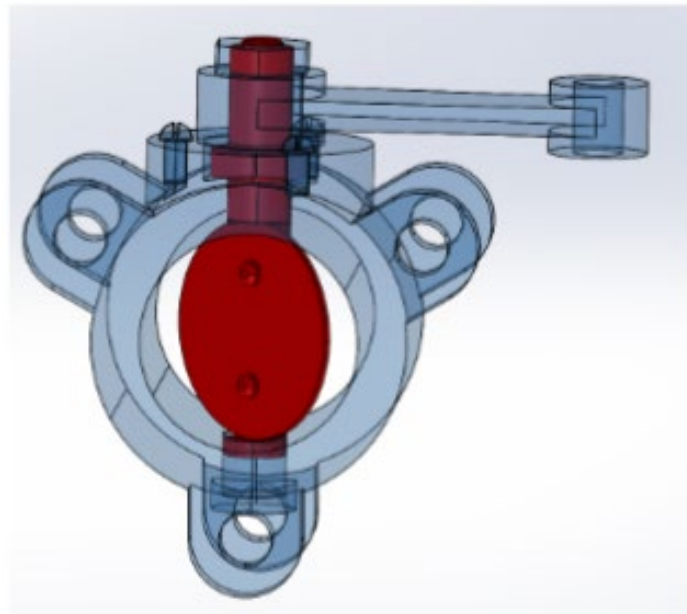


Designing a butterfly Valve: Structural Design

Report written by Andrew Thomas

University of Wales Trinity Saint David



Supported by the University of Wales

Introduction

The purpose of this assignment is to discuss optimisation techniques, both current, historic and future developments. After this, a model will be created with the aid of Design Studio to set up the plys, followed by an optimisation exercise to understand the minimum amount of material needed to prevent water passing through the butterfly valve. Once the work on Design Studio is finished, it will be discussed what the next stage of the development process will be as well as how to carry out the next stage of the experiments. This will all be backed up by technical literature supporting the claims.

Discuss FEA optimisation techniques and by referencing appropriate sources, show how they have developed historically and discuss how you think they might develop in the future.

Finite Element Analysis (FEA), is a simulation process of a given physical phenomenon by using numerical techniques. From a historical point of view, FEA developed from the need to reduce the number of prototypes being produced, in order to work out the greatest optimisation for a part. For engineers, the constant need to reduce waste, reduce lead time on new components and keep up with competitor's new developments were all factors in the development of FEA.

FEA optimisation can save money by reducing the number of physical prototypes parts and experiments. It also needs to be considered that using FEA will give us the optimized amount of material needed for the component. In the case of the butterfly valve, the purpose of FEA is to show the engineering team the optimized amount of material needed to prevent any liquid passing through the valve.

Continuous Optimization and Discrete Optimization are different types of Optimization. Models that have discrete variables are discrete optimization problem. These are a lot harder to solve than Continuous Optimization. A technique that needs to be discussed is the objectives and constraints while optimizing. It's already well aiming setting a goal of getting a certain amount of material as thin as possible, but it is important to remember that there will always be constraints that need to be made, to make the part safe and functional. This can become more complicated when there are multiple objects and multiple constraints.

Historically, FEA started by engineers using partial differential equations to work out the optimised techniques. The partial differential equations are complicated equations that need to be solved to feedback to engineers the strains and stresses, so that the engineering teams can understand the structural behaviour under given loads. The true origins of FEA are not clear, some sources say it was being used as early as the 16th Century by Euler, however, the earliest recording of FEA being used is by Schellbach, in 1851. Until the 1950s, FEA wasn't applied in industry in general and where it was being applied, independent practices were used in different industries, such as aerospace, civil and mechanical engineering. All this being said, work by Turner, Clough, Martin and Topp (1956) and Strang and Fix (1973) were the real foundations in which FEA software would be built upon. It should be noted that early software was slow and could take days to process even a simple object.

To correctly optimise using FEA we need to make simulations, a mesh which consists of millions of small elements that come together to create a structure. Calculations are made in this case for every single element. When combining the individual results, then it is possible to work out the final result of the structure. Using this, we will know the values at certain points within the element, but not necessarily at every point. We call these points nodal points and are usually located at the boundary

of the element. FEA mesh convergence is often the most overlooked issue. This is referring to how small the elements need to be, to ensure that the results of the analysis are not affected by the changing size of the mesh.

To understand where FEA will go in the future, there first needs to be an understanding of what it's being used for at the moment. Right now, the Boeing 787 is being designed using FEA technology for an optimised design and in the medical field on heart stents.

The advancements in FEA will also come from advancements in other technologies. Thirty years ago, it couldn't have been imagined that computers would have the power that they have today. Right now, Quantum computers are becoming more readily available. This will mean that complicated designs will be able to have potential infinite number of nodes and will be able to calculate the effects in a matter of seconds. The latest technology revolution that will assist FEA, is AI working with humans to save even more time from the initial idea to the first successful product.

Also, FEA could be upgraded to be more specific to the manufacturing role. There could be an optimise option that takes into consideration the needs of manufacturing. For example, in the automotive industry you don't work to one specific dimension. Engineers work within a tolerance range. So, once the software has optimised the process, it could then automatically add on a factor of safety and make that the bottom limit, and then suggest to the engineer what it believes would be the safest nominal and upper limit.

Create a model of the plate with the correct material properties in Design Studio. Lay this up with 25 plies of the UD. 5 at -90 degrees, 5 plies at -45 degrees, 5 plies at 0 degrees, 5 plies at 45 degrees and 5 plies at 90 degrees.

Layer 1					
	857142 Steel	0.005	-90.0	Yes	+ -
Layer 2					
	857142 Steel	0.005	-45.0	Yes	+ -
Layer 3					
	857142 Steel	0.005	0.0	Yes	+ -
Layer 4					
	857142 Steel	0.005	45.0	Yes	+ -
Layer 5					
	857142 Steel	0.005	90.0	Yes	+ -

Optimise the structure to minimise the mass of the plate whilst meeting the displacement constraint. Your variables are the number of plies at each orientation. These plies can be placed at -90 degrees, -45 degrees, 0 degrees, 45 degrees and 90 degrees.

The objective in this task is to minimise the mass of the object which will overall reduce the cost when it goes into production. The balance involved with this is that the valve must seal, and in a practical environment, an expectation of 0.1mm maximum deformation is all that is allowed.

Based on work from the previous assignment, the overall force acting upon the valve below is 1553.32N. There are 1438 nodes below, so the individual force acting upon all of the nodes is 1.0799N. It's also noted that the centre of the valve is supported by the pin of the valve and therefore is not subject to any deformation.

Edit a Group : Choose Composite Property Variables

Z0:	Not Designed
Nonstructural Mass:	=0.0 (Not Designed)
Q0:	Not Designed

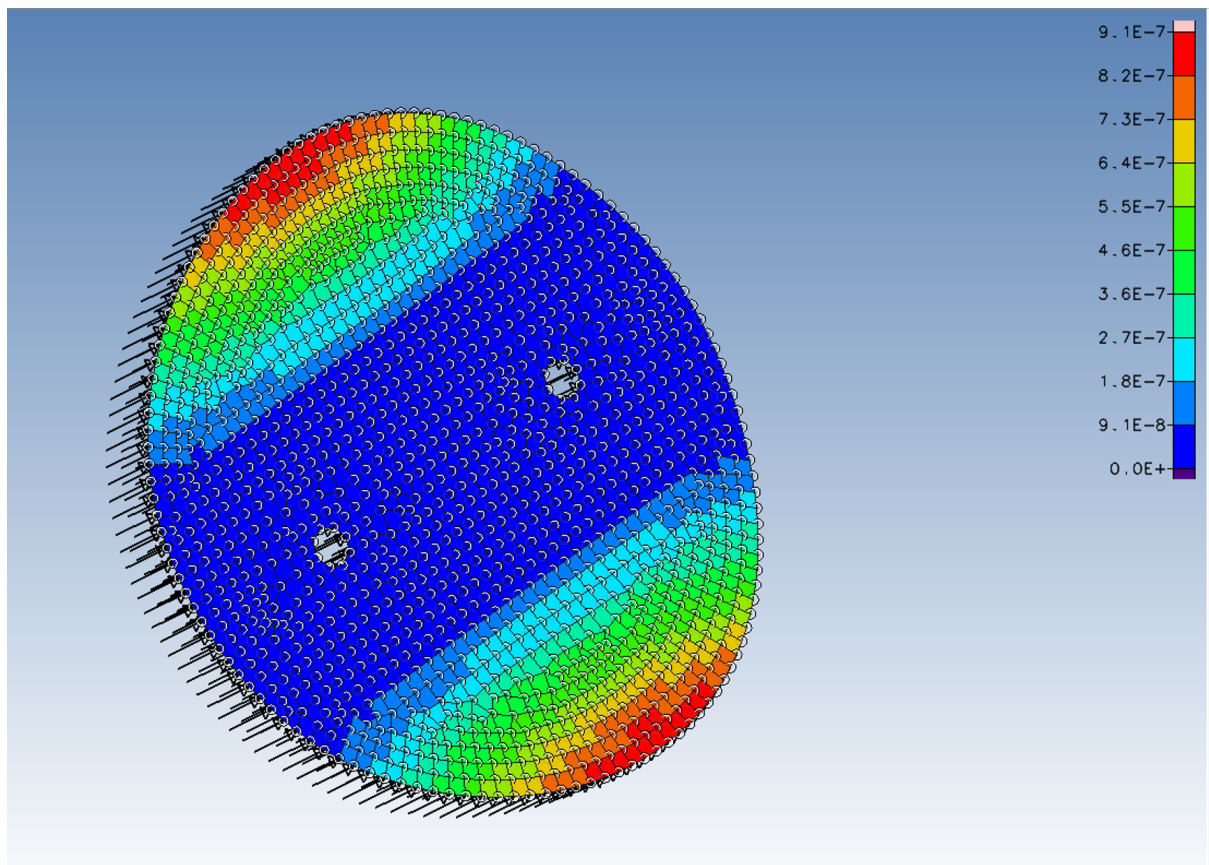
Layer 1	
Material:	857142 Steel
Thickness:	1 LK1TH $\approx 1.E-4$
Angle:	≈ -90.0 (Not Designed)

Layer 2	
Material:	857142 Steel
Thickness:	2 LK2TH $\approx 1.E-4$
Angle:	≈ -45.0 (Not Designed)

Layer 3	
Material:	857142 Steel
Thickness:	3 LK3TH $\approx 1.E-4$
Angle:	≈ 0.0 (Not Designed)

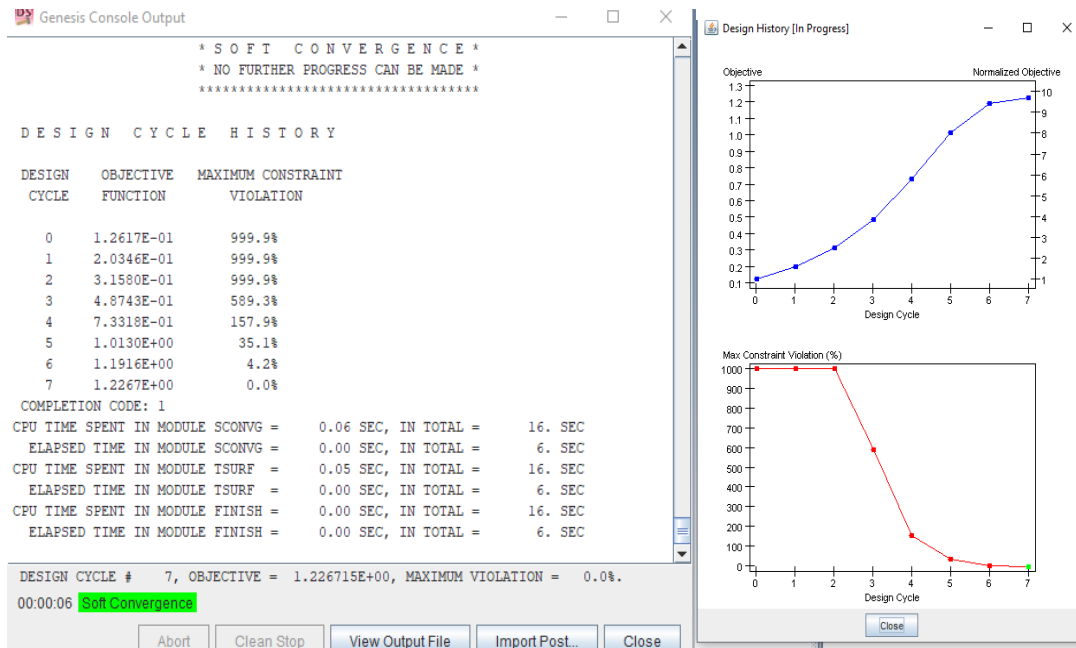
Layer 4	
Material:	857142 Steel
Thickness:	4 LK4TH $\approx 1.E-4$
Angle:	≈ 45.0 (Not Designed)

Layer 5	
Material:	857142 Steel
Thickness:	5 LK5TH $\approx 1.E-4$
Angle:	≈ 90.0 (Not Designed)

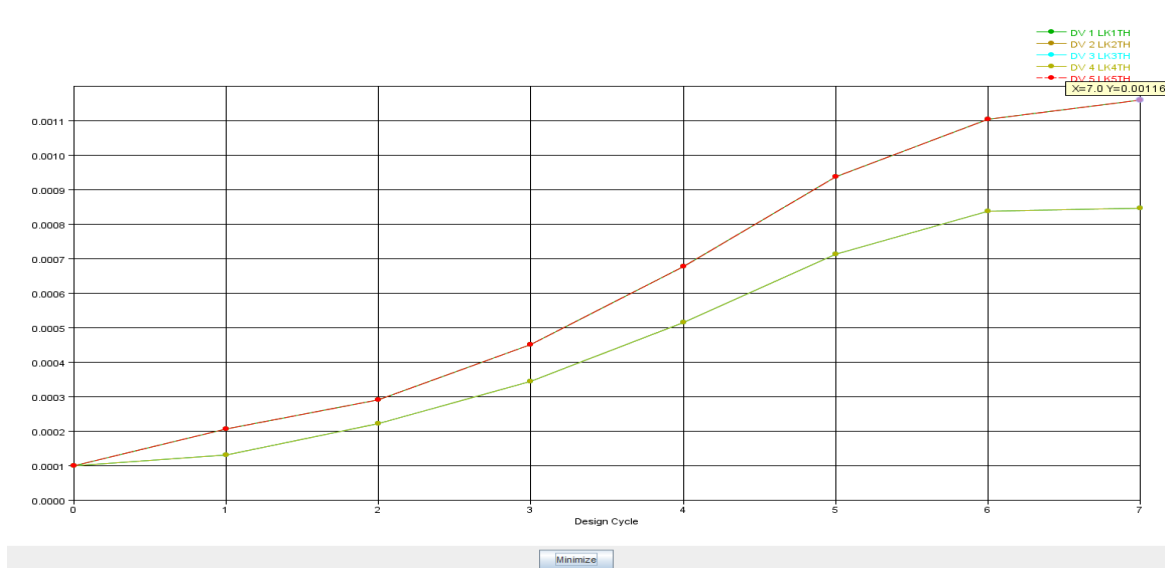


The above photo shows a single analysis of the valve with the forces acting against it. From this it can be seen that the area where there is maximum deformation is on the edges. This is understandable as it is the area that is furthest away from the supporting pin. It can also be seen that the maximum displacement on this is 9.1mm. This goes against the purpose of the valve, as in the real world this would lead to petroleum seeping through.

As a result of this, more material will be needed to be added. At this point, optimisation is needed to determine the minimum amount of material for the maximum amount of deflection.



During the optimisation process we can see how the initial thickness violates the constraint by over 1000%. The initial value was 0.1 grams and by the end this had gone up to 1.2 grams to achieve the constraint of 0.1mm deformation.

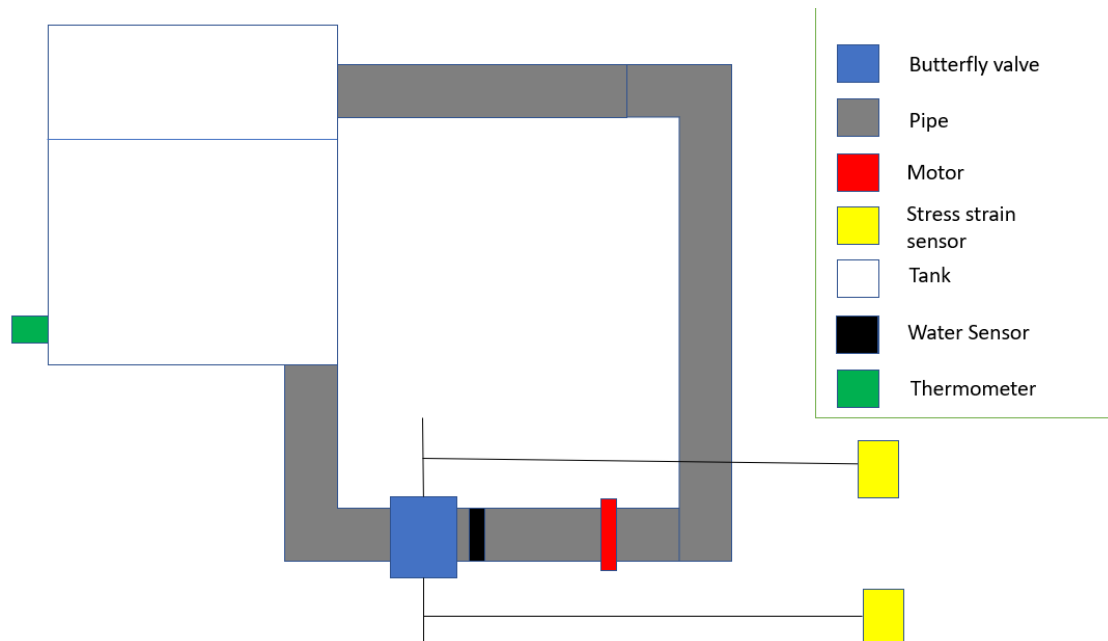


The above shows how the plys at +90 and -90 started at 0.1mm but have then been optimised to increase by just over 1.1mm. The plys at +/- 45 degrees have changed to 0.75mm.

During the development of this component, it is proposed that a prototype is produced and tested in a laboratory to understand the deformation of the valve plate when subjected to the hydrostatic forces seen and to correlate it to the simulation carried out in Task 3. Discuss how you think this experiment should be carried out and how the acquisition of data should be obtained and compared to the simulation.

To move this project from the FEA software to a lab environment, it would usually be tested in a water tunnel, or on a small-scale practical simulation. Once the experiment is set up, if any changes were required it should be again checked using FEA to make sure that practically all of the butterfly

valve parameter would not deform. Not all of the time, there are occasions when we have to scale down the size of the component to fit it inside of a water tunnel.



The above shows a diagram of the practical experiment that could be set up after the FEA analysis. This experiment would give the experimenters a practical idea of what the stress and strain would be on the butterfly valve. To explain what is being shown, a tank would be fitted and secured on the floor, from this there would be two openings. For the sake of the experiment, there would be a known proven butterfly valve next to the tank so the set up of this operation could take place. Piping would be fitted and to save on waste, there would be an inlet and outlet opening. The optimised butterfly valve would be set up and next to it a motor. This is so when the fluid eventually passes through the butterfly valve when it is opened the motor can return it to the tank, so that the experiment can continue. Attached to the butterfly valve would be a stress and strain gauge, so to monitor the real-world effects on the butterfly valve. Behind the butterfly valve there is also a water sensor. The purpose, after all, is when the butterfly valve is closed to prevent water from coming through.

The purpose of the stress gauge is to show a comparison between what the forces predicted on the FEA are against the real-world forces. If there is a considerable difference, then the new forces should be tested in the FEA. If the forces were less, this could mean that a cost saving could be made by optimising to use less material. If the forces are greater, it will mean that more material is required to create a safe product.

The reason that a real-world experiment is needed is FEA only gives an approximate solution to the problem and is a numerical approach to get the real results of these partial differential equations.” (4.1) This is referring to FEA being a numerical method which should be used as a prediction tool of how the part should behave under a given condition. At the moment, FEA shouldn't be completely used instead of practical experiments. For now, FEA should be used only as a simulation to understand the design problems.

To take the above point further it needs to be remembered that FEA is based around assumptions. As engineers, it is assumed that the load will be equally distributed, it's assumed that the load will never go above what has been calculated and it is assumed that the material will be the specific

dimensions/material properties that have been entered. This isn't true to the real world where mistakes can occur.

From the optimised FEA model, specific points of stress and strain can be seen. Even though the centre of this part should be checked to make sure there isn't any stress and strain, the main focus needs to be on the outside areas, more specifically in the areas that were highlighted earlier on during the FEA optimisation.

The first part of the experiment after the initial set up is to simply apply gravity fed forces against the butterfly valve. From this, the initial stress will be detected and the fluid sensor will pick up upon if any petroleum has passed the valve. Once the petroleum has become stable against the valve (before the petroleum entered there would have been air, the butterfly valve would be slightly open so the air can escape.) A criticism of this set up would be that some petroleum would initially slip through due to the valve being slightly open. Once the valve is closed the sensor could be reset. This test is referred to as the closure test.

Providing the initial test is passed, several further tests will then be performed, to fully understand the capabilities of the valve, a test fluid temperature test should take place. This is where the petroleum is dropped to 5 degrees Celsius and then increased to 40 degrees Celsius to see if this has any effect on the valve.

A shell test should also be completed, which is the cold working pressure rating of the valve for the purpose of validating the strength and soundness of the valve pressure containing and retaining structures.

On the specific comparison to the simulation, the stress and strain gauge should provide feedback showing when there is the most amount of stress on the real valve. This can then be compared to the simulation. Questions such as "Is this the same deflection being measured? If not, why?"

Relate your design to any technical literature and standards that cover this component

Most of the focus around this experiment was set up with the International Organization for Standardization (ISO) in mind. The specific ISO that covers this is ISO 5208:2008, the purpose being to standardise practices and testing of valves. Note, that this ISO doesn't cover health and safety, that will be discussed later.

During the test/experiment it was mentioned that high pressures should be tested that are greater than what the system is to be designed for. ISO 5208:2008 clearly defines those parameters that need to be decided upon so to adequately inform the buyer of what the maximum pressure that can be applied to the product is. In ISO 5208 it states several of the tests that should take place. This includes the Shell test and test fluid temperature.

Other standards that cover this component include ISO 10497:2010 Testing of valves. As petroleum is being used it is highly likely that the end usage of this will be that it goes through the combustion process. To comply with this the system it must be tested at 2 bar which is the equivalent of the pressure that would be expected in the event of a fire at normal factory setting.

A piece of literature that has been written on butterfly valves is by Kuldeep Narwat (5.2) and is a review of the butterfly valve. The main points are that the butterfly valve should always be as simple as possible from a construction point of view. The review also points out that it's the weight of the disk that is also the most important factor. The cost of the butterfly valve is often closely associated with the weight of the overall product.

Conclusion

From this the optimised condition of the valve can be seen, based on the individual forces acting upon the nodes. The FEA software is a great tool in supporting an engineering team in the development of products and getting them to mass market. Being trained on FEA alone however wouldn't be sufficient enough, as a broader understanding of how different manufacturing sites requirements is also essential in understanding the tolerance bands and cost-effective procedures. Developments in FEA at the moment are not advanced enough, where a total abandonment of practical tests can be achieved. The software and hardware supporting it is improving though and it does seem like in the future we could live in a world where practical testing is greatly minimised.

References

[Structural Optimization in Finite Element Analysis \(nafems.org\)](#) – 4.1

[ISO 5208:2015\(en\), Industrial valves — Pressure testing of metallic valves](#) – 5.1

[ISO 10497:2010\(en\), Testing of valves — Fire type-testing requirements](#) – 5.2