

CHALMERS



Parameterized model for stress analysis of nozzles

Master's Thesis in the Master's programme Product Development

ERIK BJÄRKBY

Department of Product and Production Development
Division of Product Development
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2011

MASTER'S THESIS IN PRODUCT DEVELOPMENT

Parameterized model for stress analysis of nozzles

ERIK BJÄRKBY

Department of Product and Production Development
Division of Product Development

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2011

Parameterized model for stress analysis of nozzles

ERIK BJÄRKBY

© ERIK BJÄRKBY, 2011

Department of Product and Production Development
Division of Product Development
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone: + 46 (0)31-772 10 00

Chalmers reproservice
Gothenburg, Sweden 2011

Preface

This work is a thesis for the Master of Science degree in Mechanical Engineering at Chalmers University of Technology. It was carried out within the master program in Product Development during the spring of 2011.

During the project a parameterized stress calculation tool for pressure vessels with large nozzles has been created. The tool can be used to evaluate the stresses using FEA and a method called stress categorization, which is described in the pressure vessel codes ASME VIII div 2 Part 5 Design by analysis requirements – *Protection against plastic collapse* and EN13445-3 Annex C Design by analysis. The tool was developed for analyzing shells of evaporators used for evaporation of black liquor in pulp mills.

The initiator of the project was Metso Power, where the project also has been carried out. Thanks to Martin Apel, Pontus Johansson and Karin Sonderegger for your support during the project and for introducing me to the world of pressure vessel design. It has been truly interesting and fun to work with you and explore fields of knowledge that sometimes were new to both you and me. I also want to send thanks to Mikael Lauth, your comments and thoughts have been truly valuable.

Finally I want to thank my examiner Lars Lindkvist and Chalmers supervisor Anders Forslund, both at the department of Product and Production Development.

Gothenburg June 2011

Erik Bjärkby

Parameterized model for stress analysis of nozzles
Master's Thesis in the Master's programme Product Development
ERIK BJÄRKBY
Department of Product and Production Development
Division of Product Development
Chalmers University of Technology

ABSTRACT

The design of pressure vessels is often strictly regulated by codes in order to ensure that the products are safe and functional. Two commonly used codes for pressure vessels operating in the process industry are the American ASME VIII Division 2 and its European counterpart EN 13445-3. Historically, formulas have been used for dimensioning pressure vessels but as the technology has evolved, computerized calculation methods have emerged as complements. For common product configuration, such as when a nozzle intersects a larger cylindrical shell, specialized software based on Finite Element Analysis (FEA) are available. Yet, problems occur when the designer wants to use product configurations that are not covered by these software. Therefore, there is a need for a calculation tool that covers such product configurations.

Within this project, a semi-automated calculation tool for pressure vessels has been developed. It handles both the case where a nozzle intersects the cylindrical part of a shell, and when a nozzle partially intersects the cylindrical, and partially an adjacent conical part of the shell. The tool consists of a parameterized CAD model which is integrated with an FEA software. The results are evaluated according to the codes mentioned above using a method called Stress Categorization and provide the analyst with a simple way to examine whether the design fulfills the requirements of the code or not. By implementing such a tool into the product development process the need for time consuming modeling, pre-processing and evaluation tasks can be significantly reduced.

The results generated by the tool, when analyzing the case where the nozzle intersects the cylindrical part of the shell, shows a rather good accuracy compared to previous published literature. The results of the second case, where the nozzle partially intersects the cylindrical part and partially the adjacent cone, were verified by a consultant with extensive knowledge in the field, who could confirm the accuracy of the model.

Key words: Automation, Pressure vessel design, Stress categorization, ASME, EN

Contents

1	INTRODUCTION	1
1.1	Background	1
1.1.1	The pulp process	1
1.1.2	Project background	3
1.2	Aim and purpose	5
1.3	Limitations	5
1.4	Report outline	5
2	THEORETICAL FRAMEWORK	7
2.1	Knowledge Based Engineering	7
2.2	Pressure vessel codes	8
3	METHODS	11
3.1	General method	11
3.2	Specific methods	12
3.2.1	Creating and parameterizing the model	12
3.2.2	Stress categorization according to codes	13
3.2.3	FE analysis definition	17
4	THE PARAMETERIZED CALCULATION MODEL	21
4.1	Definition of parameters	21
4.2	Parameter export	23
4.2.1	Geometry parameters	23
4.2.2	Analysis parameters	25
4.3	Performing the analysis	25
4.4	Stress evaluation	26
5	STRESS CALCULATIONS	29
5.1	Comparison of designs	29
5.1.1	Case 1 - Existing design	30
5.1.2	Case 2 - New design	31
5.2	Stress result verification	32
6	DISCUSSION AND CONCLUSIONS	35
6.1	Stress evaluation	35
6.2	Automation of model	36
7	RECOMMENDATIONS	39
	REFERENCES	41
	APPENDIX A	I

1 Introduction

Safe and functional products are essential within the process industry. Failing to deliver such products can result in serious accidents and large expenses. As a supplier of process equipment this also applies to Metso Power and therefore efforts are made during the design process to ensure the functionality of its products. To guide and restrict the design of pressure vessels, regional regulations have been developed by standardizing organizations providing requirements on materials, design, manufacturing, inspections, etc. Traditionally formulas within these codes are used to determine critical dimensions, but also *Finite Element Analysis* (FEA) can be used. Today specialized commercial software are available that integrates FEA with the codes, but Metso Power lacks a tool that covers some of their desired geometries of pressure vessels. As a solution, FEA of a parameterized model that easily can be adapted to each specific project would be interesting. By examining the obtained stresses from the analysis using the codes, weak areas can be identified early on in the product development process, allowing necessary design changes. Alternatively, the design can be further optimized by reducing the amount of material used, which would reduce costs.

1.1 Background

Within this section, the background of the project is described. First the basics of pulp processing which motivates the use of evaporators are presented. Then the more specific background to this project is discussed.

1.1.1 The pulp process

Metso Power develops process equipment for the pulp industry, including boilers and chemical recovery and evaporator systems. The scope of this project is to analyze a heat exchanger used within the evaporator system. This is a crucial system when producing pulp using the kraft process. The process enables the pulp mill to be almost self-supporting in terms of chemicals and energy with only wood chips used as input. A simplification of the part of the process that is interesting in this project can be seen in Figure 1. It starts by cooking the wood. Several different chemicals, all together called white liquor, are used in this step in order to separate the cellulose in the wood from residuals that are unwanted in the paper production. The residuals are usually called *black liquor* and can be burned in a recovery boiler. The chemicals in the ashes from the combustion, called green liquor, can then be recovered to white liquor and reused within the cooking process. Simultaneously the thermal energy can be used for producing electricity, district heat or vapor which can be used in other parts of the process.

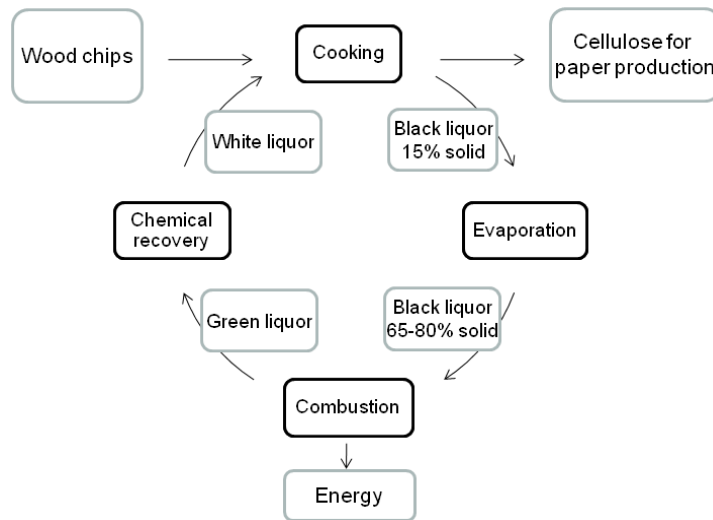


Figure 1 Basics of the kraft process

The black liquor, directly from the cooking, usually contains around 15% solids and 85% water. To get the most out of the recovery process the black liquor needs to be condensed, usually to levels where the solid content reaches between 65% and 80%. To obtain this, evaporator plants consisting of a number of heat exchangers linked to each other in a complex system are used. In the main heat exchangers of the system (also called evaporators), vapor from the recovery boiler is let in and flows between several vertical tubes within the vessel. This can be seen in Figure 2. The black liquor is pumped to the top of the evaporator and falls inside the tubes while vapor flows on the outside. This causes the water in the black liquor to evaporate which increases the solid content of the black liquor. The dimensions of the evaporators vary depending on the plant's capacity, but are today up to 11 meters in diameter and 25 meters in height.

The black liquor is then accumulated in the bottom of the evaporator before it is pumped to the next evaporator, which operates at a higher temperature and enables further water content reduction. This process is repeated through several evaporators until the desired solid content is reached.

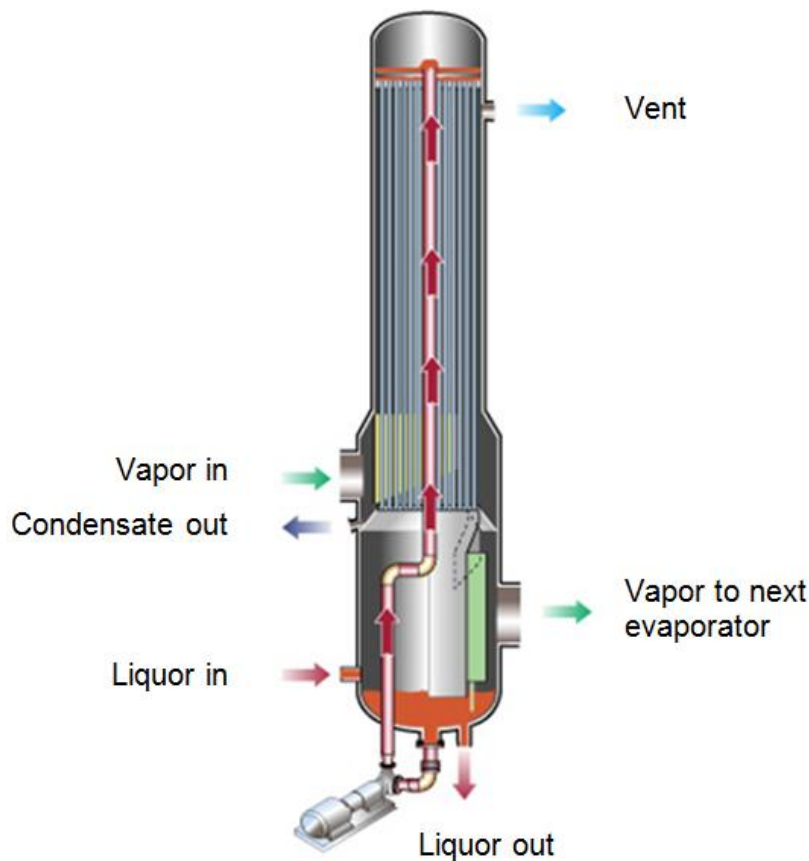


Figure 2 The basic principle of an evaporator.

The black liquor vapor is reused as heating medium in the next evaporator and most commonly seven serial evaporators are used. These are numbered from one to seven where evaporator number one operates at the highest temperature and pressure, and number seven at the lowest. The temperature and pressure difference is what force the vapor to run through the system from evaporator number one to seven. To make the process as effective as possible, the black liquor usually enters the system at evaporator number four and flows in the same direction as the vapor through number four to seven. Then it flows in the opposite direction compared to the vapor through evaporator three to one. This means that high temperature vapor is used in the energy intensive last evaporation step, where the last portion of the water content in the black liquor is reduced.

1.1.2 Project background

During recent year a trend has been seen in the business of producing evaporators where an increased capacity of the evaporator plants is wanted by the customers. As an example Metso is, in time of writing, designing the world's largest evaporator line. As the capacity increases, the plants and the pressure vessels get larger and new problems occur during the design phase with new questions to be answered. New design concepts of internal components of the evaporators have now made it possible to reduce the height of the lower part of the evaporators, see Figure 3, which gives several benefits. For example, if the amount of used material can be reduced, the handling of the product would be easier and the installation would be facilitated. These factors together can contribute to reduced cost resulting in a product that is more profitable.

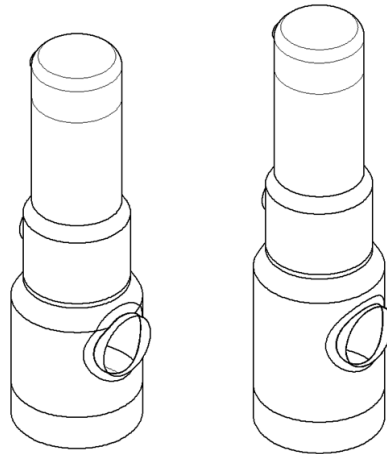


Figure 3 New conceptual and present evaporator design.

To be able to realize a major design change where the amount of material is reduced, there are some obstacles to overcome. There are still some uncertainties associated with the process data that is needed for validating the concept, such as temperatures, pressures and vapor velocities that are crucial for the function of the evaporator. Additionally, more in depth knowledge is needed about the stress distribution which needs to be evaluated according to the pressure vessel codes, in order to make sure that the product is safe and functional while operating.

To ensure that the calculation tool is usable and compatible with the design process that is utilized today, it needs to fulfill some requirements stated by adjacent systems. The mechanical design process of today's evaporators starts with process calculations to ensure the functionality of the plant. The resulting data is used as input for a computer application that calculates the basic dimensions for each specific evaporator. To make sure that the design fulfills the requirements of the codes, critical pressure vessel structures are analyzed using specialized software. These use either formula calculations or the *Finite Element Method (FEM)*. The analysis shows whether the design fulfills the requirements and the next process step can be taken, or if the product fails to do so and further development is needed. In the later case, one enhancement could be to increase the thickness of a certain plate, and as a consequence the stresses are reduced. The calculation tool developed in this project is intended to replace the specialized analysis software that is used today, since it does not support the geometry of the new conceptual design. Yet the same in- and output parameters should be used.

The idea of the calculation tool is to start by entering the input parameters into a user interface configuration sheet. From this sheet the parameters are exported to a geometry-generating tool that creates a *Computer Aided Design (CAD)* model which is the base for the FEA. When the analysis is completed the stresses are evaluated according to the codes which give a result of whether the design fulfills the requirements or not. The process is schematically described in Figure 4.

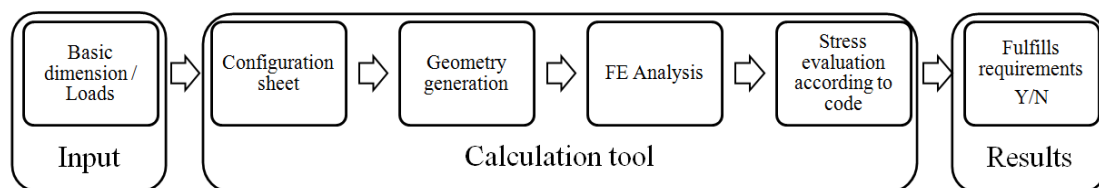


Figure 4 Process steps in the calculation tool.

1.2 Aim and purpose

The purpose is to investigate the possibilities for easily calculating and evaluating the stresses within a pressure vessel with a nozzle, when design by formula according to codes is not applicable. The aim of the project is therefore to develop an automated calculation tool for pressure vessels with nozzles. The tool should be based on a CAD model used for FEA. The geometry and analysis parameters must be easy to change and the model should therefore be parameterized. The calculated stresses should be evaluated according to the American pressure vessel code ASME VIII div 2 and its European counterpart EN13445-3.

1.3 Limitations

Internal parts, such as tubes and the droplet, will not be taken into consideration during the project. In the case of vacuum within the vessels, no buckling analysis needs to be done, since this failure mode is considered by other parts of the codes. Yet, the model must still have the ability to calculate the static stresses that occur.

The evaluation of stresses should be done according to the pressure vessel codes ASME VIII div 2 Part 5 Design by analysis requirements – *Protection against plastic collapse Elastic stress analysis method*, or EN 13445-3 Annex C Design by analysis – *Method based on stress categories*. No nonlinear analysis should be used since these two methods are based on linear calculation models.

All software that are used must be available at the company today. The modeling and analysis must therefore be done in SolidWorks and its solid mechanics analysis tool SolidWorks Simulation. The company's intention is to use the software even more in the future, which is a good base to ensure that the parameterized model will actually be utilized. However, it is outside the scope of the project to incorporate the model within the *Product Data Management* (PDM) system.

1.4 Report outline

In Section 2 - *Theoretical framework* previous work within Knowledge Based Engineering and its integration with FEA will be described. The research associated with the EN and ASME code will also be described to present what previously been done and in which direction the research is heading. In section 3 - *Methods* the methodology that was used is described. Both the general method for developing the calculation tool is presented and more in depth methods for developing the parameterized model. It also contains an introduction to stress categorization and how the FE model is defined. In section 4 - *The parameterized calculation model* the set up of the model is described and how parameters are exported and imported. The chapter also describes how the calculation model works. In the next section, 5 - *Stress calculations*, the developed tool is used for analyzing the existing and the new design concepts. A comparison is also done with previous work in order to validate the model. Subsequently there is a discussion in section 6 - *Discussion and Conclusions* of the used methods and the results. To sum up, some recommendations for future work and improvements are given in section 7 - *Recommendations*.

2 Theoretical framework

The creation of a parameterized calculation tool involves interdisciplinary work - built mainly on two fields of previous research; *Knowledge Based Engineering* (KBE) and pressure vessel design according to codes. Both of these two areas involve FEA which can be seen as the link to connect them to each other. In the section below the background theory and previous work within the area of the project will be presented.

2.1 Knowledge Based Engineering

Benefits that can be achieved by using Knowledge Based Engineering systems are for example shorter development cycle time, significantly increased engineering productivity, rapid evaluation of different design alternatives (which facilitate concurrent engineering strategies) and connections between the isolated technology islands that different software with incompatible interfaces constitutes (Saxena & Irani, 1994). The basic principle is to make use of knowledge gained from previous projects and reuse it in new ones, or as Stokes (2001) defines it

“The use of advanced software techniques to capture and re-use product and process knowledge in an integrated way”

From an operational point of view, KBE could be used to automate routine engineering tasks by the use of computer aids. Also Chapman and Pinfold (1999), describes that the aim of the KBE systems is to capture information about processes and products in such a way that businesses can model the engineering design process and automate it. The design process of each product can then be documented and information about the product can be captured in a *product model*. Within the model, information is saved that can not only answer the question “what” the design intent was when creating the model, but also “why” and “how” it came to be that way (Chapman & Pinfold, 1999). It is outside the scope of this project to automate the entire design process of evaporators. However, by automating the FE analysis of the design process, a contribution has been done in the right direction. As described previously in section 1.1.2, some work has already been done at the company by automating the calculation of basic dimensions, and this project will be the next step.

The integration of CAD and FEA tools has been a subject of research for a few decades. One early identified issue was that CAD and FEA tools did not utilize the same product model within the product development process. This means incompatibility and limitations in the information transfer between the different tools. Several attempts have been made to overcome this problem by different strategies where the information is more closely connected to only one model during all process steps (Chapman & Pinfold, 1999; Gujarathi & Ma, 2011; Saxena & Irani, 1994). By using such a tool the designer will be more free to choose software and is not restricted to only one system. Within this project the same model is used for geometry modeling and FEA but it could be advisable for future developments to use the model in other steps during the product development cycle.

Within the FEA field of KBE the focus has previously been on the pre-processing of geometry models prior to analysis. Chapman (2001) describes how a car body structure is pre-processed using a knowledge based engineering system. Also Boart, Andersson and Elfström (2006) focus on the pre-process phase when designing a fan

for a commercial aircraft engine. Studies in post-processing tools for design optimization are rather uncommon. Novak and Dolsak (2008) present a model where the FE analysis results are automatically post-processed, identifying design optimization opportunities. These optimization results are then implemented and the analysis rerun. A similar project is described by Pinfeld (2004) where the automated design optimization of an engine-mounting bracket is described. This provides a new dimension of KBE where the system could be said to have some artificial intelligence, not just performing routine tasks but also is creative and gives new design proposals. This seems to be the frontline of today's research within KBE and automated product development processes. Although the calculation tool created in this project is not intended to propose design modifications, such features would be of interest to include in future work. The calculation tool presented in this report identifies weak areas of the design but still relies on the engineering knowledge of the analyst to make design changes.

2.2 Pressure vessel codes

The history of pressure vessel codes dates back to the early 20th century. After several devastating accidents within the pressure vessel industry a demand for design laws and regulations emerged (Chuse & Carson, 1993). Traditionally, to determine the acceptability of the design, formulas and charts based on analytical solutions and empirical data have been used. This is today is called Design By Formula (DBF) within the EN code and also described by ASME VIII Division 1. In the 1960's an alternative to DBF was established known as Design By Analysis (DBA). It served as a complement for the design cases which were not covered by DBF, and was based on a method where stresses were classified into different categories. The method was successful within the pressure vessel industry and was incorporated within many other national codes (Baylac & Koplewicz, 2004). In 2002 a harmonized European standard was presented for pressure vessels called EN13445-3. Two alternative methods for DBA were presented, where one was based on the traditional method of stress classification. The second method, that was new and based on nonlinear analysis, was called *Direct Route*. Since *Direct Route* achieves benefits from more advanced design methods and gives more accurate results, the evolution of, and the documentation around, the stress categorization method has stagnated (Baylac & Koplewicz, 2004).

Since the ASME code is most commonly used worldwide, it also affects the literature within design by analysis, which also focuses on ASME. There are some uncertainties associated with the stress categorization procedure and to minimize them, general methods for classifying stresses are described by several authors (Lu, Chen, & Li, 2000; Roche, 1989; Seshadri, 2009). Suggestions are given in some previous work on how to make use of FEA for evaluate stresses according to stress classification. Hollinger (2000) identified the need for guidelines of how to use FEA with 3D elements to fulfill the requirements of the code and suggests that a more profound description of how to implement FEA should be added to the ASME code. Since the classification procedure can be difficult to implement for 3D elements, Seshadri (2009) suggests a method for easier classification of the stresses.

Since it is common that pressure vessels consist of relatively small material thicknesses and large diameters, they can be modeled with shell elements. This has been further investigated and it was found that compared to using shell elements, the use of more resource demanding 3D elements in pressure vessel applications could

not be justified (Porter & Martens, 1998). But the results of the FEA can also vary depending on which software is used. For shell elements, different FE software have been compared and some differences were identified (Porter & Martens, 1996). Continued work has been done where specific guidelines have been developed of how to interpret the code, and how to analyze pressure vessels with a nozzle according to the ASME code (Porter, Martens, & Caldwell, 2008; Porter, Martens, & Marcal, 1999).

In 2007 a major revision of the ASME standard was released in order to make it more competitive in comparison with other standards. It was restructured and adapted to modern technical tools (for example FEA) in order to make it more user friendly. Some enhancements were made to facilitate the FE integration to the code, but within the design part of the code most of the effort was made on improving the Direct Route methodology (Pastor & Osage, 2007). An identified trend within the literature is that more sophisticated nonlinear methods will become more common in the near future (Baylac & Koplewicz, 2004; Porter, Reinhardt, & Martens, 2001). This was also confirmed by the ASME code that admits that in the case of complex geometries and complex load cases, stress categorization can result in ambiguous results and nonlinear methods within DBA would give more accurate results.

3 Methods

In this project several different methods on different levels are used. The following chapter will be divided into two parts where the first one focuses on the overall method for creating the calculation tool, and the second one describes more step-specific methods more thoroughly.

3.1 General method

Integrating FE analysis with stress evaluation is a relatively new area of knowledge for the company and consequently, there are no company specific guidelines for developing such a calculation tool. As previously mentioned, specialized software for similar tasks are available, but they lack the ability to analyze the geometry of the new design concept. In addition there is a controversial question whether the stress categorization methods in the codes are suitable to use when analyzing FEA results since there is a risk for ambiguous results. With this background there is generally a weak documentation of how to evaluate the stresses by the use of the specific software that limits the project. The used methods are therefore a mix of methods that are available for generation of geometry, analysis design and evaluation of the calculated stresses.

The holistic approach to the project was to develop the different subsystems of the calculation tool counterflow. This in order to understand what requirements the process steps downstream put on the previous steps. A schematic figure of the process can be seen in Figure 5. To grasp what was needed to evaluate the stresses according to the codes, the required input to the code was needed to be identified. The next step was to examine how to get this information from the analysis results and what tools the FE software, which limits this project, provided to enable export of such information. When the abilities of the FE software were clear and the parameters required by the code could be exported, new limitations of the calculation tool were found and hence, these needed to be overcome and several iterations of step A were needed until satisfying results were obtained. This iterative method of working was practiced in all steps where communication between different software was needed, see step B and C in Figure 5.

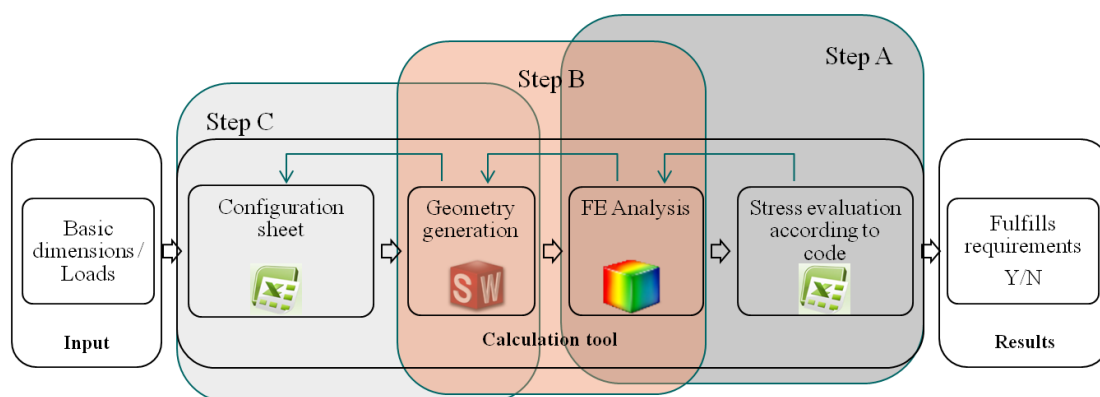


Figure 5 Used software and method for understanding downstream requirements.

The methodology described above was first used for the case where the nozzle is situated in the cylindrical part of the shell. This enabled verification of the results by comparison with similar, previous studies. Since no documentation is available for the case where the nozzle intersects both the cylindrical and the conical part of the shell, some modifications of the first model was made to enable the parameterizing and to facilitate the stress categorization process.

3.2 Specific methods

For the specific needs of this project there was no comprehensive methodology available of how to develop this type of calculation tool. Thus, several different methods needed to be used. The choice of methods was based on project specific circumstances and is presented below for the different steps within the development process.

3.2.1 Creating and parameterizing the model

Early in the project it was stated that the software that must be used were the ones presented in Figure 5. One limitation was that SolidWorks should be used for geometry generation and strength calculations. This software is a feature-based CAD tool that supports parametric modeling. It enables the user to define design variables that links dimensions of different features to each other. With the prerequisite that the model is properly defined, the product design can easily be changed by just modifying these parameters. This gives great freedom and opportunities to examine alternate design solutions accurately and efficiently (Kuang-Hua 2001). To facilitate the automation process the software Automate Works was used for exporting and controlling the calculation process. Since this software is closely connected to Excel it seems natural to use Excel for parameter input and also partially for stress evaluation. Other benefits with Excel is its availability and calculation potential both in the preparations of creating the geometry and when evaluating the stresses. It was also preferred by the project-initiating division of the company that Automate Works would be used, in order to examine its benefits and limitations and assess its feasibility for future products.

Some company specific-guidelines were needed to be followed when modeling the geometry, something that affected the methods within this step. The geometry was created based on a 3D sketch where all characterizing dimensions are represented by controlling sketch segments (lines, circles, etc.) and planes. A 3D sketch is a set of entities that works as a 2D sketch with the difference that, instead of constantly working in one plane within the sketch, any arbitrary plane can be chosen or created as work plane. By using this strategy, the amount of created 2D planes can be kept to a minimum, which reduces the complexity of the model. If, for example creating an extruded feature connected to the datum plane top with an extension of x mm, an offset plane was created at a distance x and the feature was then set to be extruded up to this plane. The length of the extruded feature can then be varied by just changing the distance between the planes controlled by the variable x . The sketch was based on three orthogonal planes to which all other entities are related. The intention was that the parameterized dimensions would be based on these planes as far as possible, in order to reduce the information content of the model as suggested by Silva & Chang (2002).

Features were created based on the 3D sketch. Already at this stage it was necessary to consider how the analysis and stress evaluation could be facilitated as much as possible. Therefore, the pre-process work started already here, with some simplifications of the geometry that would make the mesh procedure easier, requiring less resources during both the mesh- and calculation steps. Although some information about the model was lost while making such simplifications, the model was considered to be adequate since it was to be used only for FE analysis. Consequently, the model is not useful for other purposes, such as serving as a base for production drawings.

3.2.2 Stress categorization according to codes

As previously described there are several different ways of fulfilling the requirements of both the ASME and the EN codes. Historically Design by Formula is the most widely used method where critical dimensions are calculated using formulas for some standard configurations. However, these formulas are not always applicable and therefore design by analysis serves as a complement within both the ASME and the EN code.

Since the calculation tool must meet the requirements of the ASME VIII division 2 Part 5 and EN13445-3 Annex C, both were needed to be taken into consideration. However, this is not a problem since the parts of the codes that cover stress categorization are only slightly different (Baylac & Koplewicz, 2004). These differences affect mainly the fatigue parts of the codes which should not be considered within this project. A hierarchical structure of the codes for unfired pressure vessels can be seen in Figure 6.

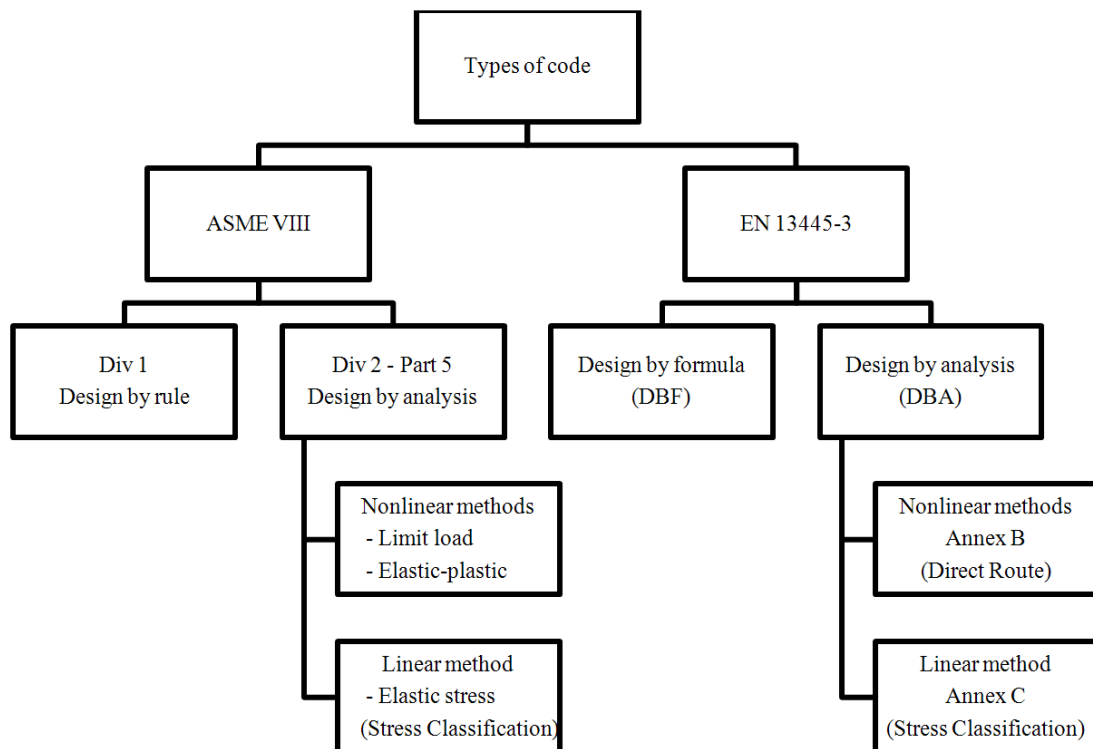


Figure 6 Hierarchical structure of the ASME and EN codes for unfired pressure vessels.

ASME VIII division 2 Part 5 states the design by analysis requirements and includes several methods for evaluating the design against four different failure modes. Methods are described for evaluating against plastic collapse, local failure, buckling, and cyclical load. The “Stress classification” method, which this project is limited to, is included in the analysis for plastic collapse and thus, this is the only failure mode that will be considered. The other failure modes must be analyzed explicitly but are out of scope for this project. Several different methods are suggested by the code for each failure mode, but the code does not give any recommendation of which method to use and leaves it to the user to determine which is the most suitable depending on the specific circumstances (ASME, 2010).

Different material models are used within the different underlying methods when the design is evaluated against plastic collapse, see Figure 7.

- For the *elastic stress analysis method* a linear elastic model is used. This means that the stress in the material is assumed to be linearly proportional to the strain also above the yield stress. This method is also called stress classification and will be utilized in this project.
- The *limit-load method* assumes an elastic perfectly-plastic material where the stress is assumed to increase linearly proportional until the yield stress. Thereafter the stress is constant, independent of the magnitude of the strain. By calculating at what loads the structure will collapse it can be dimensioned.
- The most advanced method is the *elastic-plastic stress analysis method* which can also utilize a nonlinear elastic-perfectly plastic material model. This is the most accurate way to model the material behavior but also the most resource intensive (Sunnarsjö, 1999).

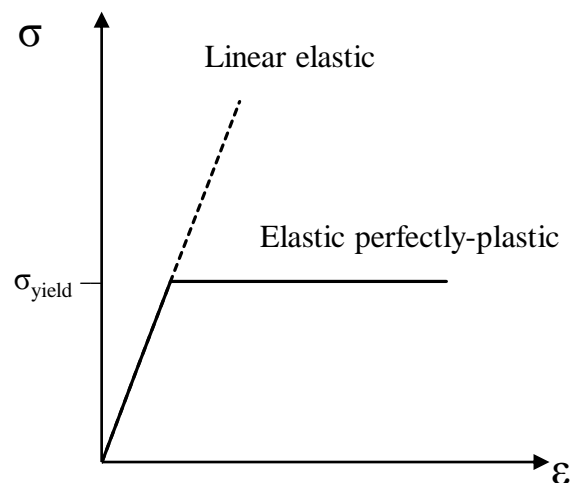


Figure 7 Stress – strain curves of linear elastic and elastic perfectly-plastic material models.

The “Stress categorization” or elastic stress analysis-method is used since it is a relatively straightforward way to obtain a result. Though, it should be kept in mind that the results of the categorization process may end up being ambiguous (ASME, 2010). This is especially true for more complex geometries and load cases, where a considerable portion of judgment and knowledge is needed to correctly categorize the stresses.

EN13445-3 has a somewhat different structure of the code. Within the design by analysis strategy the method *Direct Route* can be used, which is the EN counterpart of the nonlinear ASME methods listed above. Just as in the ASME code, direct route can be used as an alternative to stress classification.

When the elastic stress analysis or Stress Classification method is used, the calculation model is the same for both ASME and EN. The computed stresses in the structure, called equivalent stresses, are assigned to different categories which are compared to material specific values that conservatively represent the maximum allowable stresses. While staying within these limits no plastic collapse will occur. A guideline from ASME of how to evaluate the stresses in each point is visualized in Figure 8.

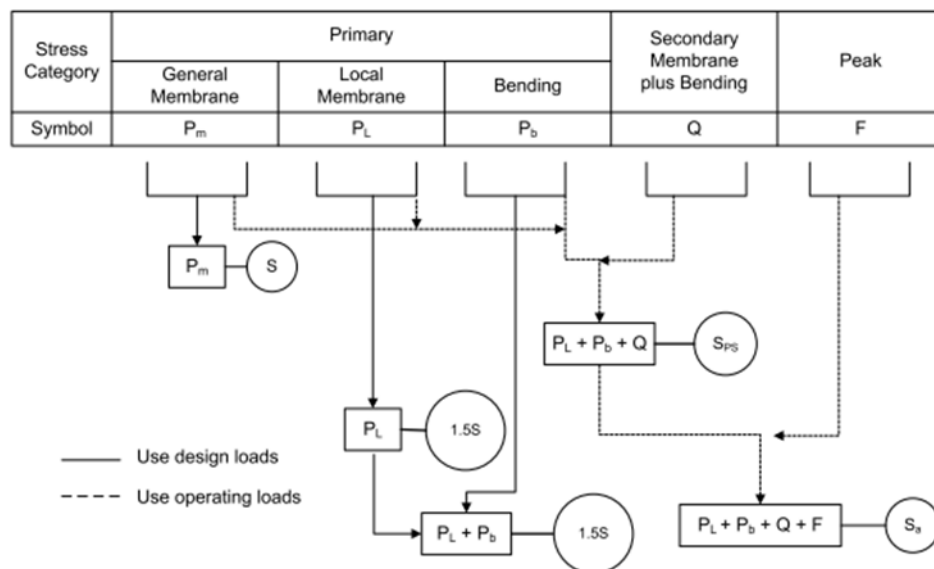


Figure 8 Guidelines for how to evaluate the different stress categories. Adapted from ASME (ASME, 2010).

Initially it needs to be understood what loads that will act on the geometry. There are several different load cases that need to be analyzed, taking into account different scenarios during the lifetime of the product. Usually two cases are analyzed; design loads and operating loads. Design loads are extreme loads that are present intermittently, if present at all. As an example it represents the cases when there is a disturbance in the evaporation process and the inner pressure is significantly increased or reduced. This means that if both an internal over- and under pressure should be considered as design loads, two calculations and evaluations are needed. Operating loads are the continuous loads that are present when the evaporation process is running on a daily basis. By using the more extreme design loads when analyzing, it is ensured that the design will also fulfill the requirements of the operating loads.

Based on the loads, elementary stresses (σ_{ij}) are calculated at all points that are in scope of the analysis. In a three dimensional case these elementary stresses consist of a vector of six components of normal- and shear stresses, which should all be classified as membrane or bending stresses.

Membrane stress is the part of the elementary stress that is constant over the thickness, representing the average value of the stresses. The second type, bending stress varies linearly over the thickness. Together these two form the linearized stress

which can be seen in Figure 9. In addition there are sometimes a nonlinear stress part which is critical in case where fatigue needs to be considered, however since fatigue is out of scope of this project it is hereafter omitted (ASME, 2010). As can be seen in Figure 9 the total linear stress varies along the cross section but the maximum value will either be found on the top or the bottom surface and therefore both of them need to be evaluated.

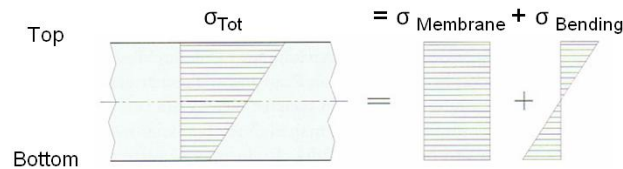


Figure 9 Decomposition of linear stresses over a cross section. Adapted from Sunnersjö (Sunnersjö, 1999).

After decomposition of the stresses into membrane and bending parts they are classified into one of the following categories.

Primary stresses (P) are the most critical stresses for the design, meaning that if they exceed yield stress, they will undoubtedly lead to failure (Moss, 1987). They are caused by mechanical loads and needed for fulfilling the laws of equilibrium of moment, forces and pressures. Primary general stresses are characterized by not being self-limiting, which means that they are distributed within the structure so that no stress redistribution will occur as a result of yielding. An example of a primary stress is the hoop stress within a pressure vessel induced by an inner pressure. By increasing the pressure the stresses will increase until yielding. There are three sub-categories of primary stresses:

General primary membrane stresses (P_m) are self explanatory; membrane stresses with the characteristics of primary general stresses. These are found remote from large discontinuities like cone-cylinder intersections and nozzles. Examples of general primary membrane stresses are membrane stresses at the center of a flat head or the circumferential and longitudinal stresses within a pressure vessel (Moss, 1987).

Primary bending stresses (P_b) are those bending stresses that fulfill the requirements of primary general stresses. One example is bending stresses in the center of a flat head of a cylindrical pressure vessel.

Primary local membrane stresses (P_L) are a type of membrane stresses that are present near structural discontinuities. They originate from mechanical loads and though they have some characteristics of secondary stresses (which will be described later), to be conservative they must be considered as primary stresses. S is defined as the material specific maximum allowable stress intensity, R is the radius of the shell and t is the shell thickness. The membrane stress can then be considered as local if the region with stresses exceeding $1.1S$ does not extend more than \sqrt{Rt} in the meridional direction (ASME, 2010; European committee for standardization, 2009).

Secondary stresses (Q) are characterized by being self-limiting, which allow them to exceed yield since small plastic deformations will redistribute the stresses to other parts of the structure. Common examples of this stress type are general thermal stress and bending stress at a gross structural discontinuity, for example a nozzle (ASME, 2010).

As described previously, the computed stresses should be compared to material specific stress limits, S in ASME and f in EN but hereafter denoted as S . The stresses that need to be considered are P_m , P_L , P_L+P_b , $P+Q$. Note that P_m , P_L , P_b and Q are tensors consisting of six different components. Therefore when adding categories (for example P_L+P_B) this is done on component level, resulting in a sum of six different components. To get one scalar that represents each stress class and is comparable to the stated limits, the equivalent stresses are calculated using von Mises stress criteria based on the three principal stresses, as can be seen in equation (1).

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

Finally the computed equivalent stresses (S_e for P_m , P_L , P_b+Q) can be compared to their corresponding allowable value according to equation (2-5), where S is a code specific value based on temperature and material properties.

$$P_m \leq S \quad (2)$$

$$P_L \leq 1.5S \quad (3)$$

$$P_L + P_b \leq 1.5S \quad (4)$$

$$P + Q \leq 3S \quad (5)$$

If any of these inequalities is not fulfilled, the design code is not met, which means that the design needs to be revised and a new stress categorization procedure must be carried out.

3.2.3 FE analysis definition

As mentioned in section 1.1.2, there is a new design that should be evaluated according to the codes and there is limited documentation available for this particular case. A more extensive documentation is available for the present design where the nozzle intersects the cylindrical part of the shell. Therefore appropriate concepts used for the present design have been combined and applied to the new design. Extensive guidance has been obtained from Porter and Martens previous work, where different aspects of pressure vessel design of a nozzle intersecting a shell are discussed (Porter et al., 2008; 1999; Porter & Martens, 1996; 1998).

Elements

There are several different categories of elements available for analyzing pressure vessels. In order to get meaningful and useful results the right category of elements must be chosen. Usually the choice is between 3D or shell/plate elements based on a compromise between accuracy and costs in terms of computational capacity and time. In the particular cases that are analyzed during this project the ratios r/t are relatively high, where r is the radius and t is the thickness of the vessel part. Several studies suggest that if this ratio is greater than 10 it is adequate to use shell elements instead of 3D elements since it is not proven that 3D elements will produce more accurate results (Porter, Martens, & Caldwell, 2008; Porter, Martens, & Marcal, 1999; Porter & Martens, 1998).

When using 3D elements, a linearization procedure is needed in highly stressed areas of interest, in order to divide the stresses into membrane and bending stresses for the categorization. All elements along a Stress Classification Line (SCL) running through the material thickness must therefore be analyzed and by compiling all stresses the stress distribution can be determined. The process of defining SCL's can be rather time consuming since they have to be defined manually in critical areas.

Shell elements have the advantage that they will report membrane and bending stresses separately. The elements can also report the top- and bottom stresses where the membrane and bending stresses are summed up. This method provides a more convenient way of working for the analyzer and since it is recommended for thin walled applications it is used within this project. The nodes of a 3D element can transfer translating displacement in three orthogonal directions to the next element. These three directions in each node are enough for representing the behavior of the structure, and based on the displacements the stresses can be calculated according to the material model applied. Within this project linear material models are assumed where the stress is directly proportional to the strain (Hooke's Law), which can be deduced from the displacement. Since shell elements have no thickness, three directions of displacement is not enough for transferring bending stresses between elements. Therefore three additional displacement directions need to be introduced that represent the rotations around three orthogonal axes. Even though each shell element node needs twice as many degrees of freedom as 3D elements, the global effect will be that the total number of degrees of freedom will be reduced. This is because the number of elements over a cross section is reduced to one. The shell element theory is based on the assumption that stress and displacement in the direction of the thickness of each element can be neglected. This applies to the case within this project where the major loads are inner pressure and the nozzle loads.

Due to limitations of the software, triangular elements need to be used. There are two types of triangular elements available; first order elements with three nodes and second order elements with six nodes as presented in Figure 10. The main advantage in using first order elements is that the type is less intensive in terms of computational resources. Yet, it usually leads to an overestimation the stiffness of the structure and should therefore only be used for early estimations (Sunnersjö, 1999). The second order element is therefore used, since it generates more accurate results including a more realistic stiffness. It has also the ability to better represent the geometry since the surface can be curved.

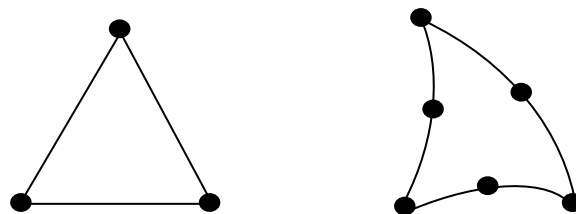


Figure 10 First and second order triangular element with three and six nodes respectively.

Boundary conditions and loads

Poorly defined boundary conditions and load application are two of the most critical contributors to errors within FE analysis (Porter, Martens, & Marcal, 1999). If the structure is insufficiently restrained the effects can be both under- and over estimations of the stresses. The application of restraints and loads must therefore be based on well motivated decisions, making sure that the model represents the real situation as well as possible. Loads are applied to the outer circumference of the nozzle, representing the behavior of the pipe that is connected. The origin of the loads is for example from axial thermal expansion of the pipes which leads to forces and moments that act on the nozzle. The magnitudes of the loads are derived within each project using a pipe analysis software. In addition to the external loads that are calculated by a dedicated software, an extra radial directed force ($F_{x_external}$) is present. The radial load is due to the inner pressure (p) of the vessel that causes a force on the adjacent piping system, usually resulting in a significant force. The total force F_x is calculated according to equation (6), where A_{Nozzle} is the inner cross section area of the nozzle.

$$F_x = F_{x_external} + p \cdot A_{Nozzle} \quad (6)$$

It is recommended that the distance between a discontinuity that induces disturbances in the stress field, like the junction between shell and nozzle, and an applied boundary condition should be at least $7.5\sqrt{rt}$ (Porter, Martens, & Marcal, 1999). This criterion is truly fulfilled for all restraints and loads that are applied. To further reduce the effects that the applications of loads induce, the nozzle has been extended to represent a part of the adjacent pipe and its stiffness.

The only restraint that fixes the vessel is located at the bottom of the skirt where the vessel is mounted onto a framework of steel or stands on a concrete foundation. Since the interesting areas of stresses are in the junction between the shell and the nozzle, and in the transition between the cylindrical and conical part of the vessel, the nodes along this line can be assumed to be fixed (in three translational and three rotational directions) without affecting the result significantly.

Meshing

Another significant factor that will affect the possibilities to obtain acceptable results from the analysis, is how the mesh is defined (Porter, Martens, & Marcal, 1999). A finer mesh will generate more accurate results, at the price to pay of longer calculation time. However, the magnitude of the stress in an arbitrary element will converge at some point and when an increased mesh density results in an increase in stress by less than 5%, the result can be considered as having a satisfying level of accuracy (Porter, Martens, & Marcal, 1999). Since this project focuses on the effects of moving the nozzle to intersect the cone, the distribution within the area around the nozzle is the most interesting. Therefore, the mesh density will be considerably higher within this region.

Validation

In order to validate the calculation model created within this project, the results of the analysis of the configuration where the nozzle intersects the cylindrical part of the shell is compared to the results presented by Porter, Martens and Caldwell (2008), where a similar structure is evaluated. By making sure that the results correspond reasonable good, there is a higher probability that the analysis of the configuration where the nozzle intersects both the cylindrical and conical parts of the shell is reasonably accurate. To further prove the accuracy of the model, some simple checks

are made. One example is to evaluate the primary membrane stress away from a discontinuity using FEA. If the analysis is correctly set up this stress should be equal to the value calculated analytically using the formulas for hoop stresses created by an internal pressure on a thin walled pressure vessel (Lundh, 2000).

$$\sigma_1 = \sigma_\varphi = \frac{p \cdot r_{cylinder}}{t_{cylinder}} \quad (7)$$

$$\sigma_2 = \sigma_z = \frac{p \cdot r_{cylinder}}{2 \cdot t_{cylinder}} \quad (8)$$

$$\sigma_e = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1 \sigma_2} \quad (9)$$

In equations (7-8) σ_φ is the membrane stress in the circumferential direction and σ_z is the stress in the axial direction. Combining these by the use of von Mises yield criterion for plane stress equation (9) will give the comparable equivalent stress.

The method is also validated by examining the reaction forces reported by the software and comparing them with the external loads that are applied. Even though FE is an advanced calculation tool, the conditions of equilibrium must still be satisfied and the supporting forces must, since FE is a simplification of the real structure, be at least approximately equal to the ones applied.

4 The parameterized calculation model

As presented in section 3, the calculation process involves several steps within different software modules. This section describes the analysis procedure and the underlying functions that enable it. The process starts and ends within the same Excel document but in different worksheets, one for configuring the analysis and one for presenting the results. The document also consists of several supporting work sheets. These are used for calculation of for example forces and mesh parameters, interface sheets for communication with other software within the tool, sheets with material properties, etc. In addition, a help sheet is created where the user can get more information about the parameters that should be defined.

4.1 Definition of parameters

All significant project specific properties are stated in an Excel sheet that is divided into several modules. The first and the last part can be seen in Figure 11 and Figure 12 respectively and the part in between has the same layout, but with other parameters. Naturally the basic dimension needs to be stated and also the material for different parts of the pressure vessel. To make the input procedure logical the sheet is divided into modules that each represents one specific part of the geometry. In Figure 13 the different sections are represented and named.

The image shows a screenshot of a configuration sheet divided into two main sections: GENERAL and SHELL. The GENERAL section includes a 'Configuration' link and two radio button options: 'Nozzle over cone' (selected) and 'Nozzle on shell'. The SHELL section contains several parameter groups, each with a link to the parameter name, a text input field, and a unit. The 'Upper channel' group has a 'Thickness' of 9 mm. The 'Heater shell' group has a 'Diameter (inner)' of 6082 mm and a 'Thickness' of 9 mm. The 'Inlet plenum' group has a 'Diameter (inner)' of 7180 mm. The 'Inlet shell' group has a 'Height' of 5700 mm, a 'Thickness' of 10 mm, and a 'Material' dropdown set to 'SA-240 Gr.304'. The 'Inlet shell strip' group has a 'Height' of 200 mm, a 'Thickness' of 10 mm, and a 'Material' dropdown set to 'SA-240 Gr.304'. The 'Lower channel cone' group has a 'Height' of 1040 mm, a 'Thickness' of 16 mm, and a 'Material' dropdown set to 'SA-240 Gr.304'.

Section	Parameter	Value	Unit	Material
GENERAL	Configuration			
GENERAL	Nozzle over cone	<input checked="" type="radio"/>		
GENERAL	Nozzle on shell	<input type="radio"/>		
SHELL	Upper channel Thickness	9	mm	
	Heater shell Diameter (inner)	6082	mm	
	Heater shell Thickness	9	mm	
	Inlet plenum Diameter (inner)	7180	mm	
	Inlet shell Height	5700	mm	SA-240 Gr.304
	Inlet shell Thickness	10	mm	
	Inlet shell strip Height	200	mm	SA-240 Gr.304
	Inlet shell strip Thickness	10	mm	
	Lower channel cone Height	1040	mm	SA-240 Gr.304
	Lower channel cone Thickness	16	mm	

Figure 11 The first part of the configuration sheet where all parameters are stated.

PRESSURE, LOADS & TEMPERATURE			
Inner pressure	<input type="text" value="0.05"/> MPa	Temperature Shell side	<input type="text" value="120"/> °Celsius
Fx	<input type="text" value="-1700"/> N	Temperature Tube side	<input type="text" value="100"/> °Celsius
Total Fx	463004 N		
Fy	<input type="text" value="12000"/> N		
Fz	<input type="text" value="3500"/> N		
Mx	<input type="text" value="1150"/> Nm		
My	<input type="text" value="1500"/> Nm		
Mz	<input type="text" value="10000"/> Nm		

INFORMATION		Forward
<div style="border: 1px solid black; padding: 5px;">No Warnings</div>		
<input type="button" value="Create Simulation Input"/>	<input type="button" value="Run"/>	<input type="button" value="Back"/> <input type="button" value="Forward"/>

Figure 12 The last part of the configuration sheet where all parameters are stated.

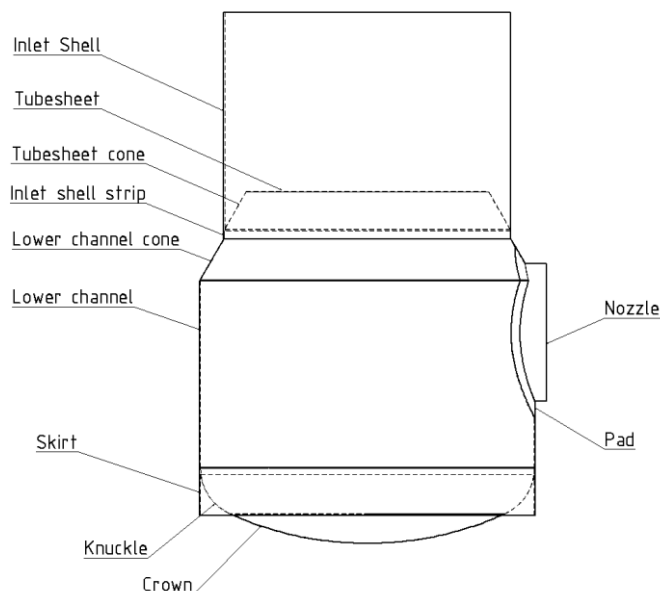


Figure 13 Parts that are included in the calculation tool.

As a start, the user needs to specify what configuration that should be analyzed; if the nozzle intersects only the cylindrical part of the shell or if it intersects both the cylindrical and the conical part of the shell. Since different features are used for geometry modeling depending on how the nozzle is positioned, the right configuration needs to be activated. This information also serves as input for evaluating if the specified dimensions are compatible with each other and if they represent the chosen configuration properly.

In the next module the parameters of the shell should be stated, including diameters, heights, thicknesses and materials. All dimensions that should be stated are presented in Appendix A. The same applies for the subsequent modules where parameters defining the pad and nozzle should be stated. Later on, the nozzle forces, pressure and temperatures should also be stated. When defining the position and diameter of the nozzle, the user will get an instant indication of whether the nozzle is correctly

positioned according to the chosen configuration or not. All names of the parameters in the sheet are linked to a help sheet where descriptions and figures are presented in order to further explain what information is requested.

In the configuration sheet the material of all parts should be defined. Today, mainly four material types are used for evaporators at the company and therefore the number of available materials within the tool is limited to these. The choice of material together with the temperature serves several purposes. First, since some material parameters are temperature dependent and since the operating temperature of the vessel usually varies between 100 and 200 degrees Celsius, it is important that the right parameters are used. One example of this is Young's modulus that is reduced as the temperature increases. If the value of Young's modulus is taken at 25°C instead of at the operating temperature, the result will be non-conservative and overestimates the strength of the structure. Secondly, the combination of material and temperature determines the maximum allowable equivalent stress. Therefore a material table is included where the material properties at different temperatures can be read and used as input for both the FE analysis and the stress evaluation.

The external loads and inner pressure are also defined in the configuration sheet. The forces and moments that act on the nozzle end are defined together with the inner pressure. Note that the inner design pressure sometimes can be negative. In such a case the risk for failure due to buckling is most likely to occur, but according to the codes the design must still fulfill the requirements of gross plastic deformation (evaluation using stress categorization).

4.2 Parameter export

A major challenge in the project was how to export all the information that was needed as input for the analysis from Excel to the simulation software. The final solution consists of two parallel information flows, where one contains information for the geometry generation and one contains information for the analysis.

4.2.1 Geometry parameters

By the use of Automate Works the parameters defined in Excel are linked to the geometry generation in SolidWorks. The surfaces of the geometry are defined by a 3D sketch that works as a skeleton and can be seen in Figure 14. All dimensions in the 3D sketch are linked to, and controlled by the Excel sheet by using Automate Works.

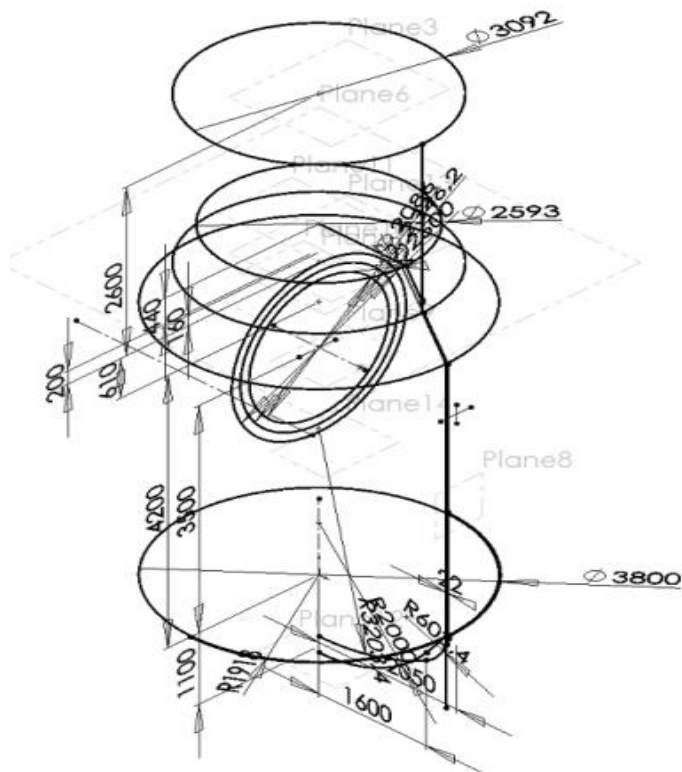


Figure 14 Principle structure of a 3D sketch.

Automate Work reads the rows from a certain work sheet within the file that is formatted so that each row describes an action that should be executed in SolidWorks. It can be a dimension that shall be updated, a part that shall be opened or a certain keystroke combination. When classifying the stresses the categories are partially based on the location of the stresses. To facilitate the evaluation of certain stresses, some areas are therefore defined already at this stage. One example is the area for primary local membrane stress (P_L) that is defined at the junction between nozzle and shell, see Figure 15.

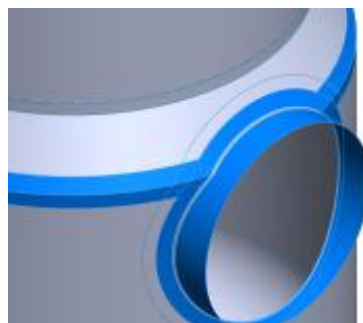


Figure 15 Identification of areas for evaluating Primary local membrane stresses.

In order to reduce the need for computational capacity the model consists of surfaces controlled only by the 3D sketch. If it was desirable to use the model for other purposes within the product development process a solid model could have been established. Yet, this implies that the model gets more complex and a more extensive pre-processing work is needed to get an appropriate mesh. Additionally, if solid elements would have been used the computational time would have increased

drastically. As an alternative, the solid features could be used for creating midsurfaces, implying that shell elements could be used for meshing. However, this would, as described above, probably result in a more advanced pre-process when idealizing the geometry. It is therefore assumed that using a surface model is an appropriate way of modeling the geometry for the purpose of this project.

The parts of the pressure vessel are in general welded together. When shell elements are used instead of 3D elements, problems with modeling the welds need to be solved. There are several different methods available for representing welds by the use of shell elements, all with benefits and limitations. The risk of failure due to fatigue is highest in the welds since peak stresses are present in these areas. Since fatigue is not within the scope of this project, the assumption is made that the welds can be omitted (Porter, Martens, & Caldwell, 2008).

4.2.2 Analysis parameters

Since Automate Works only support parameters that are exported to the SolidWorks part and assembly modeling modes, it does not support export to the solid mechanics analysis mode. Therefore an alternative is needed for parameter export, and by the use of a function within the analysis mode called *Design Scenario*, parameters can be imported in *.csv format. The acronym csv stands for *comma-separated values* and can be created using Excel. A *.csv file consists of rows of data where the data on each row is separated by commas. By creating a new *.csv file from the parameters sheet described, an export-file from Excel is created which can be imported in the analysis software. Within the analysis mode of SolidWorks, variable parameters are defined, representing for example material properties, mesh parameters, dimensions, forces, moments/torques, etc. Their value can be changed by the *.csv file, which can be partially seen in Figure 16. The *.csv file is updated by running a macro within the configuration sheet prior to running the analysis.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q
1	Design Set, 1, 53																
2	Parameters, Status, t_LowerChannel, t_PadLowerChannel, t_InletShell, t_Skirt, t_TubeSheetCone, t_TubeSheet, t_Gable, t_PadLowerChannelCone, t_LowerChannelCone, t_Nozzle, t...																
3	Units, , mm, mm, mm, mm, mm, mm, mm, mm, mm, mm, mm, N/mm^2 (MPa), N, N, N, N-m, N-m, N-m, Celsius, Celsius, mm, mm, mm, mm, mm, mm, N/mm^2 (MPa), N/mm^2 (MPa)																
4	Set1, 1, 14, 32, 10, 25, 25, 35, 16.9, 34, 16, 14, 10, 0.05, 447704, 1297, 378, 1150, 1500, 10000, 120, 120, 751, 181, 10, 526, 10, 185890, 185890, 185890, 185890, 185890, 185890																
5																	
6																	
7																	
8																	
9																	
10																	
11																	

Figure 16 Export file for analysis parameters from Excel to the FE software.

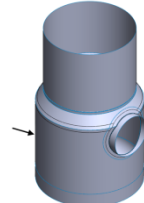

4.3 Performing the analysis

When all parameters are set and the *.csv file is created the analysis can be performed. When clicking the “Run”-button in the configuration sheet, Automate Works starts to run through the list of activities starting by opening the right configuration and then running through the subsequent rows where all basic dimensions are updated. When the geometry is updated a keystroke combination imports the FE analysis parameters and initiates the analysis. The new mesh is then created and the solver calculates the displacements from which the stresses can be deduced.

4.4 Stress evaluation

When the analysis is completed three different types of plots can be chosen; membrane, top and bottom stresses. Manual selection is now needed to choose what areas should be evaluated for a specific stress type. For the case where the nozzle intersects both the cylindrical and conical part of the shell the different stress types and their locations are listed in Table 1.

*Table 1 Areas for stress evaluation for different stress classes. *Note that no stresses should be evaluated in the junction ring.*

Stress category	Plot	Area	
P_m	Membrane	Areas unaffected by discontinuity effects from nozzle	
P_L	Membrane	Area on the pad close to the nozzle* and in the transition area of the cylindrical and conical	
$P+Q$	Top	All areas*	
$P+Q$	Bottom	All areas*	

For the primary membrane stress only one value is needed, chosen from the side of the shell that is opposite to the nozzle. The codes state that for the design configurations that are evaluated within this project, all bending stresses in the highly stressed areas of interest can be considered as secondary (Q). Therefore evaluation against the P_L+P_B criteria will not be necessary. All element stresses for P_L and $P+Q$ in the areas of interest in the respective plot are then exported to *.csv files to be used for stress evaluation in Excel. Since fatigue is out of scope for this project, nonlinear peak stresses over the material thickness are not taken into account. These typically occur in the junction between nozzle and shell where the weld is situated, and therefore this area will not be considered while evaluating the stresses. The extension of this region is dependent on the thickness of the nozzle and shell and is calculated according to Figure 17 and equations (10-11) (Porter, Martens, & Caldwell, 2008).

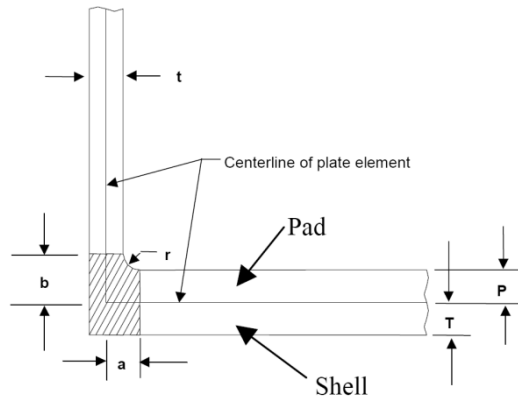


Figure 17 Definition of juncture ring (Porter, Martens, & Caldwell, 2008).

$$a = \frac{t}{2} + t \quad (10)$$

$$b = P + t \quad (11)$$

Within the Excel document where the parameters are stated there is another worksheet for evaluation of the stresses. By importing the *.csv files with the results from the analysis, the maximum element values can be determined for the P_L and $P+Q$ stresses. The value for P_m needs to be typed manually. In order to verify the model the calculated value of P_m can be compared to its analytic solution using equations (7-9). The evaluation sheet with the imported stresses and maximum allowable equivalent stresses can be seen in Figure 18.

STRESS EVALUATION					
		Allowable stress (S_m)		134,6 MPa	
Reported stresses		Factor (f)	Max allowable intensity ($S_m \cdot f$)	Analytical solution	
Refresh P_L	General Primary Stress (PM)	12,8 MPa	1	134,6 MPa	13,0
	Local Primary Stresses (PL)	52,7 MPa	1,5	201,9 MPa	
	Element no	15043			
Refresh (P+Q)	Primary + Secondary Stresses (P+Q)	88 MPa	3	345 MPa	
	Element no / stress type	15470 / Top			

Figure 18 Evaluation sheet where the maximum for each stress category is represented

5 Stress calculations

Within this section examples of geometries that are analyzed will be presented. They are evaluated using the methodology previously described starting with a model which represents the design of today's product. In the subsequent section the new design is analyzed by the calculation tool and the results of both the analyses are compared to each other. As described previously, a study has been made by Porter, Martens and Caldwell (2008) on the configuration where the nozzle intersects the cylindrical part of the shell. By using the same input parameters and comparing the results, the validity of the model has been examined.

5.1 Comparison of designs

To study the behavior of the new design and determine if it is suitable for future products it is compared with the present one. The design used today is well established and if the stress analysis results of the new design are within the same range as the present, it would probably be interesting to investigate the new design further.

To get an impartial comparison between the models the same input data has been used with a few exceptions. These exceptions reflect the reduced height of the vessel when the nozzle is repositioned upward. The height of the nozzle from the reference level is therefore kept constant and the height of the lower shell is reduced. The dimensions of the model can be seen in Appendix A. The design temperature was set to 120°C.

The forces and moments are defined as in Figure 19. The forces stated as nominal in Table 2 are the ones calculated from the pipe analysis software. Reaction forces are also presented in the table. Since the loads can be applied using either a rigid or flexible end of the nozzle and none of them properly reflects the real situation, an alternative solution is needed. By extending the nozzle and applying the loads to a rigid end, the nozzle will behave and deflect more like the real nozzle and its connected pipe. The forces that act on the nozzle end cause moments at the junction between nozzle and shell, which increase if the forces are kept constant and the length of the nozzle is increased. To avoid this, the magnitudes of the forces are reduced linearly as the nozzle length is increased. However, it should be mentioned that usually only F_x , M_y and M_z are considered when calculating the nozzle forces and in that case, the length of the nozzle is irrelevant.

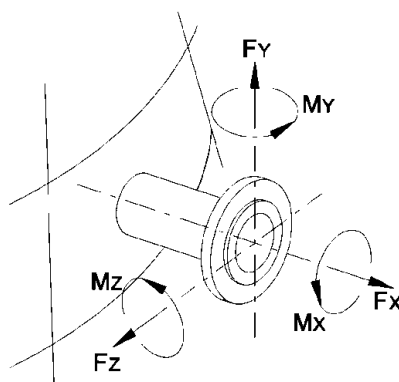


Figure 19 Direction of nozzle loads

Table 2 Applied external loads and pressure. For the forces also the reaction forces are presented.

Type	Quantity	Nominal load	Applied load	Reaction load	
				Case 1	Case 2
Force	F_x	-17000 N	-17000 N	17008	16975
Force	F_y	12000 N	1297 N	-855	-861
Force	F_z	3500 N	378 N	-374	-385
Moment	M_x	1150 Nm	1150 Nm	*	*
Moment	M_y	1500 Nm	1500 Nm	*	*
Moment	M_z	10000 Nm	10000 Nm	*	*
Pressure	p	0.05 MPa	0.05 MPa	*	*

5.1.1 Case 1 - Existing design

The analysis of the existing design where the nozzle intersects only the cylindrical part of the shell shows, as expected, that all the evaluated equivalent stresses are within the allowable limits. A summary where the maximum stresses in each category are presented can be seen in Table 3. If looking at the stress plot in Figure 20 and Figure 21 it can be seen that the highest stresses are found on the top and the bottom of the nozzle junction for the local membrane and the primary plus secondary stresses respectively. It should be noted that stresses within the same order of magnitude are found on the opposite side of the nozzle in both the case of local membrane stresses and primary plus secondary stress.

Table 3 Comparison of the calculated maximum stresses and the maximum allowable stresses.

Stress category	Maximum allowable stress [MPa]	Maximum calculated stress [MPa]	
		Case 1	Case 2
Primary General Membrane (P_m)	$S_m = 134.6$	12.7	12.7
Primary Local Membrane (P_L)	$1.5S_m = 201.9$	54.1	52.4
Primary + Secondary ($P+Q$)	$3S_m = 403.8$	88.3	89.7

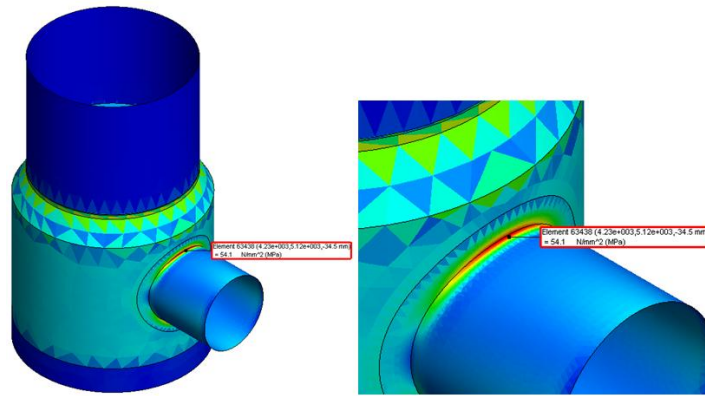


Figure 20 Maximum identified Primary local membrane stress (P_L) for Case 1.

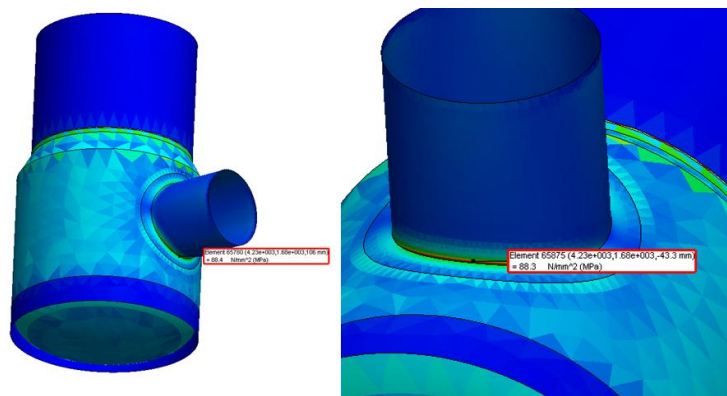


Figure 21 Maximum identified Primary plus secondary stress ($P+Q$) for case 1.

5.1.2 Case 2 - New design

When evaluating the new design using the same loads as for the existing one, some differences can be seen. The summarized result is presented in Table 3 and the stress plots can be seen in Figure 22 and Figure 23. It can be seen that the stresses are within the same range as in the first case. The analysis shows that the stresses in the region where the cylindrical part meets the conical part of the shell are well below the maximum calculated stresses of 52 MPa and 88 MPa for P_m and $P+Q$ respectively, which are located under the nozzle.

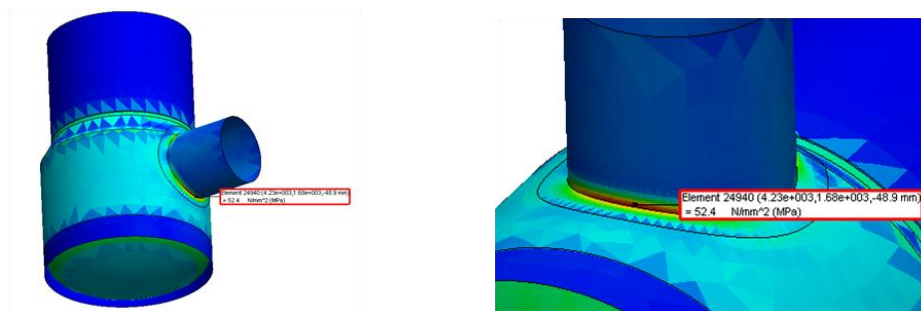


Figure 22 Maximum identified Primary local membrane stress (P_L) for Case 2.

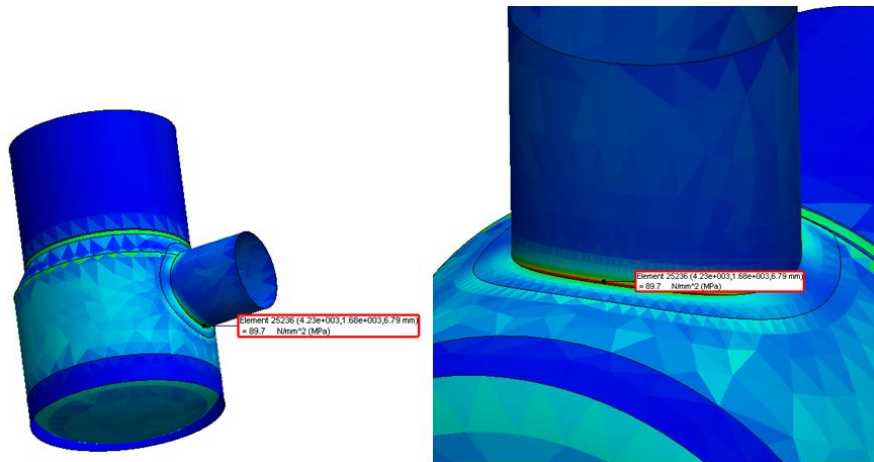


Figure 23 Maximum identified Primary plus secondary stress ($P+Q$) for case 2.

5.2 Stress result verification

Since there are many parameters that affect the result of an FE analysis it is important to verify the results in order to give legitimacy to the model. As described in section 3.2.3 there is limited documentation available for the new geometry and therefore it is hard to evaluate the design according to previous work. However, research has been done by Porter, Martens and Caldwell (2008) for the case where the nozzle intersects the cylindrical part of the shell. These results are compared to the results from the analysis, focusing on the highest stresses that occur in the junction between shell and nozzle. Based on the largest element stresses in each category, the stresses along a radial line on the shell passing through the centre axis of the nozzle are evaluated. The adjacent stresses on the nozzle are also evaluated following the same line but in the axial direction of the nozzle. The position of the line can be seen in Figure 24.

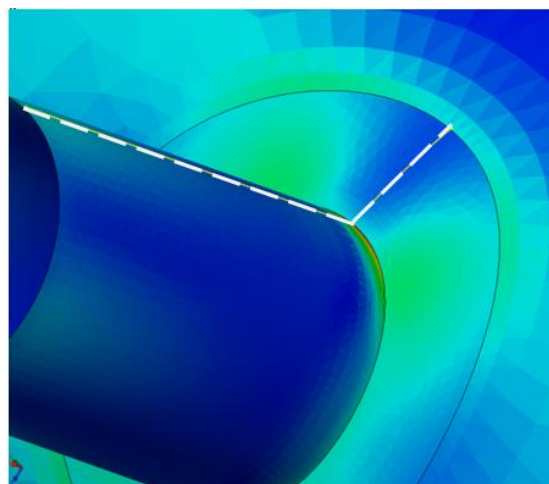


Figure 24 Evaluation line when verifying the results according to (Porter, Martens, & Caldwell, 2008).

Since three types of stresses (Membrane, Top and Bottom) are presented by Porter, Martens and Caldwell (2008), all these are compared with the ones obtained from the analysis. The stresses are compared in Figure 25-Figure 27.

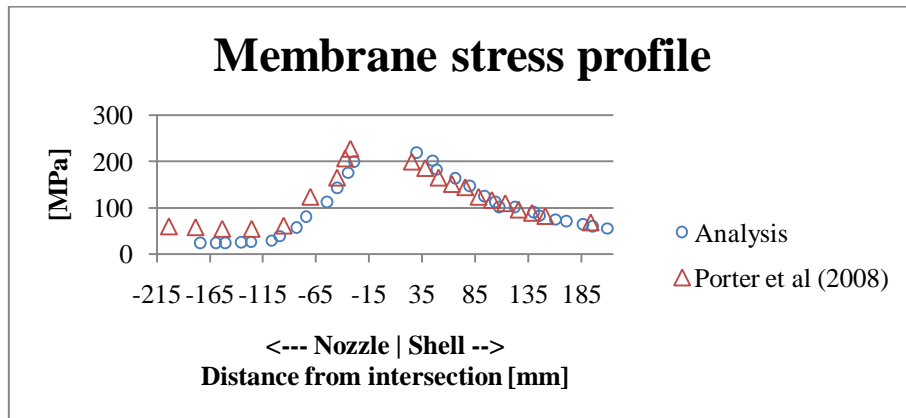


Figure 25 Membrane stress (P_m) in junction between nozzle and shell.

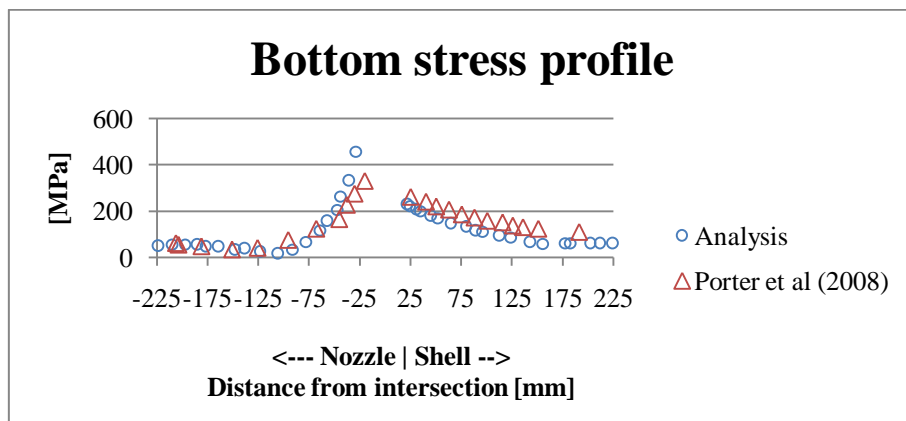


Figure 26 Primary + secondary ($P+Q$) stress in bottom layer in junction between nozzle and shell.

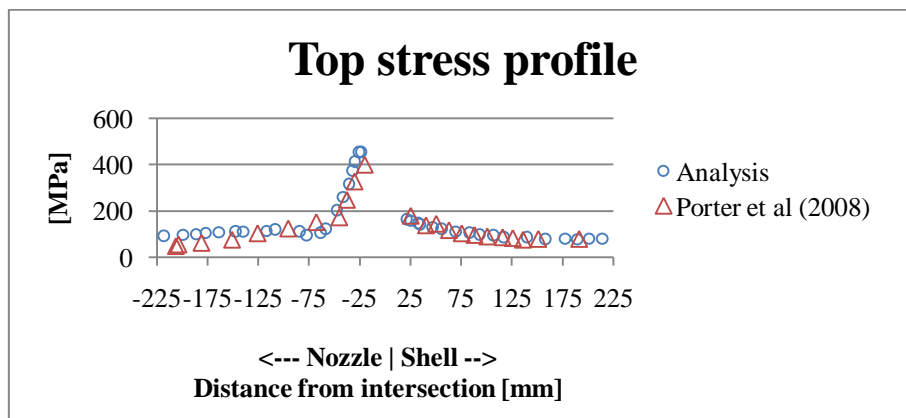


Figure 27 Primary + secondary ($P+Q$) stress in top layer in junction between nozzle and shell.

It is interesting to see that the results correspond well for all three stress types. Though, some differences can be seen which will be further discussed in later sections of the report.

The reaction forces at the bottom of the skirt are approximately zero in the directions where no external loads are applied. In the axial direction of the nozzle the reaction force is however nonzero and instead equal to the difference between the external load and the force created by the inner pressure, which is expected. This indicates that the model is properly defined.

The expected value of the membrane stress in regions unaffected by the nozzle is calculated according to equations (7-9) and is determined to be 108 MPa. By examining a few elements on the side of the shell opposite to the nozzle, the membrane stress of the analysis is determined to be 105 MPa. The difference of 3 MPa is considered to be a reasonable error, which indicates that the model is properly defined.

6 Discussion and Conclusions

A calculation tool for evaluation of pressure vessel shells with large nozzles has been developed during this project. It consists of an Excel sheet where the input parameters are stated, which is connected to a parameterized CAD geometry that is used for FE analysis. By exporting different types of stresses from different areas the design can be evaluated by using stress categorization and the results are reported in yet another Excel sheet to assess its conformity with the requirements of ASME and EN codes.

Analyzing pressure vessels according to codes puts high requirements on the analysis method. Not uncommonly, the mechanical design of a pressure vessel is verified by a third party inspection organization to make sure that it fulfills the requirement of the code. It is therefore of vital importance that the model is set up properly in order to get acceptable results. Since the intention when creating this calculation model was that it should be used within several projects, it is even more important that the model is created with accuracy based on thorough investigations.

However, the ASME and EN codes do not provide clear guidelines for FE analysis, and extensive knowledge within the field is usually required to properly divide the stresses into categories (Seshadri, 2009). It is also advisable that when the model implies complex geometry and load cases, the alternative analysis methods are more suitable and it is therefore recommended to use the limit load or elastic plastic methods that are mentioned in section 3.2.2 (ASME, 2010). With this background it is a challenging task to create a model that automatically generates the geometry and evaluates it according to the codes. The codes can occasionally be perceived as ambiguous and the result of a stress categorization therefore often depends on the analyst who interprets it.

6.1 Stress evaluation

Based on the reasoning above the results that the model produces are reasonably accurate. An evidence of this is the comparison that has been made with previous work by Porter, Martens, & Caldwell (2008). The stresses close to the junction between shell and nozzle correspond well but the maximum stresses differ in some of the cases. This could be due to the high stress gradients that occur in this region. For the membrane and top stresses, deviations can be seen in the tail of the graphs in the nozzle direction. The authors use a stiff nozzle end to which the nozzle loads are applied. The effects of this stiff end will truly affect the stresses on the nozzle but since the length of the nozzle is not specified by the authors, it is uncertain whether these effects reach the elements where the stresses are measured. If the effects reach these elements it could be one explanation to the differences.

The results of the analysis of case 2 in section 5.1.2 have also been verified by a third party consultant with extensive experience and deep knowledge within pressure vessel design and the evaluation according to the codes. This is definitely a strong argument that indicates the validity of the model.

When comparing the new and the present design of the pressure vessel only small differences in stress were obtained. This was rather expected since the pressure is the main load and it is not uncommon that applied loads and moments are of greater magnitude than those used in the presented examples. An interesting observation is that the maximum P_L is located on the top of the nozzle in case 1 and underneath the nozzle in case 2. Based on this observation it seems that the cone contributes with stiffness and improves the stress situation on the top of the nozzle.

Within the method presented by Porter, Martens, & Caldwell (2008) the stresses in the absolute vicinity of the shell and the nozzle are not considered. The strength of the method is that by using simple geometrical rules the stress concentration region in the junction can easily be omitted. This can be desirable since welds are usually difficult to model, especially when shell elements are used, and can result in inappropriate stress values. On the other hand, by not considering the stresses in the junction the purpose of the stress categorization method can be said to be violated. The intention is to design the pressure vessel to protect it from plastic collapse, see section 1.3. This means that the primary stresses must be below $1.5S$, which usually equals the yield strength. If the stresses are higher, plastic deformation will occur and since the stresses are not self limiting the vessel will eventually collapse. If these stresses are omitted from the evaluation, there is a risk of large plastic deformation in the area near the junction. This is one example of the weakness of the stress categorization method since it not clearly describes how to evaluate the stresses in the junction between the nozzle and the shell. The weak guidelines give opportunities for the analyst to interpret the code subjectively which can lead to different results depending on who is analyzing the geometry.

It should be emphasized that when evaluating according to codes there is still a need for engineering judgment which should not be underestimated. The calculation tool can be helpful for decision making, but in addition to the results presented by the tool the user also needs to ask if the results are trustworthy and do not trust them blindfolded. As a help basic formulas can be used, and are also used within this project to verify the obtained results of P_m .

6.2 Automation of model

Within the scope of this project the task was also to automate the process. Initially the possibilities to exchange information between the software within the tool were rather unexplored. Evaluation of different modeling techniques, analysis setup and import and export functions implied an iterative work process ending up with a semi-automated model. A fully automated tool would of course be desirable for the user, and could probably be achieved by the use of *Application Programming Interface* (API). This functionality within SolidWorks can manage different functions within the software, such as creating geometries and running analysis. However, this function requires a substantial amount of programming knowledge and implementing such functionalities are therefore considered to be out of scope of the project. The need for some manual steps during the calculation process could on the other hand provide the user with some insight into and understanding of the methodology. There is a risk that the calculation tool in some rare cases reports erroneous results. If the analyst has to do some manual work in that case, these errors can be identified and corrected.

More details could be included in the model in order to more accurately represent the real case. Yet, this would imply a more complex model with the risk of being more unstable. In that case the user also has to state more input variable which results in reduced user friendliness if too many variables need to be defined. The degree of automation is hence a balance between user friendliness and accuracy.

There are already software for similar purposes available on the market today. These are usually developed by research groups working fulltime on the subject. The model developed during this project is not fully automated but provides reasonable results suitable for the intended purpose. In addition, the implementation of this calculation model into the development process will definitely reduce the time spent on calculations, compared to the case where the analyst needs to set up an entire new model for each project. Knowledge of how to analyze and evaluate is saved within the model and as stated by Stokes (2001) benefits as savings in money and time can be achieved. Additionally, it also enables the analyst to focus on more creative design tasks.

7 Recommendations

By the use of an automated calculation tool several benefits can be achieved in the product development process. However, there is potential for some improvements of the model and a few recommendations on how to utilize it, which will be presented below.

It is advisable that the robustness of the model is further improved. In its present state the model is rather sensitive to improperly defined input variables and can be affected by other software. Some inexplicable errors have also been seen while importing parameters and meshing the model. These can be further investigated to find whether they are caused by software bugs or if they can be resolved in an easier way.

To fully implement this calculation tool into the development process it should be integrated with the PDM (Product Data Management) system. In this way the documentation around the tool will be more structured, the risk for conflicts between different revisions will be minimized and it will facilitate the availability for all potential users.

The model is today only semi-automated and could probably be fully automated by the use of API. In that case, almost all functions within the analysis software can be automated, including the selection of appropriate areas of stresses which could be an important improvement. It might also be possible to overcome the limitations in robustness by using API to control the functions. Yet, if it is crucial that the model is fully automated, a reasonable amount of programming knowledge is required of the implementer.

To further develop the model in a more long term perspective it could be interesting to investigate the possibilities to implement a module that automatically proposes design improvements to further optimize the product. As presented in section 2.1, such projects have been run and the field is expected to have great potential in the future.

Since another automated tool is already used for calculating the basic dimensions of the evaporators, the possibilities to integrate the systems should be evaluated. Such integration is probably a project that could not be implemented in the nearest future, but eventually, when the analysts are feeling comfortable with the calculation tool, it could be of interest.

According to the literature, the guidelines of how to integrate FEA with the code will not be further developed (Baylac & Koplewicz, 2004). This means that the risk of ambiguous results that depends on how the analyst interprets the code will be imminent. It is therefore recommended that the possibilities of using nonlinear methods for fulfilling the requirements of the codes are investigated. By using these methods more accurate results can be obtained and it has been shown that for a similar case, less conservative results can be obtained (Porter et al., 2001). Also ASME recommends that for design cases where the linear elastic method using stress categorization may produce ambiguous results, the limit load or elastic-plastic analysis methods should be used (ASME, 2010).

References

- ASME. (2010). ASME Boiler and Pressure Vessel Code, section VIII Division 2. *American Society of Mechanical Engineers*. New York: The American Society of Mechanical Engineers.
- Baylac, G., & Koplewicz, D. (2004). *EN 13445 “Unfired pressure vessels” Background to the rules in Part 3 Design. Design*. Union de Normalisation de la Mécanique.
- Boart, P., Andersson, P., & Elfström, P.-O. (2006). Knowledge enabled pre-processing for structural analysis. *NordPLM06* (pp. 89-97).
- Chapman, C. (2001). The application of a knowledge based engineering approach to the rapid design and analysis of an automotive structure. *Advances in Engineering Software*, 32(12), 903-912. doi: 10.1016/S0965-9978(01)00041-2.
- Chapman, C., & Pinfold, M. (1999). Design engineering—a need to rethink the solution using knowledge based engineering. *Knowledge-Based Systems*, 12(5-6), 257-267. doi: 10.1016/S0950-7051(99)00013-1.
- Chuse, R., & Carson, B. E. (1993). *Pressure vessels The ASME code simplified* (7th ed.). New York: Mc-Graw Hill, Inc.
- European committee for standardization. (2009). *EN 13445-3 Unfired pressure vessels - Part 3: Design. English*.
- Gujarathi, G. P., & Ma, Y.-S. (2011). Parametric CAD/CAE integration using a common data model. *Journal of Manufacturing Systems*. doi: 10.1016/j.jmsy.2011.01.002.
- Hollinger, G. (2000). Three-Dimensional Stress Criteria - Summary of the PVRC Project. *Journal of Pressure Vessel Technology*, 122(February), 105.
- Lu, M.-W., Chen, Y., & Li, J.-G. (2000). Two-Step Approach of Stress Classification and Primary Structure Method. *Journal of Pressure Vessel Technology*, 122(1), 2. doi: 10.1115/1.556139.
- Lundh, H. (2000). *Grundläggande hållfasthetslära* (3rd ed.). Södertälje: Kungliga Tekniska högskolan.
- Moss, D. (1987). Pressure vessel design manual. *Stress: The International Journal on the Biology of Stress*, 1-14.
- NOVAK, M., & DOLSAK, B. (2008). Intelligent FEA-based design improvement. *Engineering Applications of Artificial Intelligence*, 21(8), 1239-1254. doi: 10.1016/j.engappai.2008.03.013.

- Pastor, T., & Osage, D. (2007). Modernization of Pressure Vessel Design Codes ASME Section VIII, Division 2, 2007 Edition. *Transactions of the ASME-J- Journal of Pressure Vessel Technology*, 129(4), 754–758. New York, NY: American Society of Mechanical Engineers, c1974-.
- Pinfold, M. (2004). Using knowledge based engineering to automate the post-processing of FEA results. *International Journal of Computer Applications in Technology*, 21(3), 99. doi: 10.1504/IJCAT.2004.005935.
- Porter, M. A., Martens, D. H., & Caldwell, S. M. (2008). A Suggested Shell / Plate Finite Element Nozzle Model Evaluation. *Journal of Pressure Vessel Technology*, 130(August), 3-9. doi: 10.1115/1.2937740.
- Porter, M. A., Martens, D. H., & Marcal, P. (1999). On Using Finite Element Analysis for Pressure Vessel Design. *ASME-PUBLICATIONS-PVP*, 388, 191–196. ASME; 1998.
- Porter, M. A., & Martens, D. H. (1996). A Comparison of the Stress Results from Several Commercial Finite Element Codes with ASME Section VIII, Division 2 Requirements. *ASME-PUBLICATIONS-PVP*, 336, 341–346.
- Porter, M. A., & Martens, D. H. (1998). Stress Evaluation of a Typical Vessel Nozzle Using PVRC 3D Stress Criteria: Guidelines for Application. *ASME-PUBLICATIONS-PVP*, 368, 297–301. ASME.
- Porter, M. A., Reinhardt, W. D., & Martens, D. H. (2001). Comparison of Limit Load, Linear and Nonlinear FE Analysis of a Typical Vessel Nozzle. *ASME-PUBLICATIONS-PVP*, 430, 139–144. ASME; 1998.
- Roche, R. (1989). Practical procedure for stress classification. *International Journal of Pressure Vessels and Piping*, 37(1-4), 27-44. doi: 10.1016/0308-0161(89)90138-5.
- Saxena, M., & Irani, R. K. (1994). A Knowledge-based Engineering Environment for Automated Analysis of Nozzles. *Concurrent Engineering*, 2(1), 45-57.
- Seshadri, R. (2009). Simplified Stress Categorization Using a Single Linear Elastic. *Journal of Pressure Vessel Technology*, 131(December), 1-10. doi: 10.1115/1.4000199.
- Silva, J., & Chang, K. H. (2002). Design parameterization for concurrent design and manufacturing of mechanical systems. *Concurrent Engineering*, 10(1), 3. SAGE Publications. doi: 10.1106/106329302024048.
- Stokes, M. (Ed.). (2001). *Managing Engineering Knowledge - MOKA: Methodology for knowledge based engineering*. London: Professional Engineering Publishing.
- Sunnersjö, S. (1999). *FEM i praktiken* (2nd ed.). Uddevalla: Industrilitteratur.

Appendix A

Dimensions for the comparison of the two cases. Note that the two dimensions of the height of the *lower channel* where the dimensions in brackets [] is for the first case, and the dimensions without is for the second case.

