Stress*	*Equivalent Plastic Strain*	*Equivalent Total Strain*	*Maximum Principal Stress*
State	Solved				
Scope					
Scoping Method	Geometry Selection				
Geometry	All Bodies				
Definition					

Type	Equivalent (von-Mises) Elastic Strain	Equivalent (von-Mises) Stress	Equivalent Plastic Strain	Equivalent Total Strain	Maximum Principal Stress
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Integration Point Results					
Display Option	Averaged				
Results					
Minimum	1,9942e-006 mm/mm	0,39884 MPa	0, mm/mm	1,0459e-005 mm/mm	-3,3177 MPa
Maximum	1,3155e-003 mm/mm	263,1 MPa	1,1786e-003 mm/mm	2,4803e-003 mm/mm	281,02 MPa
Minimum Occurs On	3 inch Weldneck Flange 900 RF:1		Welds	3 inch Weldneck Flange 900 RF:1	Reinforcement pad2:1
Maximum Occurs On	Nozzle2:1				
Minimum Value Over Time					
Minimum	3,4948e-007 mm/mm	6,9896e-002 MPa	0, mm/mm	2,1038e-006 mm/mm	-3,3177 MPa
Maximum	1,9942e-006 mm/mm	0,39884 MPa	0, mm/mm	1,0459e-005 mm/mm	-0,62453 MPa
Maximum Value Over Time					
Minimum	4,2329e-004 mm/mm	84,659 MPa	0, mm/mm	4,233e-004 mm/mm	84,376 MPa
Maximum	1,3155e-003 mm/mm	263,1 MPa	1,1786e-003 mm/mm	2,4803e-003 mm/mm	281,02 MPa
Information					
Time	1, s				
Load Step	1				
Substep	4				
Iteration Number	4				

FIGURE 5
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Elastic Strain

TABLE 18
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Elastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	3,4948e-007	4,2329e-004
0,4	6,9895e-007	8,4659e-004
0,7	1,2278e-006	1,2927e-003
1,	1,9942e-006	1,3155e-003

FIGURE 6
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

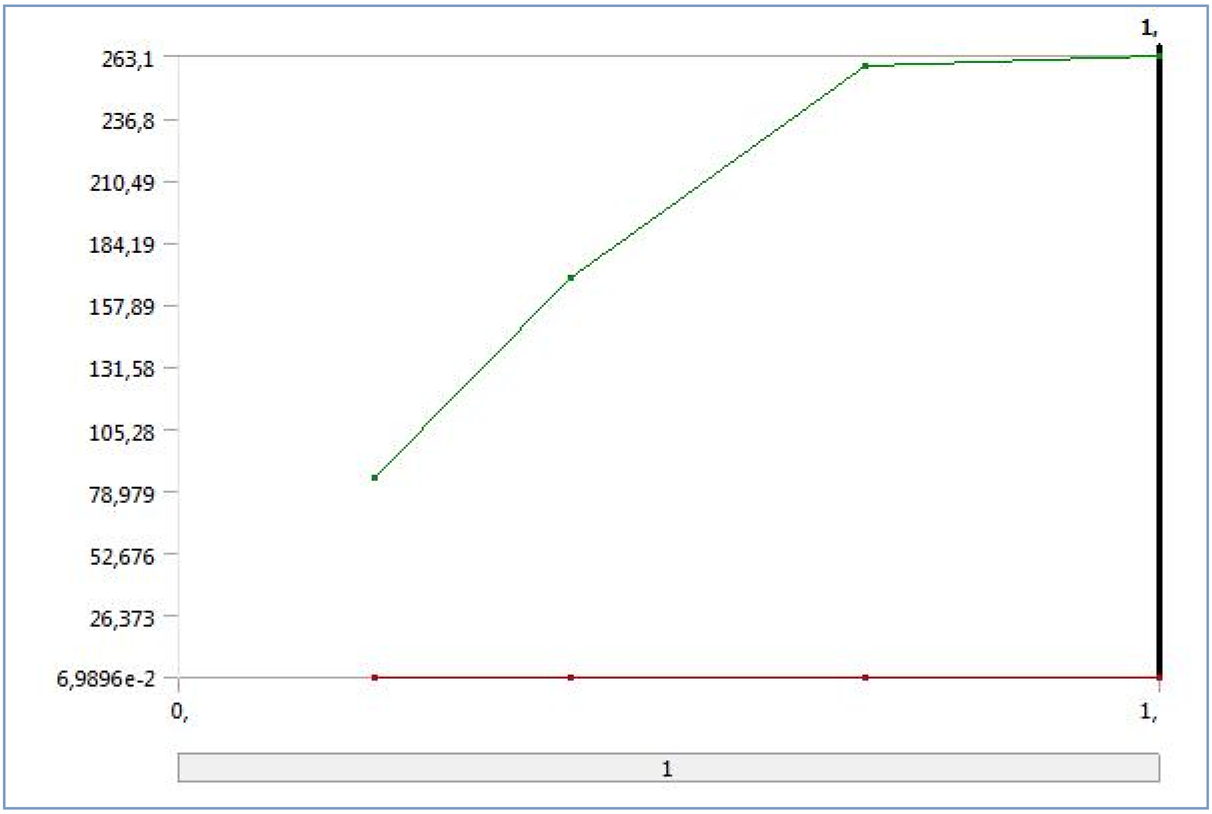


TABLE 19
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	6,9896e-002	84,659
0,4	0,13979	169,32
0,7	0,24556	258,55
1,	0,39884	263,1

FIGURE 7
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Plastic Strain

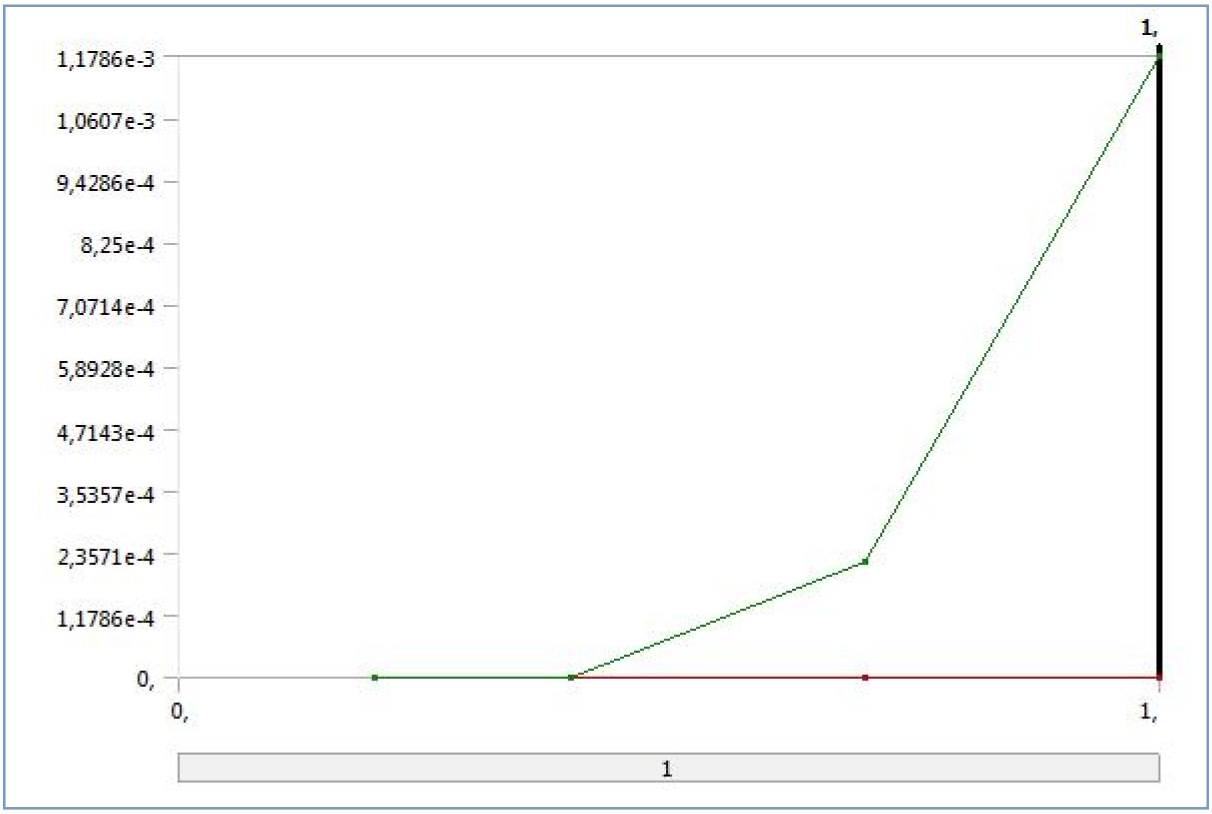


TABLE 20
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Plastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	0,	0,
0,4		
0,7		2,2127e-004
1,		1,1786e-003

FIGURE 8
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Total Strain

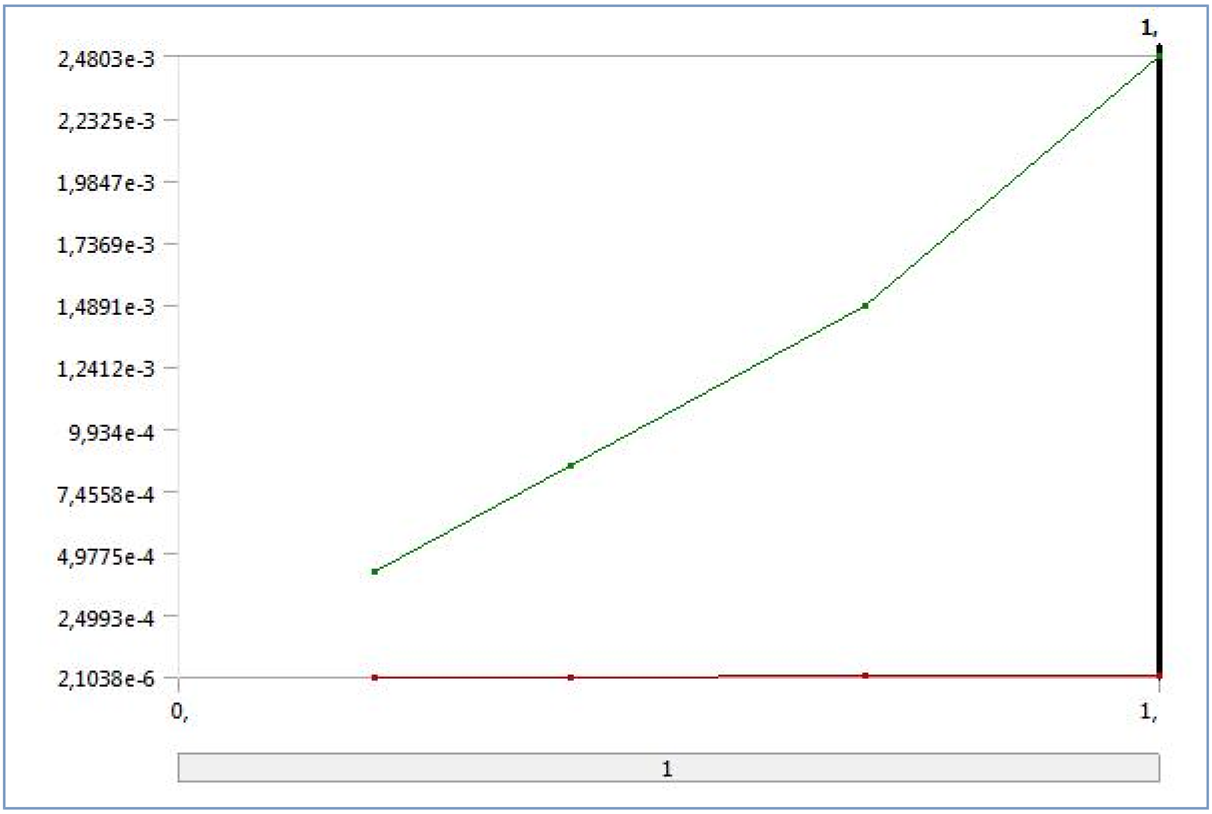


TABLE 21
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Total Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	2,1038e-006	4,233e-004
0,4	4,2075e-006	8,466e-004
0,7	7,362e-006	1,481e-003
1,	1,0459e-005	2,4803e-003

FIGURE 9
Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Principal Stress

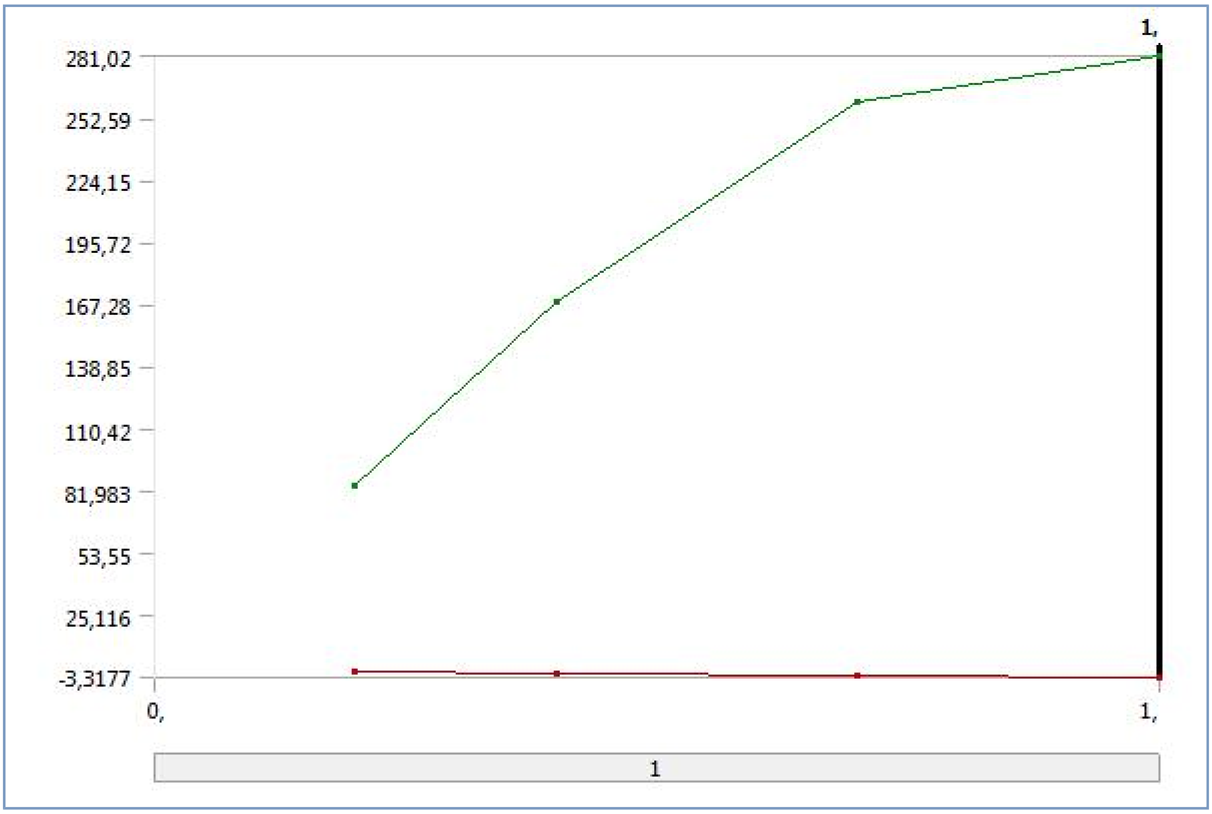


TABLE 22
Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-0,62453	84,376
0,4	-1,2491	168,75
0,7	-2,1874	259,99
1,	-3,3177	281,02

TABLE 23
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Middle Principal Stress*	*Minimum Principal Stress*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Middle Principal Stress	Minimum Principal Stress
By	Time	
Display Time	Last	
Calculate Time History	Yes	
Identifier		
Integration Point Results		
Display Option	Averaged	
Results		
Minimum	-30,584 MPa	-81,807 MPa
Maximum	203,35 MPa	184,74 MPa
Minimum Occurs On	End cap:2	Main shell:1
Maximum Occurs On	Main shell:1	
Minimum Value Over Time		
Minimum	-30,584 MPa	-81,807 MPa
Maximum	-6,1091 MPa	-16,354 MPa
Maximum Value Over Time		

Minimum	40,698 MPa	36,973 MPa
Maximum	203,35 MPa	184,74 MPa
Information		
Time	1, s	
Load Step	1	
Substep	4	
Iteration Number	4	

FIGURE 10
Model (A4) > Static Structural (A5) > Solution (A6) > Middle Principal Stress

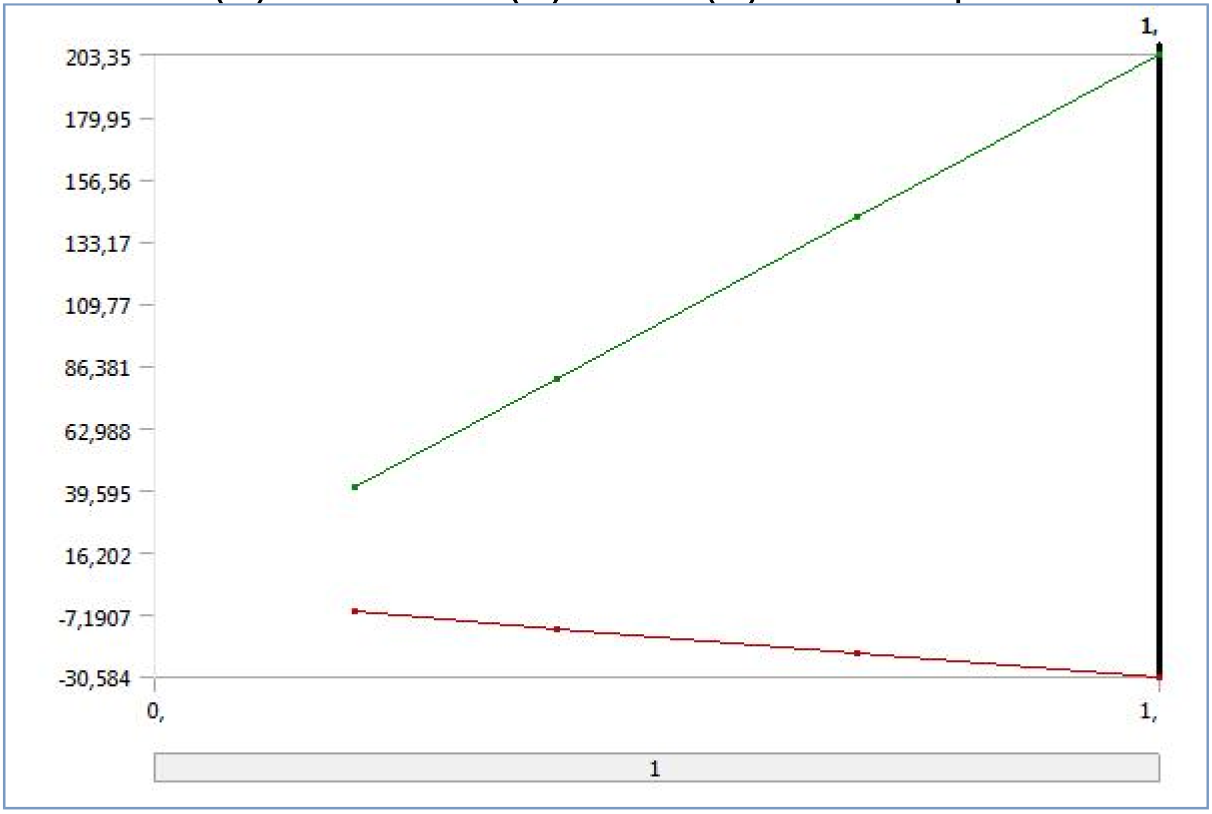


TABLE 24
Model (A4) > Static Structural (A5) > Solution (A6) > Middle Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-6,1091	40,698
0,4	-12,218	81,395
0,7	-21,383	142,44
1,	-30,584	203,35

FIGURE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Minimum Principal Stress

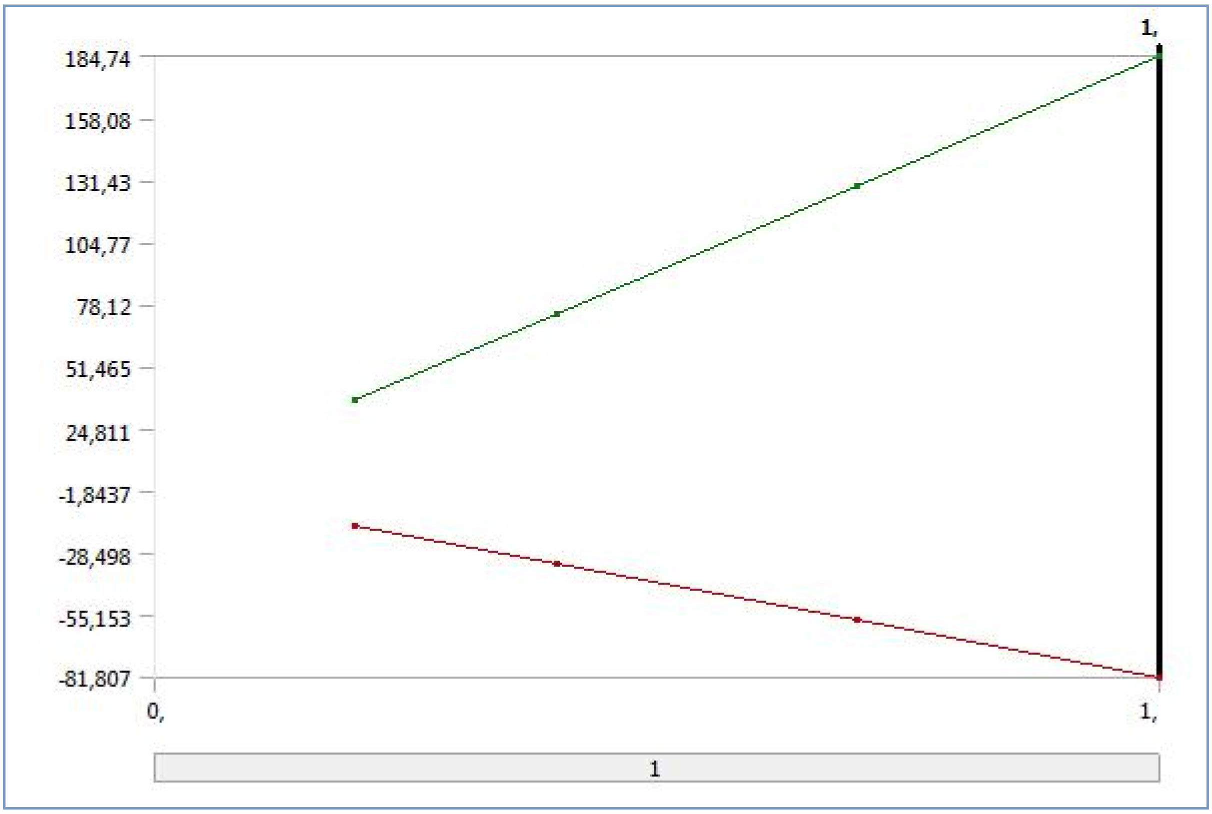


TABLE 25
Model (A4) > Static Structural (A5) > Solution (A6) > Minimum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-16,354	36,973
0,4	-32,709	73,945
0,7	-57,241	129,4
1,	-81,807	184,74

Material Data

SA-516 grade 70

TABLE 26
SA-516 grade 70 > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 27
SA-516 grade 70 > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 28
SA-516 grade 70 > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 29

SA-516 grade 70 > Isotropic Elasticity

Temperature C	Young’s Modulus MPa	Poisson’s Ratio	Bulk Modulus MPa	Shear Modulus MPa
20	2.e+005	0.3	1.6667e+005	76923

TABLE 30
SA-516 grade 70 > Isotropic Relative Permeability

Relative Permeability
10000

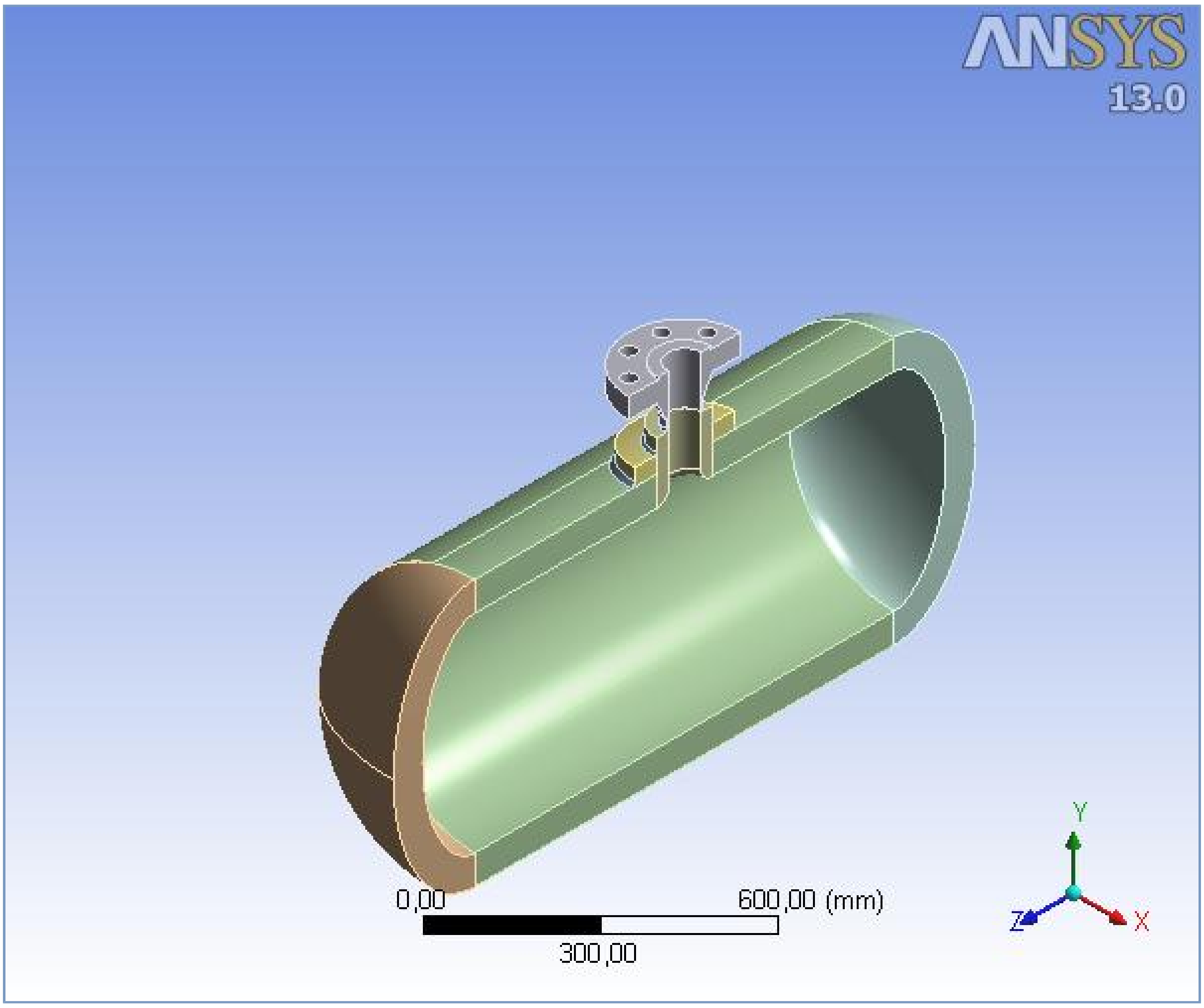
TABLE 31
SA-516 grade 70 > Multilinear Isotropic Hardening

Stress MPa	Plastic Strain m m^-1	Temperature C
250	0	20
300	5.71e-003	20
350	2.42e-002	20
400	5.02e-002	20
450	7.82e-002	20
500	0.114	20
550	0.161	20
600	0.22	20

APPENDIX I: Calculation report from ANSYS Elastic - plastic Stress Analysis, ASME VIII div. 2; 2010, Protection against plastic collapse (70 mm @ 200 bar)

Project

First Saved	Tuesday, April 10, 2012
Last Saved	Wednesday, May 23, 2012
Product Version	13.0 Release



Contents

Units

TABLE 1

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Master thesis Frode Tjelta\Inventor\Thick wall\Assembly\Weldment.iam
Type	Inventor
Length Unit	Centimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	318,32 mm
Length Y	836, mm
Length Z	1387, mm
Properties	
Volume	8,98e+007 mm³
Mass	704,93 kg

Scale Factor Value	1,
Statistics	
Bodies	7
Active Bodies	7
Nodes	38820
Elements	19469
Mesh Metric	None
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\frodet\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > Geometry > Parts

Object Name	*Welds*	*Main shell thick:1*	*End cap thick:1*	*End cap thick:2*	*Reinforcement pad thick2:1*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Material					
Assignment	SA-516 grade 70				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	127, mm	318, mm	318,32 mm		117,68 mm
Length Y	110,06 mm	636, mm	636,01 mm	636, mm	62,595 mm
Length Z	253,91 mm	1000, mm	193,51 mm		235,31 mm
Properties					
Volume	22864 mm³	6,1723e+007 mm³	1,277e+007 mm³		5,9435e+005 mm³
Mass	0,17948 kg	484,52 kg	100,24 kg		4,6657 kg
Centroid X	-67,2 mm	-182,3 mm	-135,52 mm	-135,53 mm	-60,897 mm

Centroid Y	325,62 mm	-2,3053 mm	-1,8244e-002 mm	4,1893e-002 mm	330,95 mm
Centroid Z	1,9269e-003 mm	2,5059e-006 mm	588,86 mm	-588,86 mm	-9,7601e-007 mm
Moment of Inertia Ip1	1106,8 kg·mm^2	6,024e+007 kg·mm^2	2,6829e+006 kg·mm^2	2,684e+006 kg·mm^2	21904 kg·mm^2
Moment of Inertia Ip2	1392,8 kg·mm^2	4,4447e+007 kg·mm^2	9,059e+005 kg·mm^2	9,061e+005 kg·mm^2	25881 kg·mm^2
Moment of Inertia Ip3	419,67 kg·mm^2	2,3283e+007 kg·mm^2	3,2488e+006 kg·mm^2	3,25e+006 kg·mm^2	5250,6 kg·mm^2
Statistics					
Nodes	598	16787	5073	4260	1329
Elements	145	9824	2970	789	216
Mesh Metric	None				

TABLE 4
Model (A4) > Geometry > Parts

Object Name	*3 inch Weldneck Flange 1500 RF:1*	*Nozzle thick2:1*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	SA-516 grade 70	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	133,5 mm	67,677 mm
Length Y	124, mm	155,41 mm
Length Z	267, mm	135,33 mm
Properties		
Volume	1,2214e+006 mm^3	6,998e+005 mm^3
Mass	9,5877 kg	5,4935 kg
Centroid X	-56,09 mm	-35,249 mm
Centroid Y	483,94 mm	320,86 mm
Centroid Z	2,2844e-005 mm	-5,6781e-004 mm
Moment of Inertia Ip1	45407 kg·mm^2	18168 kg·mm^2
Moment of Inertia Ip2	53311 kg·mm^2	10053 kg·mm^2
Moment of Inertia Ip3	15378 kg·mm^2	11576 kg·mm^2
Statistics		
Nodes	8481	2292
Elements	5135	390
Mesh Metric	None	

Coordinate Systems

TABLE 5
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian

Coordinate System ID	0,
Origin	
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Directional Vectors	
X Axis Data	[1, 0, 0,]
Y Axis Data	[0, 1, 0,]
Z Axis Data	[0, 0, 1,]

Connections

TABLE 6
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

TABLE 7
Model (A4) > Connections > Contacts

Object Name	*Contacts*
State	Fully Defined
Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0,
Tolerance Value	4,1262 mm
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies

TABLE 8
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Welds To 3 inch Weldneck Flange 1500 RF:1*	*Bonded - Welds To Nozzle thick2:1*	*Bonded - Main shell thick:1 To End cap thick:1*	*Bonded - Main shell thick:1 To End cap thick:2*	*Bonded - Main shell thick:1 To Reinforcement pad thick2:1*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face	2 Faces	1 Face		2 Faces
Target	1 Face	2 Faces	1 Face		
Contact Bodies	Welds		Main shell thick:1		
Target Bodies	3 inch Weldneck Flange 1500 RF:1	Nozzle thick2:1	End cap thick:1	End cap thick:2	Reinforcement pad thick2:1

Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

TABLE 9
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Main shell thick:1 To Nozzle thick2:1*	*Bonded - Reinforcement pad thick2:1 To Nozzle thick2:1*	*Bonded - 3 inch Weldneck Flange 1500 RF:1 To Nozzle thick2:1*	*Contact Region 11*	*Contact Region 13*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	1 Face			3 Faces	2 Faces
Target	1 Face			2 Faces	
Contact Bodies	Main shell thick:1	Reinforcement pad thick2:1	3 inch Weldneck Flange 1500 RF:1	Welds	
Target Bodies	Nozzle thick2:1			Main shell thick:1	Reinforcement pad thick2:1
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Pinball Region	Program Controlled				

TABLE 10
Model (A4) > Connections > Contacts > Contact Regions

Object Name	*Bonded - Reinforcement pad thick2:1 To Main shell thick:1*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	2 Faces
Contact Bodies	Reinforcement pad thick2:1
Target Bodies	Main shell thick:1
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Symmetric

Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Mesh

TABLE 11
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Medium
Minimum Edge Length	1,24280 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	38820
Elements	19469
Mesh Metric	None

TABLE 12
Model (A4) > Mesh > Mesh Controls

Object Name	*Automatic Method*
State	Fully Defined
Scope	

Scoping Method	Geometry Selection
Geometry	7 Bodies
Definition	
Suppressed	No
Method	Automatic
Element Midside Nodes	Use Global Setting

Static Structural (A5)

TABLE 13
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 14
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	1, s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Contact	No
Calculate Results At	All Time Points
Analysis Data Management	

Solver Files Directory	C:\Master thesis Frode Tjelta\ANSYS workbench for Master thesis\Thick wall configuration - Elastic-plastic Stress Analysis - ASME VIII div.2; 2010_files\dp0 \SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	nmm

TABLE 15
Model (A4) > Static Structural (A5) > Accelerations

Object Name	*Acceleration*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0, mm/s² (ramped)
Y Component	-20594 mm/s² (ramped)
Z Component	0, mm/s² (ramped)
Suppressed	No

FIGURE 1
Model (A4) > Static Structural (A5) > Acceleration

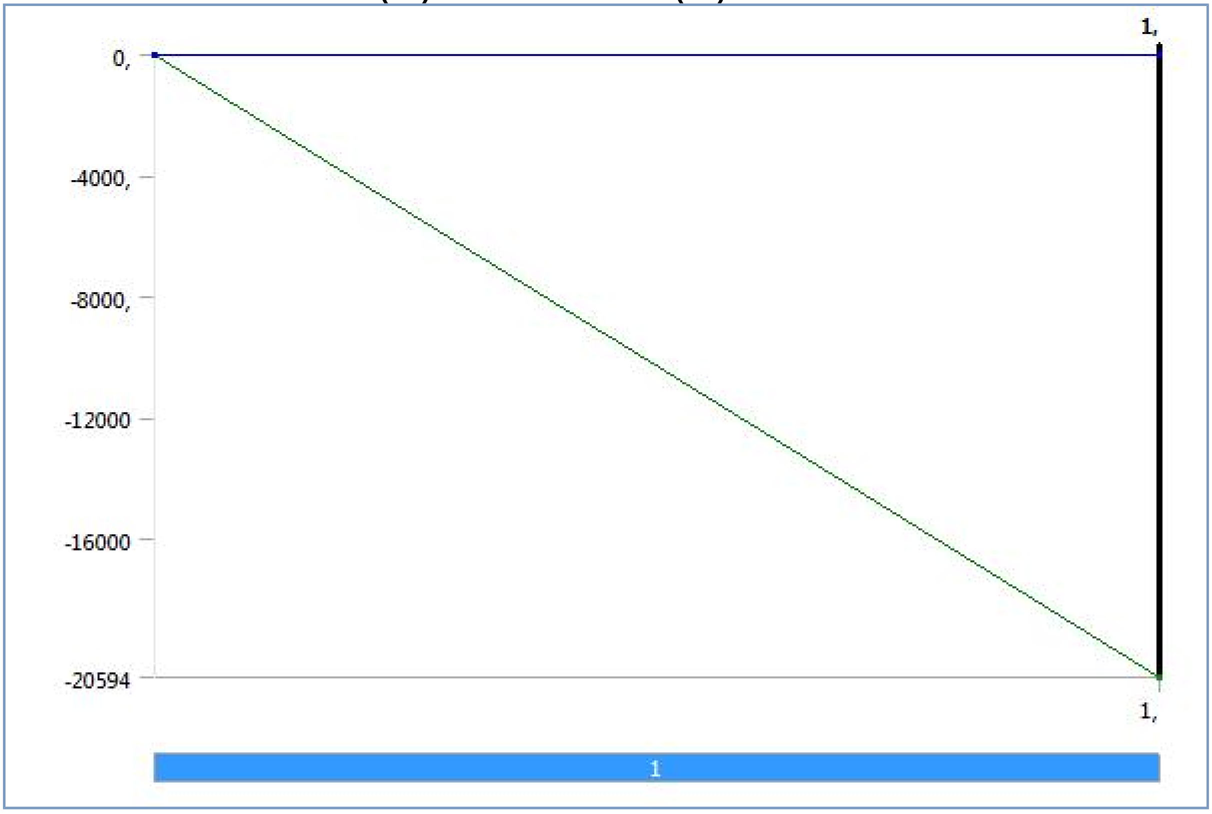


TABLE 16
Model (A4) > Static Structural (A5) > Loads

Object Name	*Pressure*	*Frictionless Support*	*Displacement*	*Frictionless Support 2*	*Force*

State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	9 Faces	16 Faces	1 Face		
Definition					
Type	Pressure	Frictionless Support	Displacement	Frictionless Support	Force
Define By	Normal To		Components		Components
Magnitude	42, MPa (ramped)				
Suppressed	No				
Coordinate System			Global Coordinate System		Global Coordinate System
X Component			0, mm (ramped)		0, N (ramped)
Y Component			Free		40500 N (ramped)
Z Component			0, mm (ramped)		0, N (ramped)

FIGURE 2
Model (A4) > Static Structural (A5) > Pressure

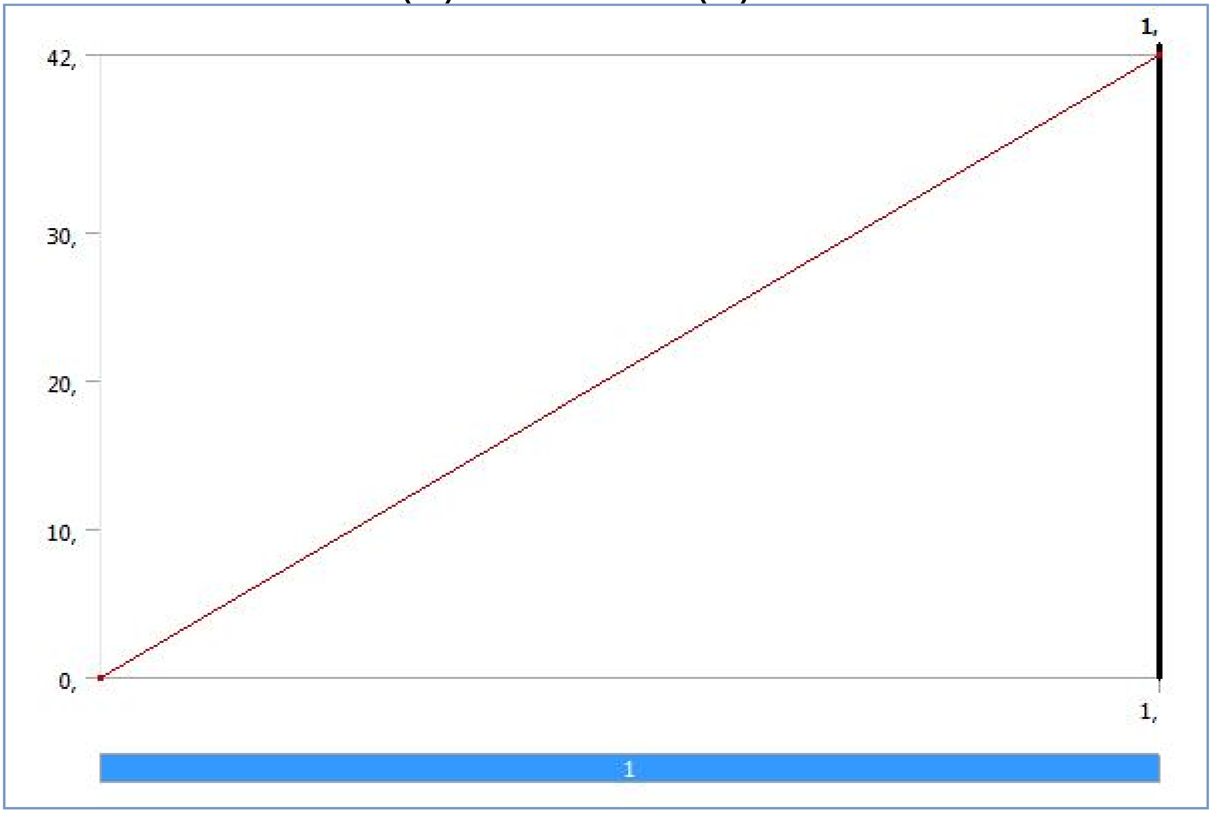


FIGURE 3
Model (A4) > Static Structural (A5) > Displacement

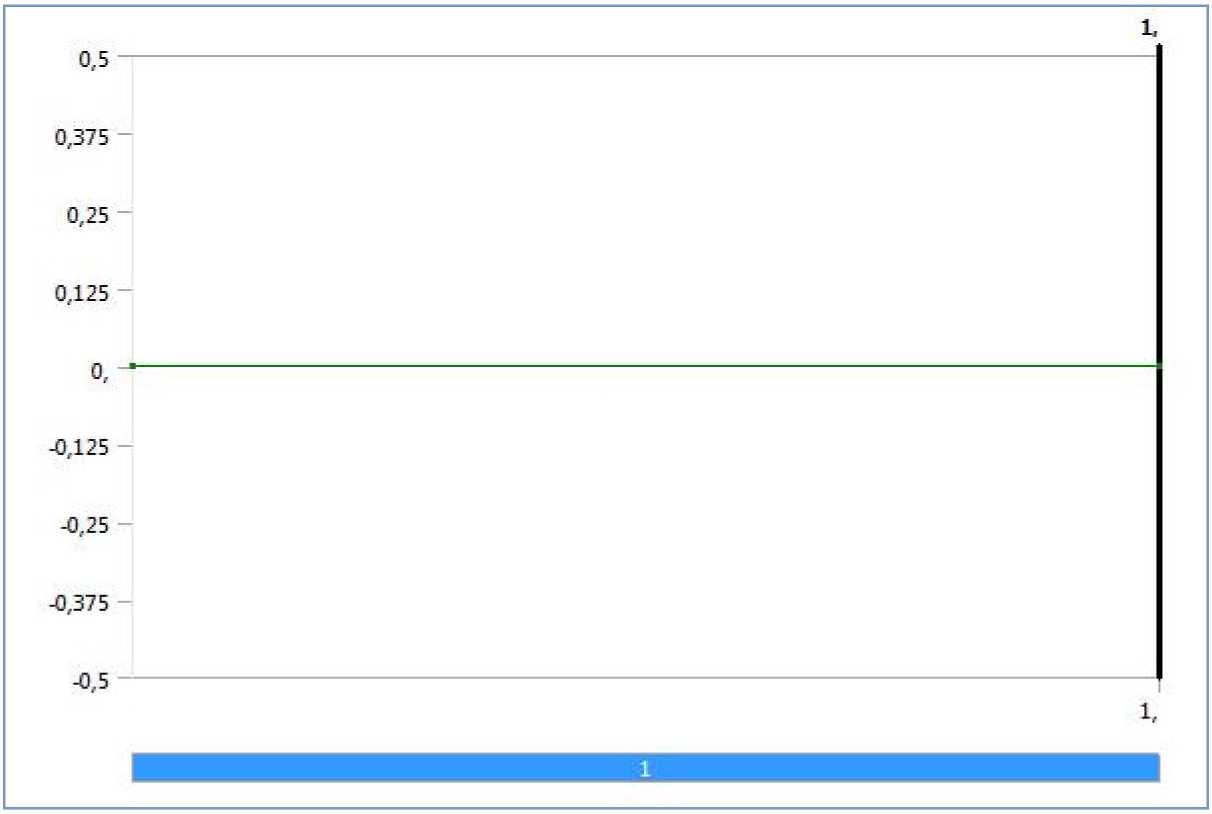
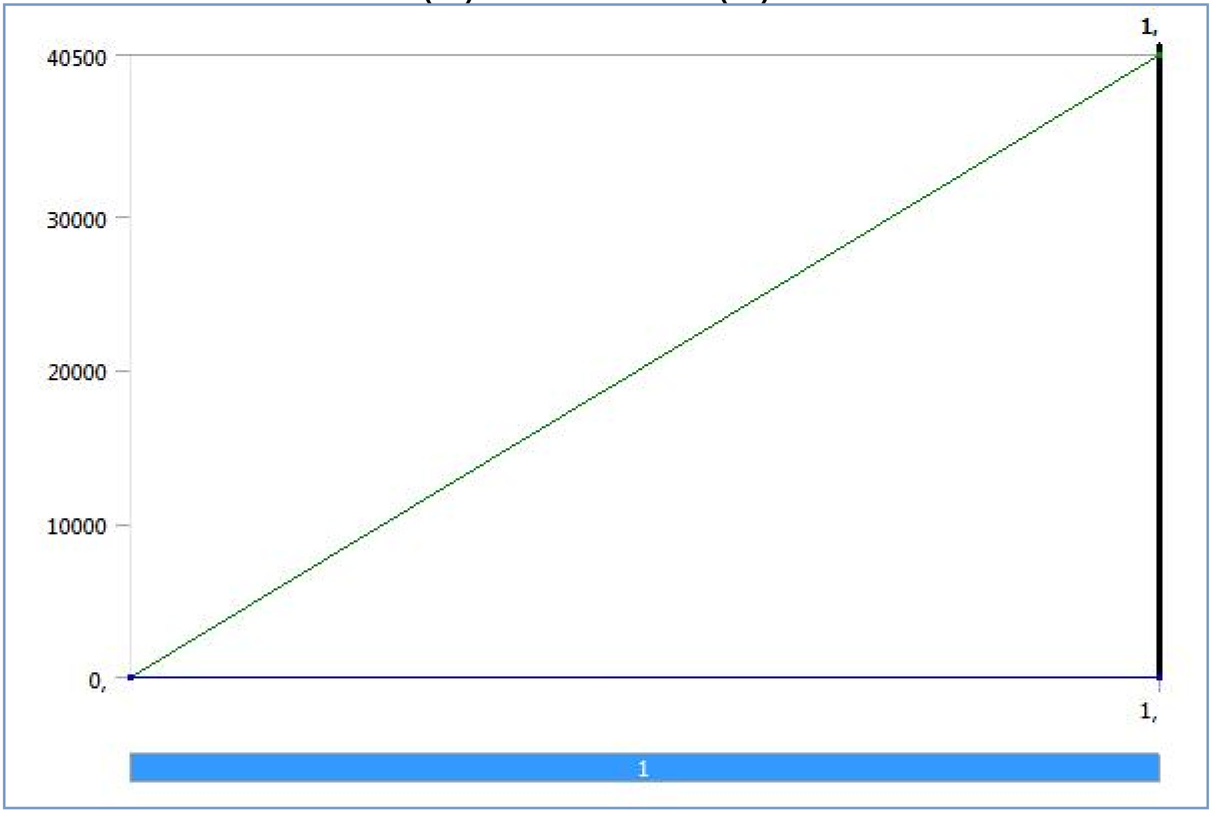


FIGURE 4
Model (A4) > Static Structural (A5) > Force



Solution (A6)

TABLE 17
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved

Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done

TABLE 18
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All

TABLE 19
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Equivalent Stress*	*Maximum Principal Stress*	*Middle Principal Stress*	*Minimum Principal Stress*	*Equivalent Plastic Strain*
State	Solved				
Scope					
Scoping Method	Geometry Selection				
Geometry	All Bodies				
Definition					
Type	Equivalent (von-Mises) Stress	Maximum Principal Stress	Middle Principal Stress	Minimum Principal Stress	Equivalent Plastic Strain
By	Time				
Display Time	Last				
Calculate Time History	Yes				
Identifier					
Integration Point Results					
Display Option	Averaged				
Results					
Minimum	0,69196 MPa	-9,8557 MPa	-42,024 MPa	-169,33 MPa	0, mm/mm
Maximum	281,36 MPa	279,47 MPa	187,11 MPa	113,22 MPa	2,0939e-003 mm/mm
Minimum Occurs On	3 inch Weldneck Flange 1500 RF:1	Welds	Nozzle thick2:1	Welds	
Maximum Occurs On	Welds	3 inch Weldneck Flange 1500 RF:1		Main shell thick:1	Nozzle thick2:1
Minimum Value Over Time					
Minimum	0,13707 MPa	-9,8557 MPa	-42,024 MPa	-169,33 MPa	0, mm/mm
Maximum	0,69196 MPa	-0,8766 MPa	-9,4008 MPa	-30,898 MPa	0, mm/mm
Maximum Value Over Time					
Minimum	108,23 MPa	104,75 MPa	36,727 MPa	22,636 MPa	0, mm/mm
Maximum	281,36 MPa	279,47 MPa	187,11 MPa	113,22 MPa	2,0939e-003 mm/mm
Information					
Time	1, s				
Load Step	1				
Substep	4				
Iteration Number	5				

FIGURE 5

Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

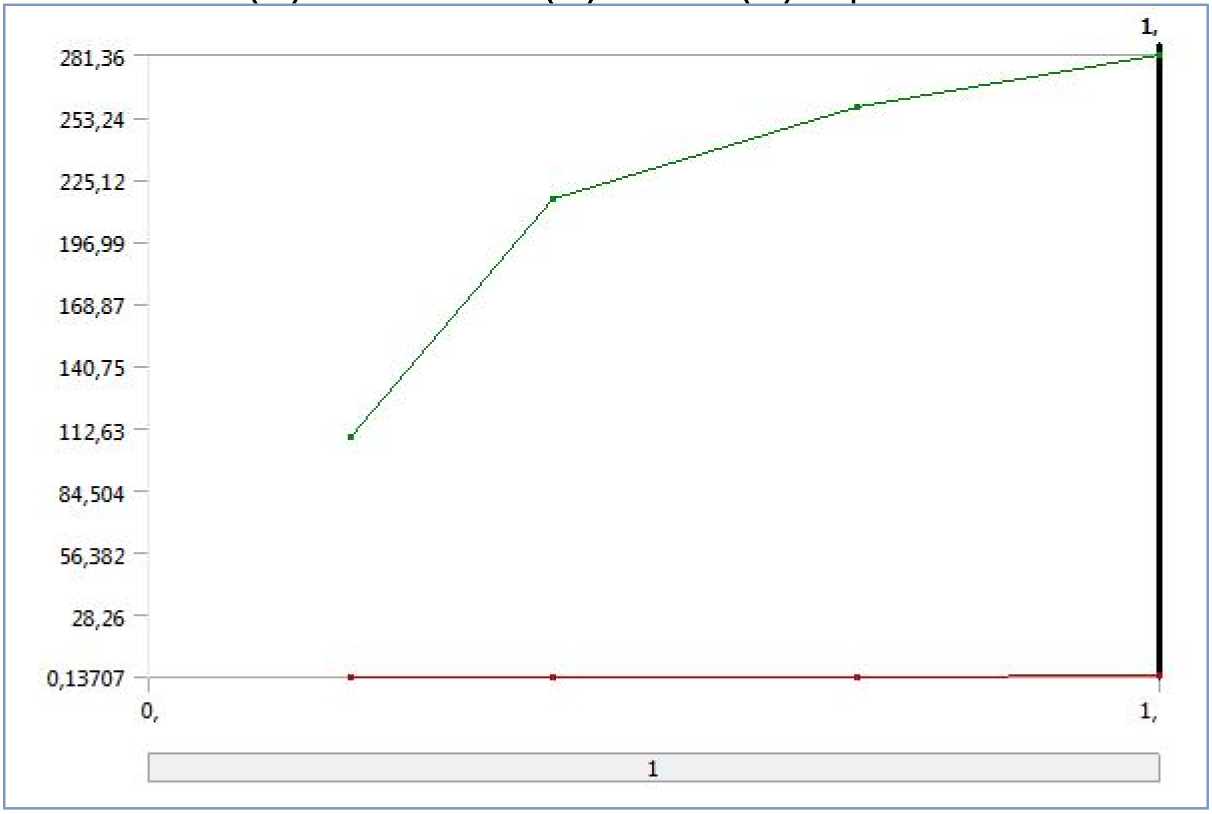


TABLE 20

Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	0,13707	108,23
0,4	0,274	216,34
0,7	0,4797	257,78
1,	0,69196	281,36

FIGURE 6

Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Principal Stress

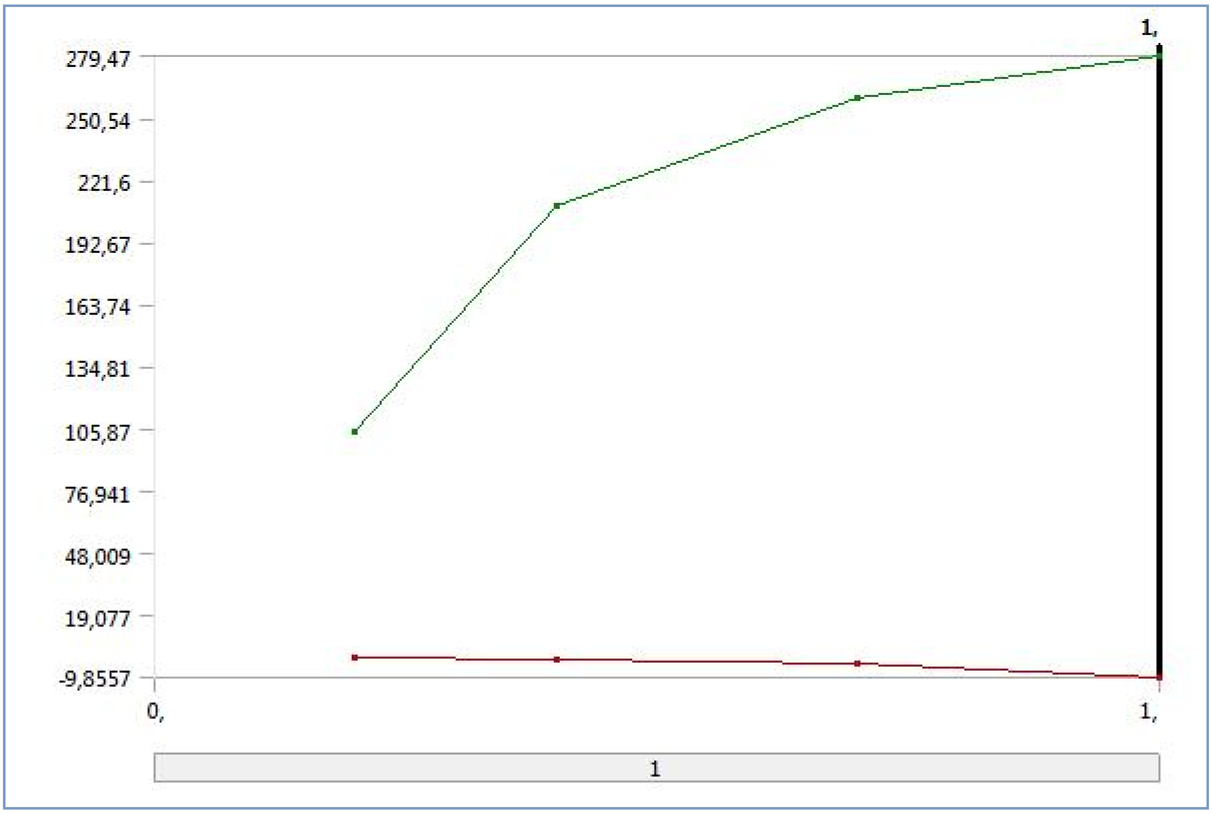


TABLE 21
Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-0,8766	104,75
0,4	-1,7542	209,41
0,7	-3,0699	260,14
1,	-9,8557	279,47

FIGURE 7
Model (A4) > Static Structural (A5) > Solution (A6) > Middle Principal Stress

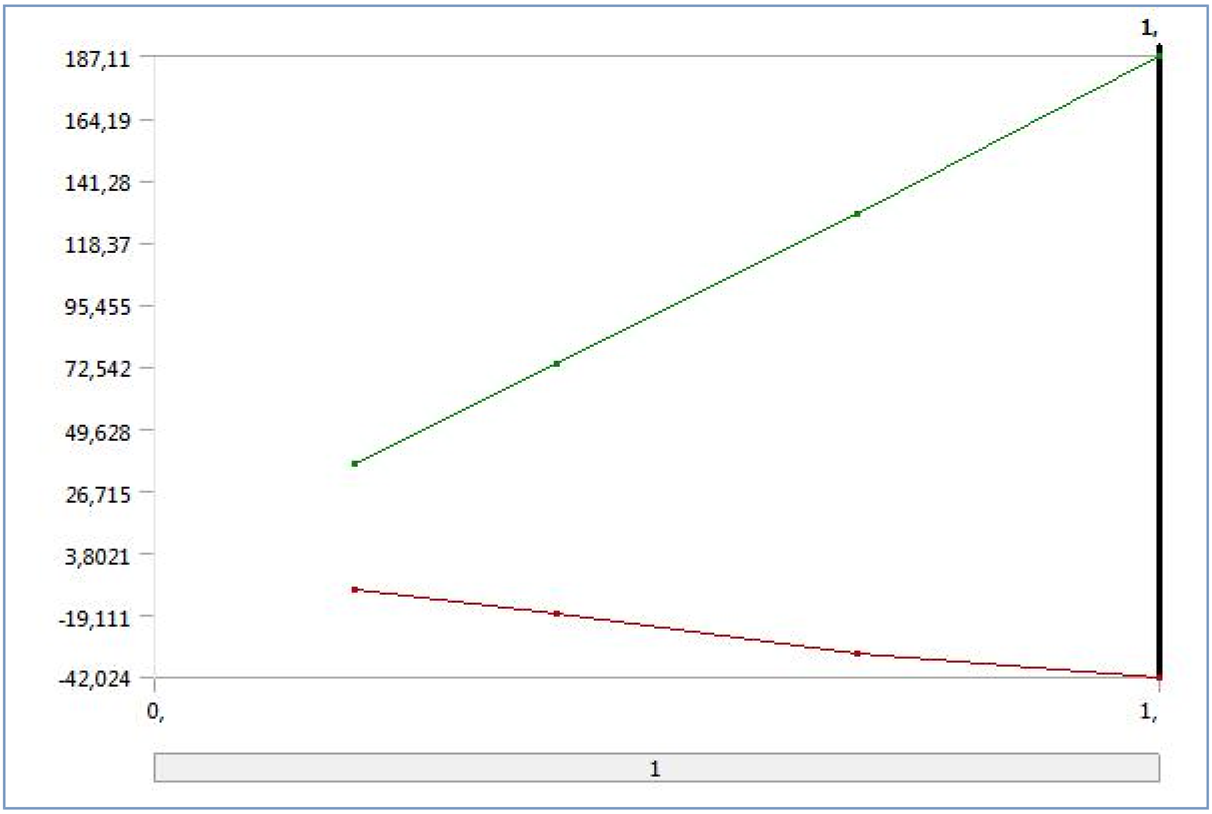


TABLE 22
Model (A4) > Static Structural (A5) > Solution (A6) > Middle Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-9,4008	36,727
0,4	-18,783	73,465
0,7	-32,883	128,89
1,	-42,024	187,11

FIGURE 8
Model (A4) > Static Structural (A5) > Solution (A6) > Minimum Principal Stress

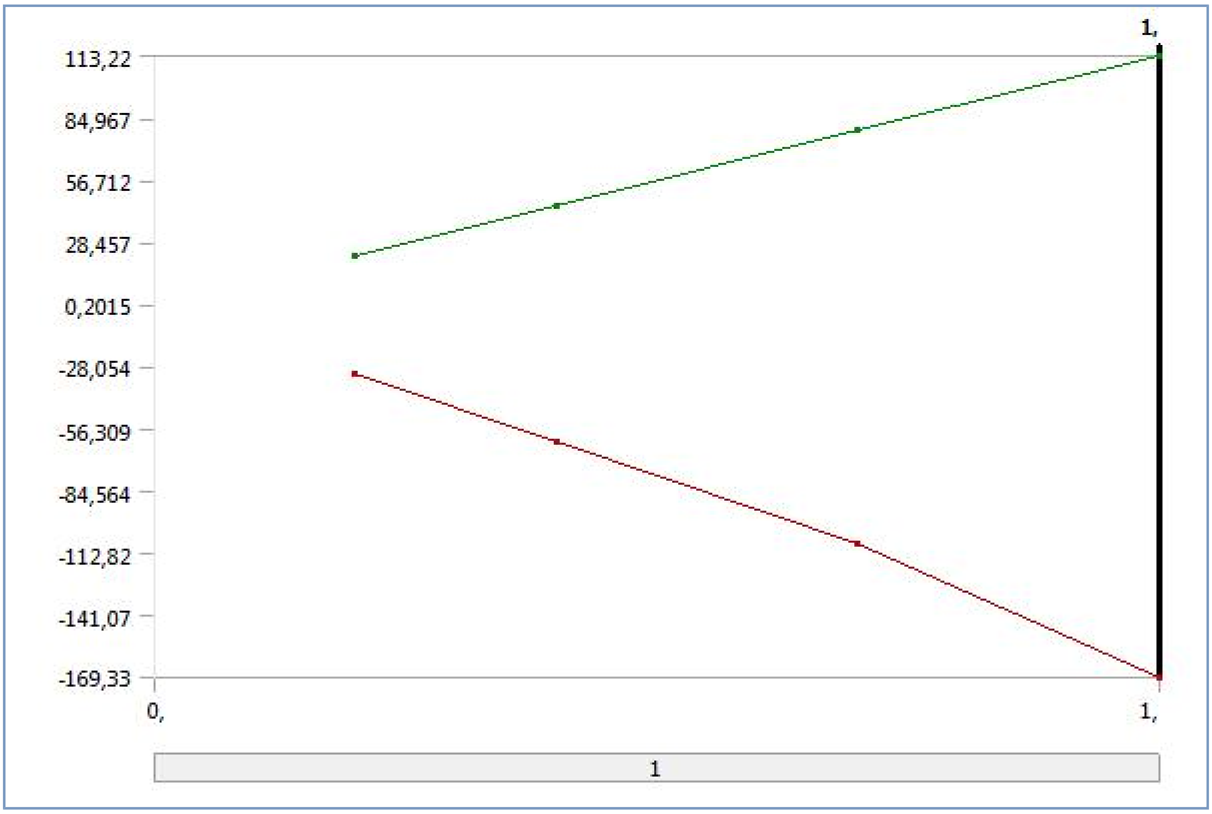


TABLE 23
Model (A4) > Static Structural (A5) > Solution (A6) > Minimum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]
0,2	-30,898	22,636
0,4	-61,849	45,338
0,7	-108,36	79,421
1,	-169,33	113,22

FIGURE 9
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Plastic Strain

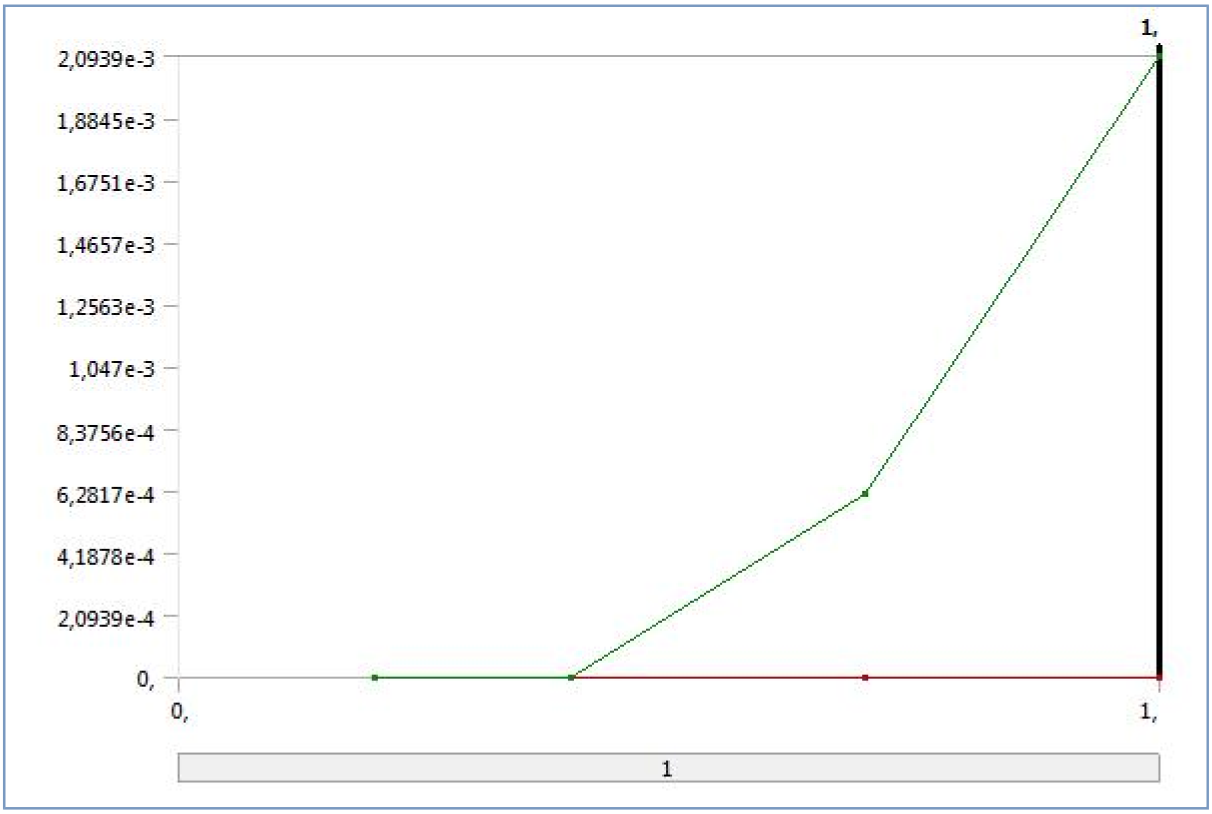


TABLE 24
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Plastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	0,	0,
0,4		
0,7		6,1689e-004
1,		2,0939e-003

TABLE 25
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Equivalent Elastic Strain*	*Equivalent Total Strain*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Equivalent (von-Mises) Elastic Strain	Equivalent Total Strain
By	Time	
Display Time	Last	
Calculate Time History	Yes	
Identifier		
Integration Point Results		
Display Option	Averaged	
Results		
Minimum	3,4598e-006 mm/mm	5,2245e-006 mm/mm
Maximum	1,4068e-003 mm/mm	3,4363e-003 mm/mm
Minimum Occurs On	3 inch Weldneck Flange 1500 RF:1	
Maximum Occurs On	Welds	Nozzle thick2:1
Minimum Value Over Time		
Minimum	6,8533e-007 mm/mm	1,0814e-006 mm/mm
Maximum	3,4598e-006 mm/mm	5,2245e-006 mm/mm
Maximum Value Over Time		

Minimum	5,4114e-004 mm/mm	5,4115e-004 mm/mm
Maximum	1,4068e-003 mm/mm	3,4363e-003 mm/mm
Information		
Time	1, s	
Load Step	1	
Substep	4	
Iteration Number	5	

FIGURE 10
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Elastic Strain

TABLE 26
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Elastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	6,8533e-007	5,4114e-004
0,4	1,37e-006	1,0817e-003
0,7	2,3985e-006	1,2889e-003
1,	3,4598e-006	1,4068e-003

FIGURE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Total Strain

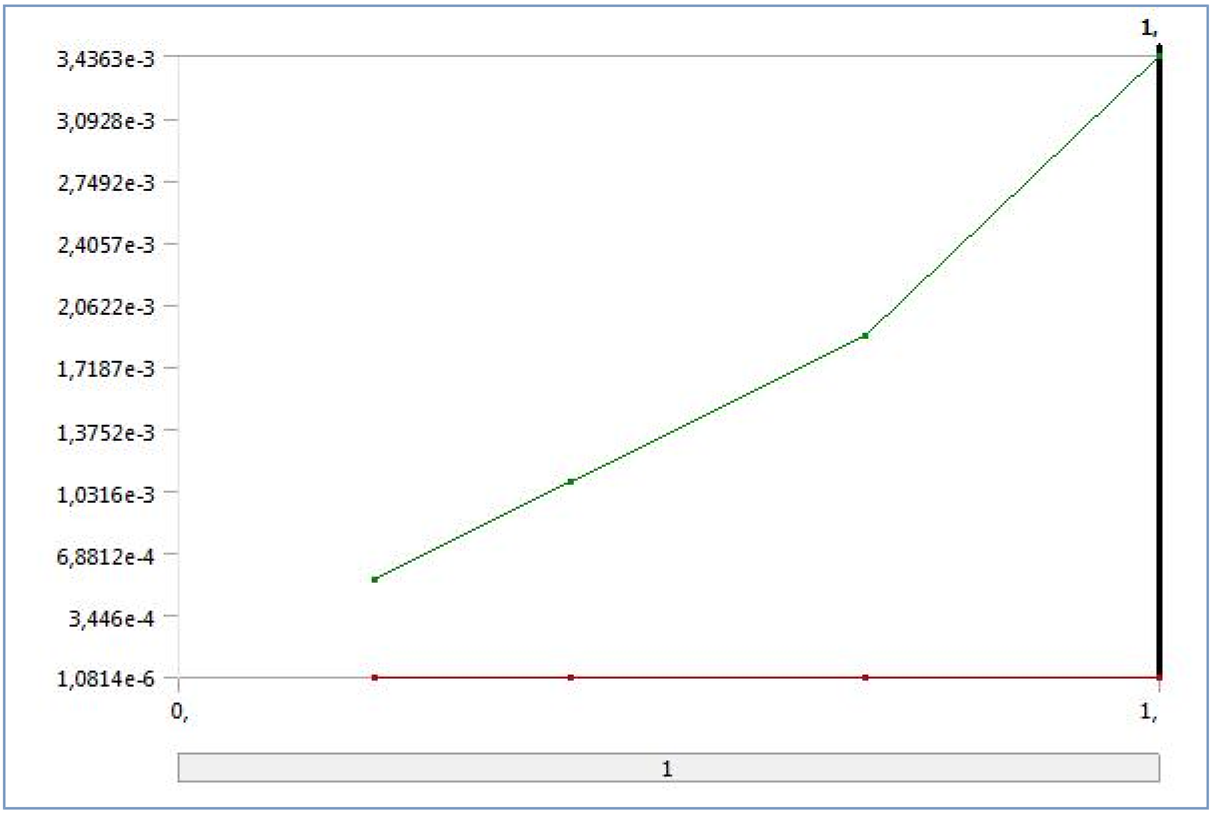


TABLE 27
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Total Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]
0,2	1,0814e-006	5,4115e-004
0,4	2,1754e-006	1,0817e-003
0,7	3,7953e-006	1,8941e-003
1,	5,2245e-006	3,4363e-003

Material Data

SA-516 grade 70

TABLE 28
SA-516 grade 70 > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 29
SA-516 grade 70 > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 30
SA-516 grade 70 > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 31

SA-516 grade 70 > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

TABLE 32
SA-516 grade 70 > Isotropic Relative Permeability

Relative Permeability
10000

TABLE 33
SA-516 grade 70 > Multilinear Isotropic Hardening

Stress MPa	Plastic Strain m m^-1	Temperature C
250	0	20
300	5.71e-003	20
350	2.42e-002	20
400	5.02e-002	20
450	7.82e-002	20
500	0.114	20
550	0.161	20
600	0.22	20
650	0.296	20
700	0.386	20
750	0.494	20
800	0.622	20