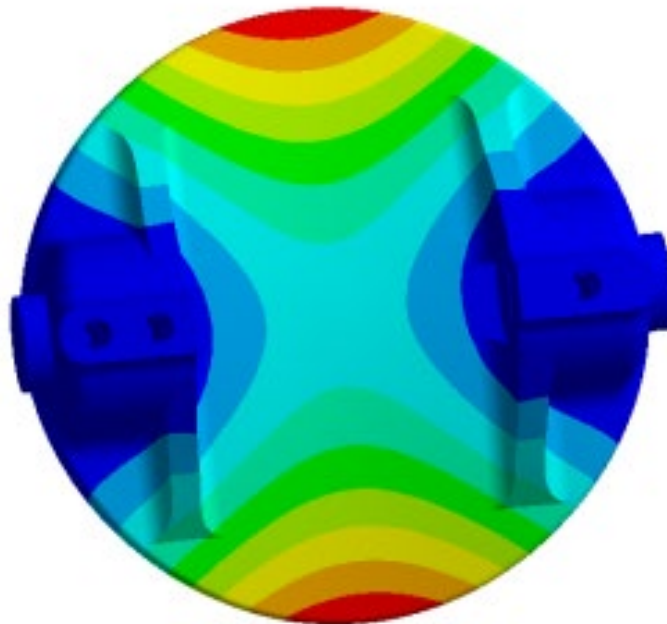


MATTHEW FORD

**DESIGNING A BUTTERFLY VALVE: STUCTURAL
DESIGN**



STUDENT NO. 1903400

LECTURER K LAKE

ABSTRACT

In the following report, Finite Element Analysis will be discussed, referring to its origins and uses. FEA will also be used to design a component for a butterfly valve. The component will be evaluated and a comparative practical experiment designed to validate the FEA data. This experiment will also be evaluated with reference to technical literature and standards.

Table of Contents

Abstract	2
What is FEA?	4
Future of FEA?	6
Optimizing a design using Design Studio Software	7
Final optimization results and comparison	16
Prototype testing	17
Testrig legislation	18
Bibliography	20

What is FEA?

Finite Element Analysis (FEA) is a method used by engineers to evaluate component designs (notes, 2009). By using known engineering principles and using computing power to solve large numbers of differential and integral equations, the various software packages available can give engineers an accurate idea of how the design of a component will stand up to the various stresses and strains that might be exerted onto it, all this in the theoretical realm, negating the need to manufacture and test expensive prototype designs. That said, it is sometimes necessary to do this to confirm the computed results. As a point of note, the “Finite Element” term was coined due to the technique of breaking the design down into more simple parts, i.e., finite elements.

FEA has been developed over the last 80 years and began in the 1940’s as a concept that was derived from several scientific papers published at the time. Principally conceived as a way to solve complex structural and elastic analysis problems within the aeronautical industry, A. Hrennikoff and R. Courant were the fore runners of FEA as they were the first to take mesh discretization and break it into subdomains, called ELEMENTS. The term Finite Elements was then coined in an article in 1960. A further recognised contribution to the history of FEA was in 1973, when Gilbert Strange and George Fix penned the book “An Analysis of the Finite Element Method”. Other key milestones in the history of FEA were: -

- 1959, the computer system Design Augmented by Computers (DAC-1) was built by GM and IBM
- 1965, NASA developed its own structural analysis tool, NASA Structural Analysis (NASTRAN)
- 2006, The American Society of Mechanical Engineers release the Guide for Verification and Validation in Computational Solid Mechanics.
- 2008, a FEA standard is released by NASA

(Crahmaliuc, 2020)

There are many FEA techniques used today, the main reason for this is to cover as many different aspects of real world situations as possible. Ideally, it is desired that all the parameters of a design could be imported into a software package, which would then present all the relevant engineering information required. Due to the infinite variations and influences of real world situations, FEA cannot cover all this in a single analysis, therefore different techniques are used depending on what data sets are required.

For example, Linear analysis is a very common FEA technique used, calculating loads, stresses, strains and other engineering data, all under **STATIC** conditions. This technique is relatively simple and involves making many assumptions, and makes no consideration for **DYNAMIC** influences, i.e., time. All calculations are based on one point in time. (Cyprien, 2017)

Dynamic analysis, this technique is more complicated than linear analysis, and takes into account the effect of time on a design.

Modal analysis concentrates on the vibration characteristics of a design. It scrutinizes the natural frequencies of components and compares them to the rest of the system thus ensuring that there aren't too many similarities, which could lead to resonance within a system. This is used extensively in the automobile industry to test engines and components under different loads and speeds to ensure there are no issues.

Other techniques include but are not limited to: -

- Crash analysis
- CFD analysis
- Fatigue analysis
- Buckling analysis
- Thermal analysis

(notes, Types of Analysis, 2021)

The Future of FEA?

There are many interesting ideas on where FEA is going over the next few years, some engineers, predict it will be completely integrated into design packages such as Solidworks, whereas others consider FEA to be too specialist a field, and state that it should be left as its own entity. Another prospect being researched, is the possibility of the process going “meshfree”. Meshing is the process of inputting a designs shape into the FEA software, currently it is not standards across different software creating a headache for engineers using multiple different platforms. The idea of going meshfree would eliminate this problem, and hugely reduce the manhours involved in creating meshes and balancing the complexity of the mesh against the accuracy of results. Currently companies are looking into this possibility (Roade, 2018). Another exciting idea currently under development is that of using artificial intelligence. This would involve Artificial Neural Network (ANN) learning the behaviour of a material from real world results. This will make FEA results far more accurate due to the fact that less approximations would be required at the input stage of the process. (Akbar Javadi, 2009)

OPTIMIZING A DESIGN USING DESIGN STUDIO SOFTWARE (Genesis Key Features, n.d.)

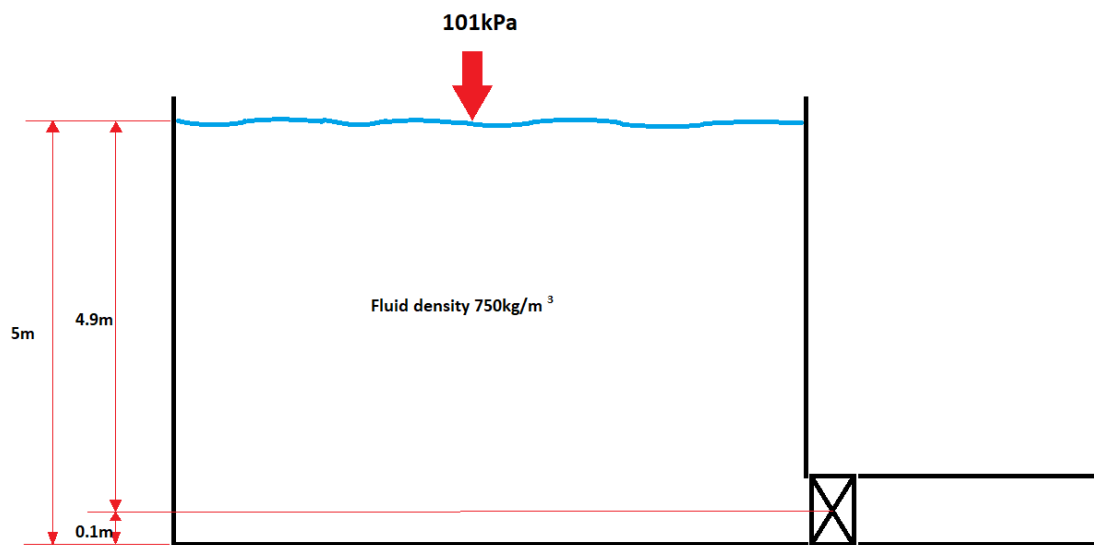
Following, is the process used to simulate a load on a component, the process of changing a component material, and the process of layering a design model in such a way that it can be manufactured practically, and with the least amount of mass whilst still maintaining functionality. This whole process is done within a software package known as Design Studio and will be carried out on the disc of a butterfly valve.

Before the software can be used, and the design optimized, there are several assumptions that must be made. These are: -

- The pin of the valve that supports the disc is totally confined, for the purpose of this analysis, it will have no deflection due to the forces acting on it
- The tank onto which the valve is intended to be mounted is 5m high, and full, as this is the condition under which the valve will experience the maximum load
- The tank is vented so that atmospheric pressure is acting on the fluid surface (101kPa)
- The valve will be mounted onto the side of the vessel at the lowest possible practical point
- For the purpose of this simulation, the force exerted on the disc by the hydrostatic head of fluid will be uniform across the disc, no allowance will be made for the presence of the support pin.
- The fluid acting on the disc will be gasoline, with a density of 750kg/m^3

The first step in using the software is to import the correct data set for the component. Following this, a load set needs to be created and applied to the design. In this case, the load set consists of the pin constraints, and the hydrostatic force from the fluid. The pin is constrained so that none of the nodes within the pins border can move, the hydrostatic force must be calculated per node of the design. This is done by first calculating the pressure force exerted on the disc:

$$\text{Hydrostatic force} = P_{\text{head}} + \rho gh$$



Where P_{head} is the pressure head = 101kPa
 ρ is the density of the fluid = 750kg/m³
 g is gravity = 9.81m/s²
 h is the height of the fluid acting on the valve center = 4.9m

Therefore, Hydrostatic force = 101000 x (750 x 9.81 x 4.9)
 = 137051.75 Pa
 = 137 kPa or 137 kN

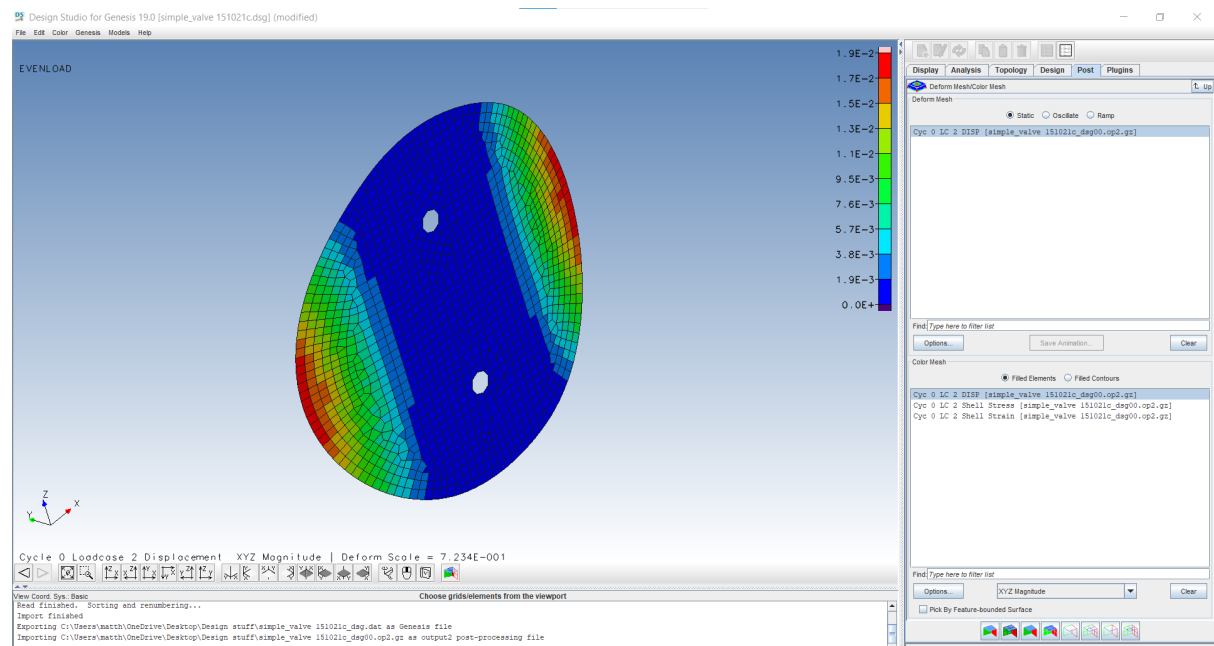
Then, the disc area is πr^2 = $\pi 0.097^2$
 = 0.02956m²

So the force exerted on the disc = 137000/(1/0.02956)
 = 4049.72N

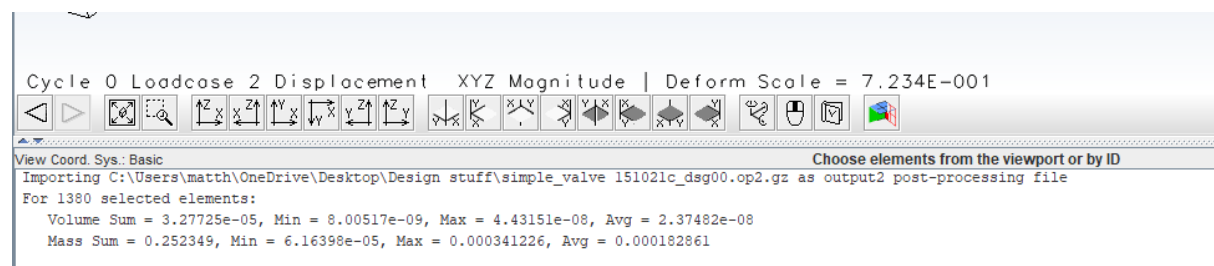
and then by dividing this figure by the number of nodes in the disc. By selecting all nodes within the disc in Design Studio, it can be determined that there are 1438 nodes, therefore each node has 4049/1438 Newtons exerted onto it.

Once this information has been input into the program, a “single analysis” can be carried out where the program calculates the deflection on the valve disc. Nb. At this point, the raw data that has been imported into the software has the disc made from steel at 0.01mm thick.

Below is the single analysis of the Steel plate, showing it loaded and deflected, the deflection at maximum is 0.019m, 19mm

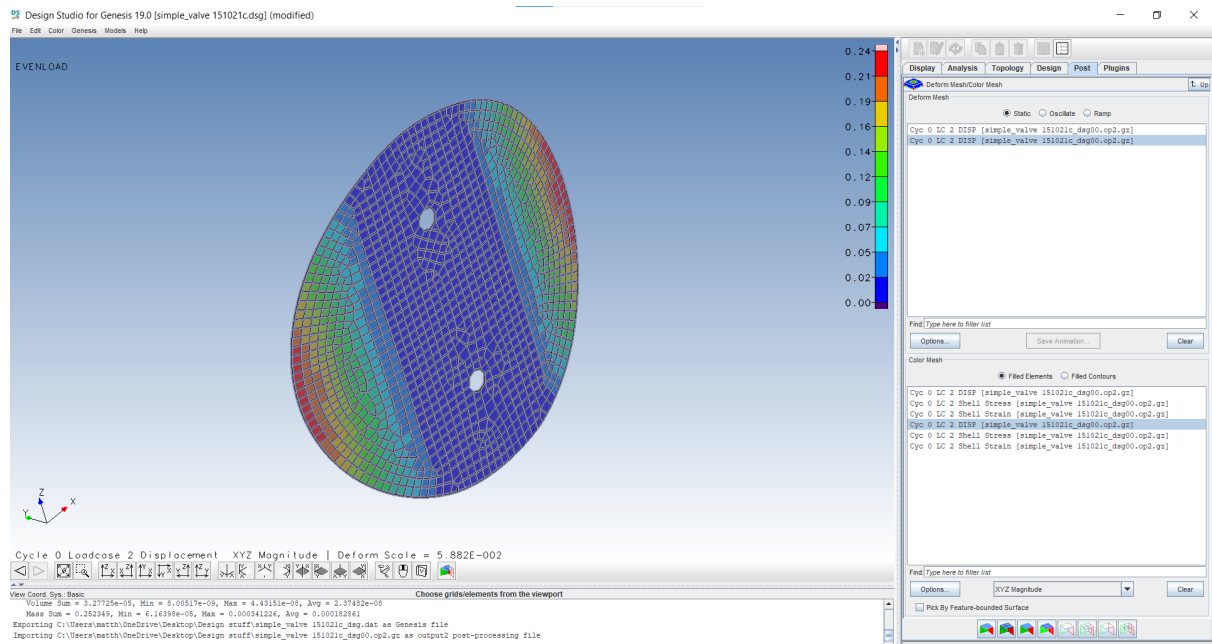


Using the software, the weight of this structure can be seen to be 0.252349kg



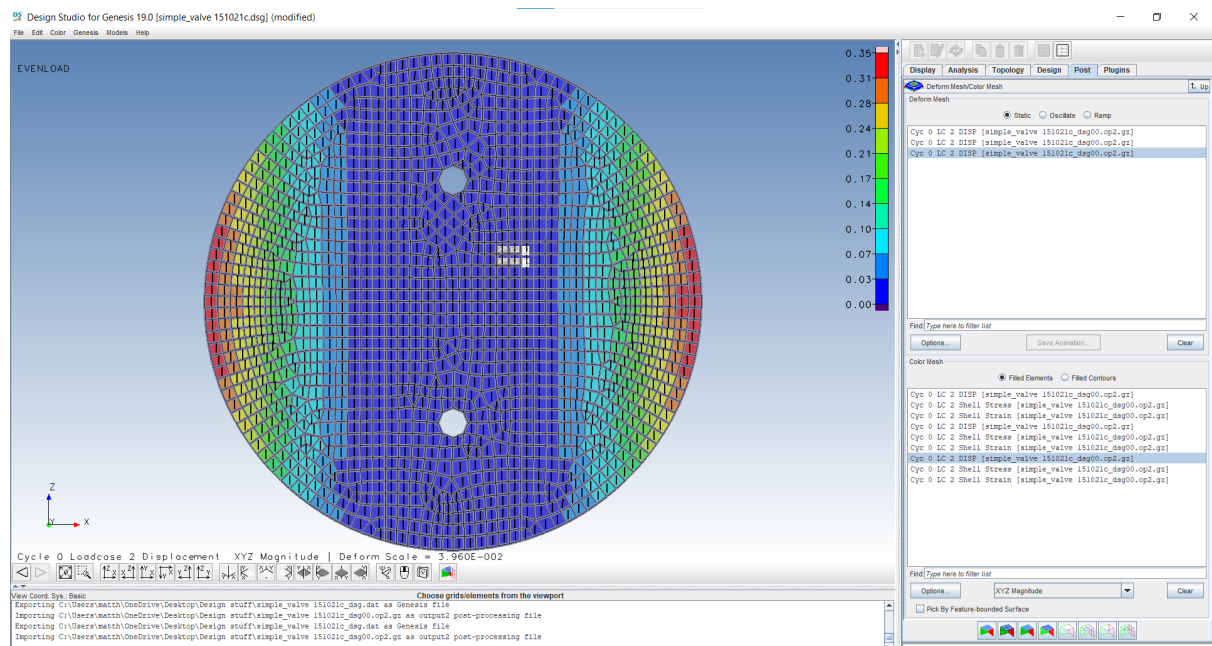
The software can then be used to create a second material by defining new material properties as per those specified for the CFR plastic. A single analysis can then be carried out on the disc now designated as being made from new material.

Below this analysis is shown, the deflection now is, 0.24m



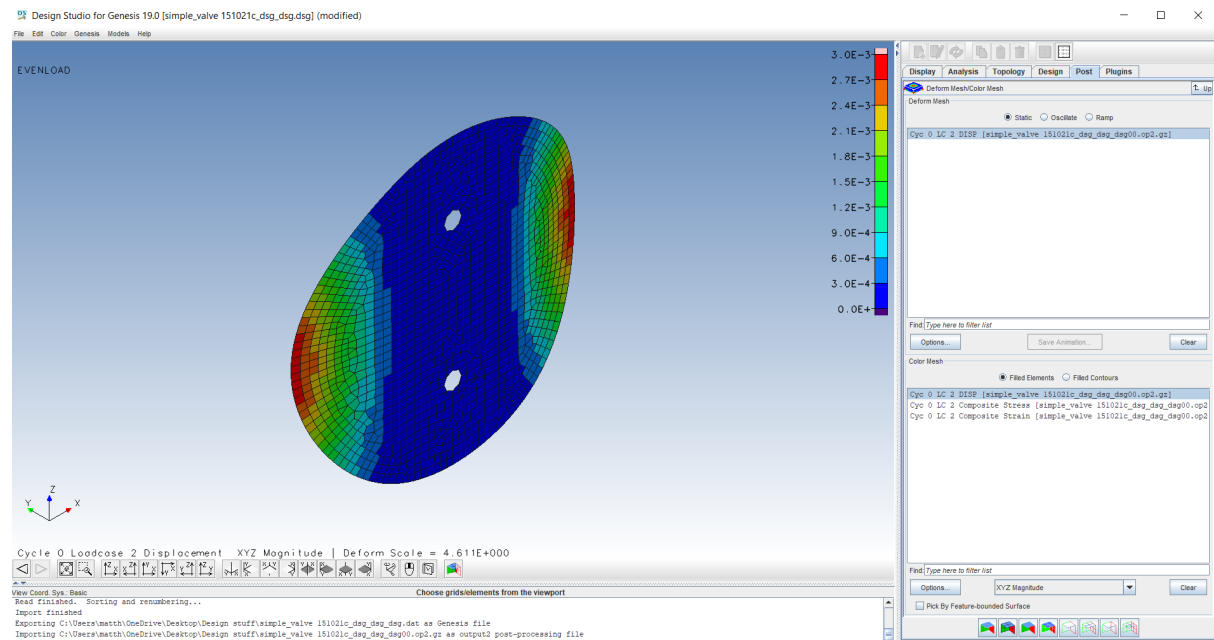
The next step in the process is to ensure that all the elements within the material are running in the same, correct orientation.

Below is the analysis after orientation of elements, the deflection is now 0.35m as all elements orientated causing a “worst case” scenario for the material due to the elements running in the weakest direction.



From here the design is given the relevant 5 layers as described in the brief, each layer running at 90° to each other, and each layer being 5 plies of 0.1mm thick.

Again, a single analysis is performed, below shows this composite layed up as described, the deflection can now be seen at 0.003m

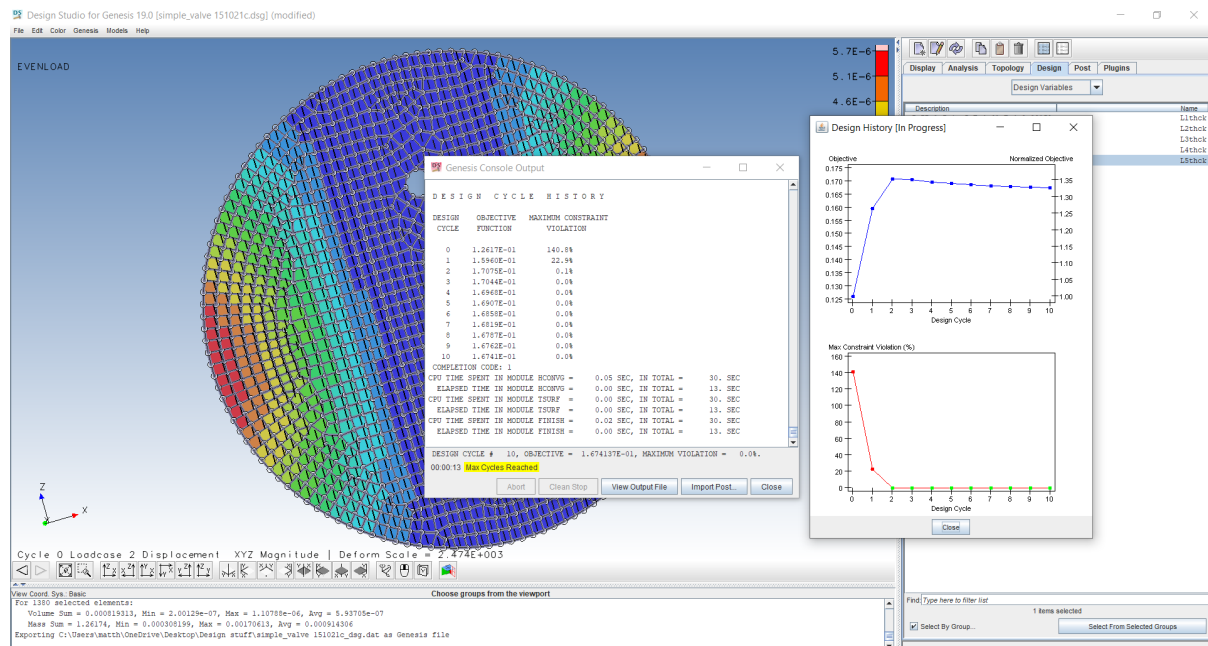


The mass of this “completed” structure is 0.126174kg

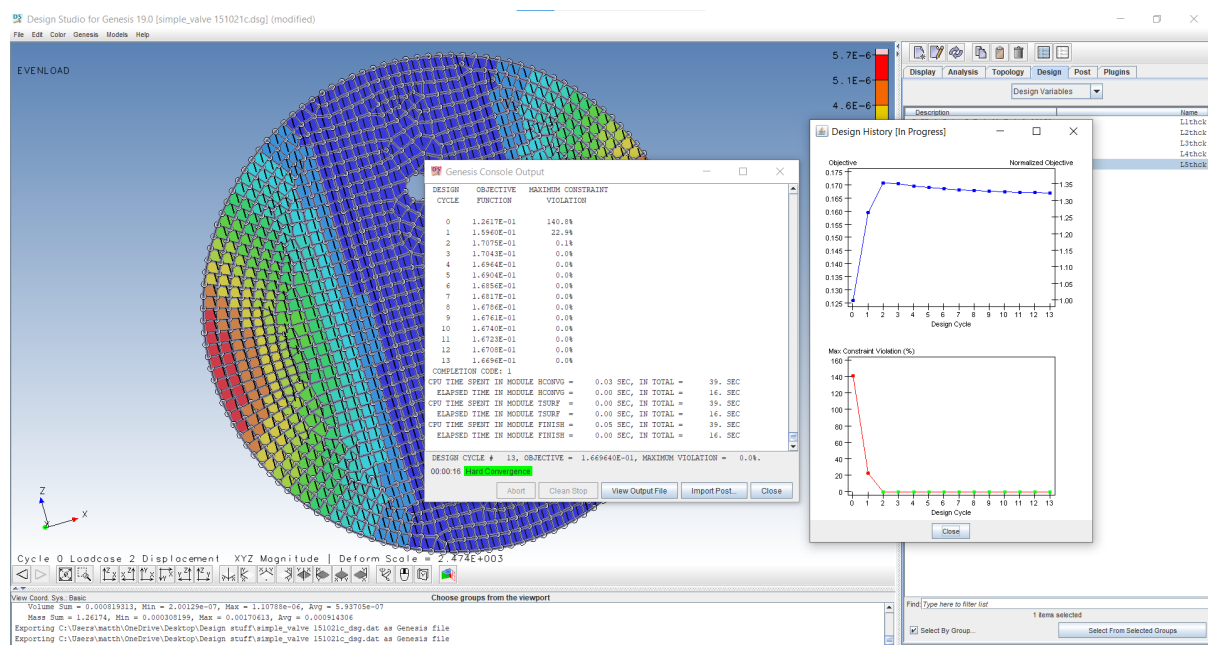
View Coord Sys.: Basic	Choose elements from the viewport or by ID
Importing C:\Users\matth\OneDrive\Desktop\Design stuff\simple_valve 151021c_dsg_dsg_dsg00.op2.gz as output2 post-processing file	
For 1380 selected elements:	
Volume Sum = 8.19313e-05, Min = 2.00129e-08, Max = 1.10788e-07, Avg = 5.93705e-08	
Mass Sum = 0.126174, Min = 3.08199e-05, Max = 0.000170613, Avg = 9.14306e-05	

The next step is to optimize the structure by setting objectives, constraints and variables. The objective is to minimize the weight of the component, the constraint is that the component should only deflect a maximum of 0.1mm, and the variables being the number of plies, for this report, this variable has been set between 1 and 15. The software can then be utilized to OPTIMIZE the design, calculating the ideal number of plies in each layer and in which of the orientations they should be layed out.

Below is shown the results from this optimization

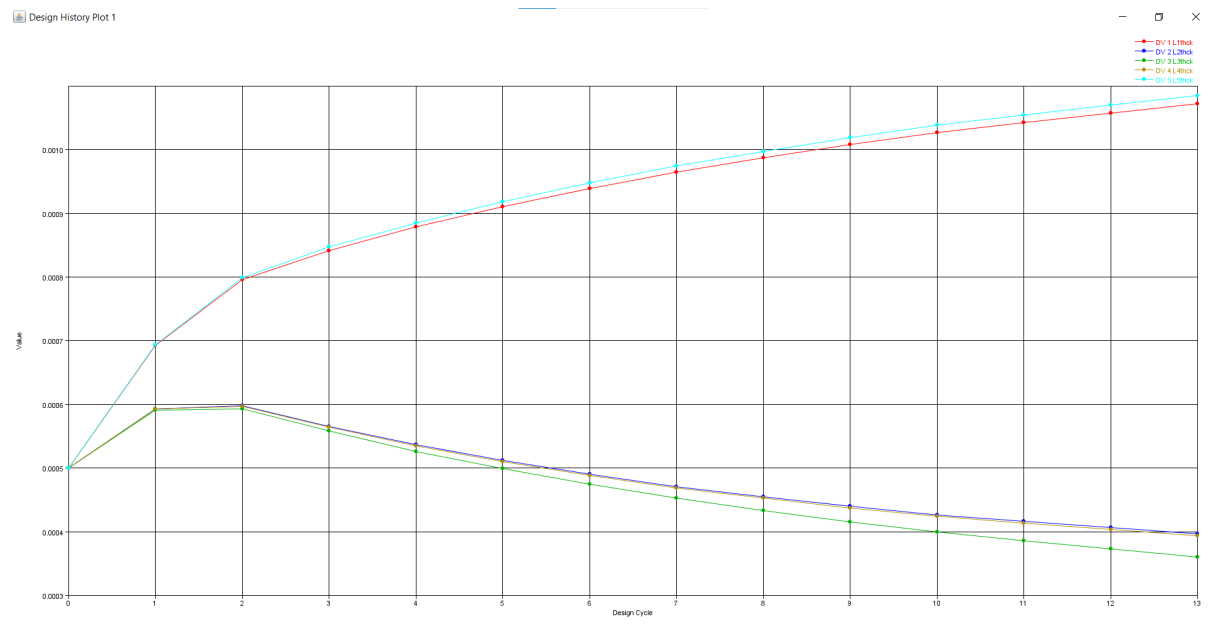


Note that the “Max cycles” has been reached in the program has stopped its calculations before a result has been found, therefore, the cycle number is increased to 50 and the design optimized again. See below.

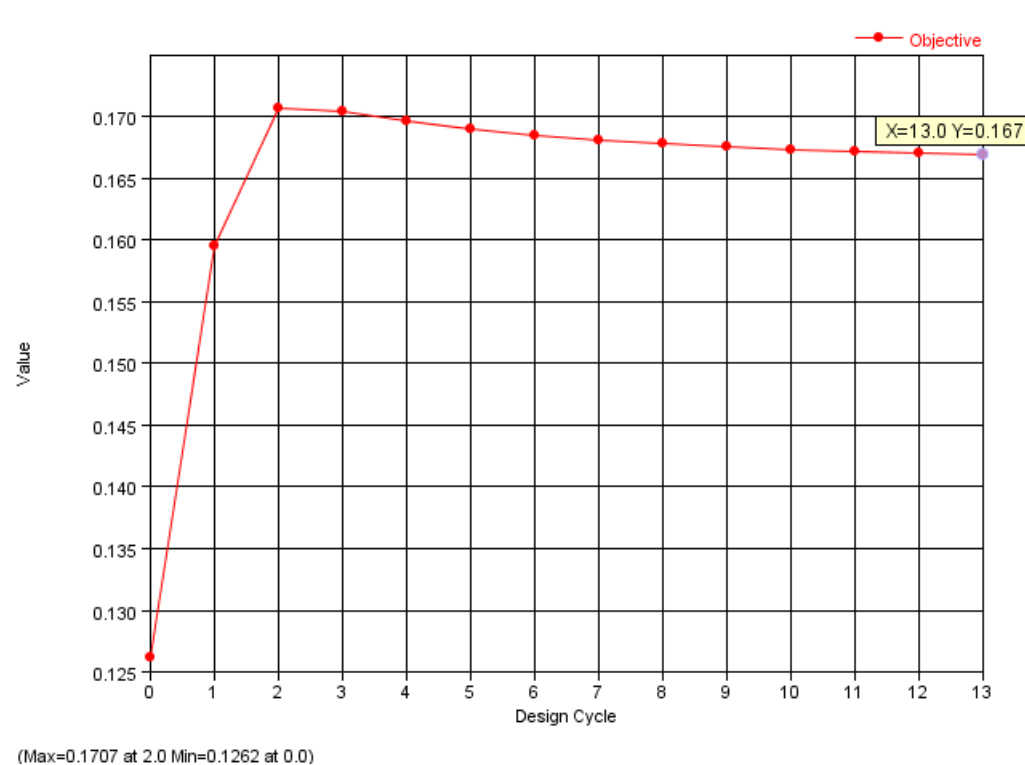


It can now be seen that a result has been obtained in 13 cycles.

The graph below shows the calculated thickness of each of the layers through the cycles

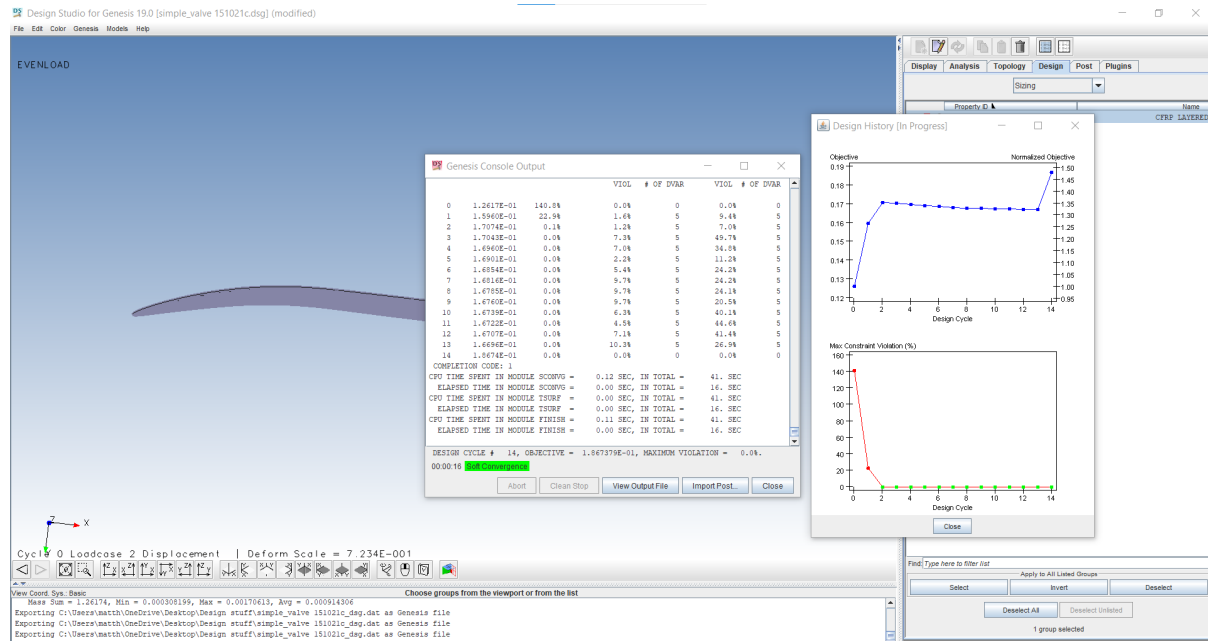


From the graph below that was obtained from Design Studio, we can see from the y axis that the optimised weight is now 0.167kg at 0.01mm deflection

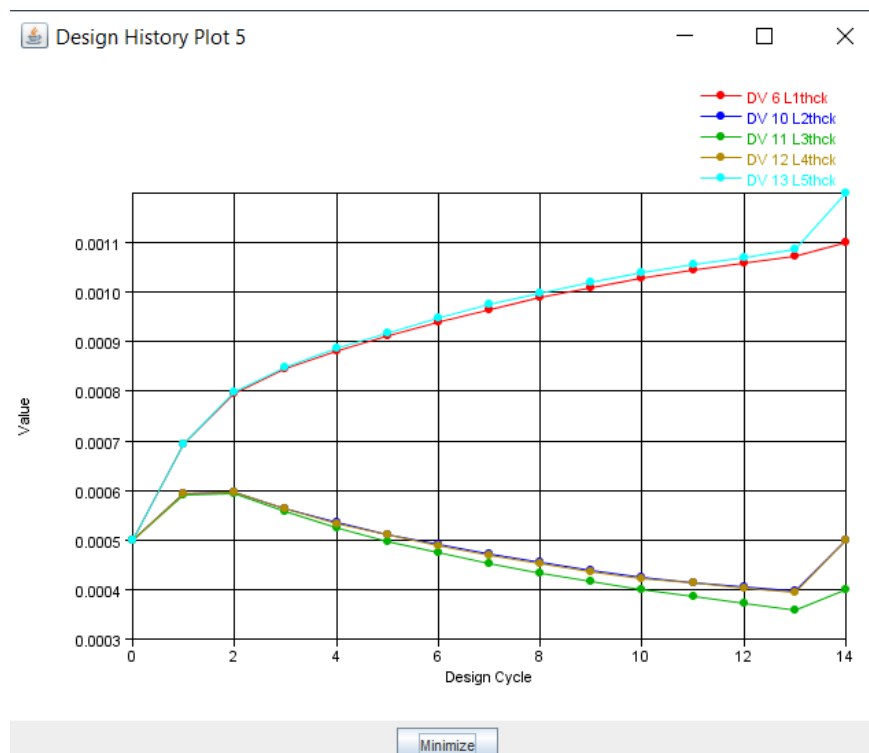


Although these results are the “ideal” layer thicknesses, parts of plys cannot be manufactured in the real world, therefore discrete values need to be set so that a design that can be manufactured can be calculated.

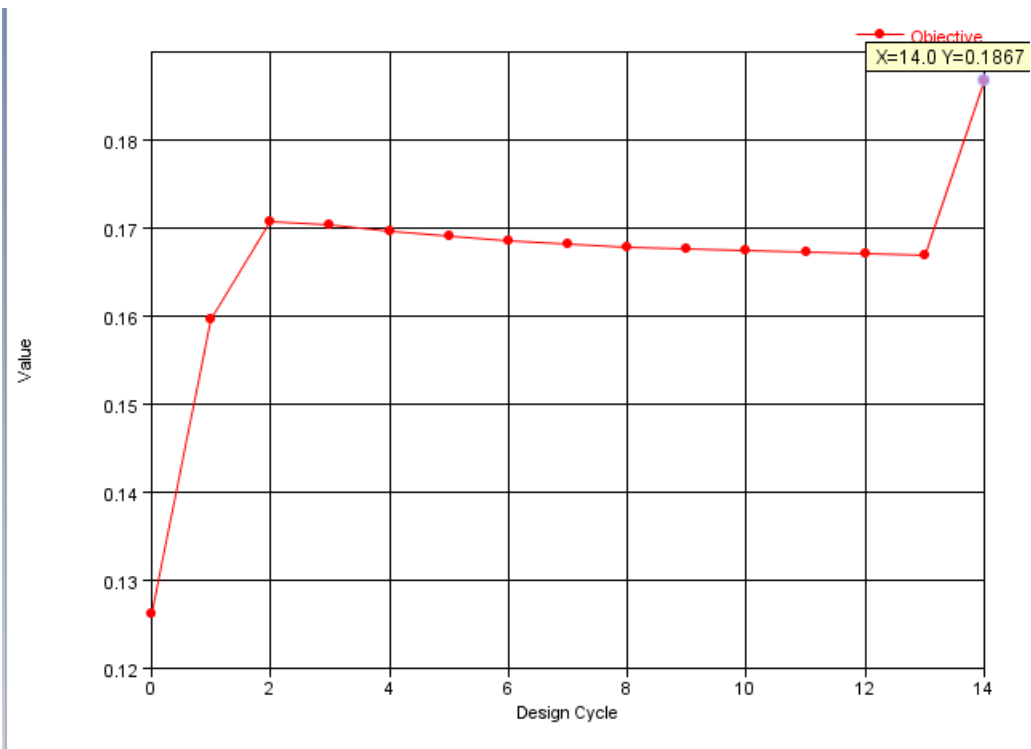
Below is a screenshot of the design being solved with discrete values



Below are the layer thicknesses as can be manufactured ie whole layers



The weight of this new design that can actually be physically manufactured is 0.186kg



Final optimization results and comparison

	THICKNESS (mm)	DEFLECTION (mm)	MASS (kg)	WEIGHT (N)	LOAD (N)	STIFFNESS (N/mm)	STIFFNESS/WEIGHT (N ² /mm)
STEEL	0.1	19	0.252349	2.2096	4049	213	86.04
CFRP	2.5	0.3	0.126	1.23606	4049	13496	10919
CFRP (Optimized)	3.31	0.1	0.167	1.6383	4049	40490	24714.6
CFRP (Optimized and discrete)	3.7	0.1	0.186	182,446	4049	40490	22190.4

From the above table, the progress through the design process can easily be seen.

The first line of the table shows the disc of the butterfly valve, made from steel at a thickness of 0.1mm, the deflection on such a thin plate is massive, 19mm. This leads to a low stiffness value and a very poor comparative stiffness/weight figure of 86.04N²/mm.

Following the material change from steel to CFRP, and the design parameters (i.e., number of plies and layers) increasing the disc thickness to 2.5mm, it can be seen that both the stiffness and stiffness/weight figures have increased substantially.

The next step in the design process (optimization) meant that the CFRP material was layed out in such a way that each layer in any particular orientation was given the ideal thickness in order to obtain the optimum stiffness values. This step increased the stiffness values to 40490N/mm and 24714.6N²/mm.

The final stage of the process takes the optimized values, and recalculates the optimized thickness of each layer, this time, producing whole values of plies, effectively creating a design that is both optimized, and manufacturable in a real world environment. Due to the need for the design to remain above tolerances, this results in the layer thicknesses increasing and therefore an increase in mass which ultimately leads to a slight drop in the stiffness/weight figure to 22190.4N²/mm.

Prototype testing

In order to ascertain that the results obtained from the FEA software are accurate, a prototype valve will be manufactured and tested. The test will be carried out by bolting the valve to a pipeline so that the pipe is vertical and the valve disc is horizontal when in the closed position. This can then be filled with any safe fluid as the fluid density is irrelevant with this test so long as the seal is effective for such a material (i.e., the fluid viscosity should not be too low). Strain gauges will then be mounted onto the “underside” of the valve disc, these will then measure the strain and therefore the deflection of the valve disc. Alternatively, the disc could have dial test indicators (dti’s) set up on it, two, both at 180 degrees to each other, perpendicular to the pin, this will be the two points where maximum deflection is expected to occur. The pressure of the fluid sat on the valve could then be slowly increased using a compressed air blanket on top of the fluid. This pressure is then increased to the point where the dti’s read 0.1mm of deflection. When this is achieved, the pressure exerted onto the disc can then be noted, and then reverse calculations from the one done initially in this report can be made to derive the force in newtons being exerted onto the disc. Theoretically, this force should equal that of the applied force in the software i.e., 4049 Newtons. There will inevitably be variations due to: -

- Fluid friction
- Measurement errors either human or “digital” in the system pressure
- Measurement errors human or digital in the valve displacement
- Manufacturing flaws and/or discrepancies

But so long as the results are close to each other it is a fair assumption that the data acquired from the software is accurate.

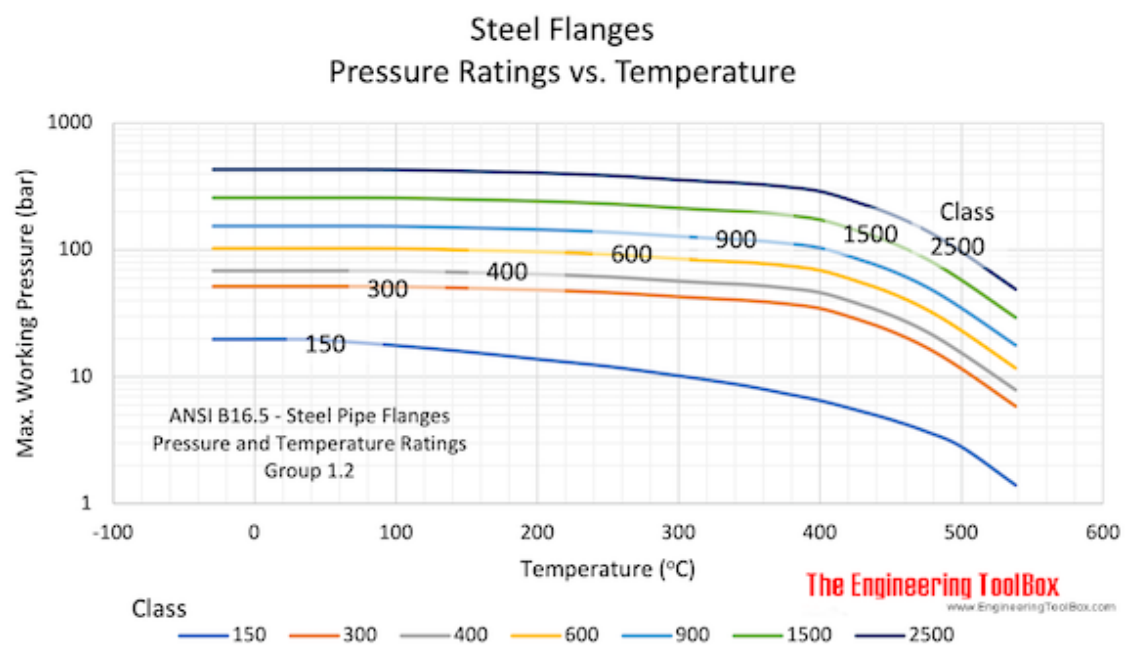
As a point of safety, note that the valve design is untested in the real world. Although it is unlikely due to the advanced modelling techniques already used to specify the valves mass, there is possibility of the valve failing catastrophically. This must be considered, therefore the test should be carried out to minimize risk to personnel and property, either by conducting the experiment at a distance, or in an appropriately designed enclosure.

Testrig legislation

Following on from the safety point mentioned, there are several other safety standards that relate to the designed experiment and must therefore be adhered to.

As the experiment will be carried out under pressure, the pressure vessel code provided in British Standards industry PD 5500:2021 must be adhered to (BSi, 2021)

Assuming that the valve is bolted onto steel pipe, the correct gaskets and pipe specifications must be followed, these can be found under ANSI B16.5. From this standard, it can be seen that a flange rating Class 150 would be more than suitable for the design in question ie. Up to 19.5 bar at 50°C



(Engineering Toolbox, 2021)

If the valve were to be operated remotely for any reason during the experiment, all moving parts must be guarded correctly as per PUWER (the Provision and Use of Work Equipment Regulations 1998) The PUWERegulations will also influence the type of guarding positioned around the experiment as a means of containing any catastrophic failure.

In addition to these safety standards, it is also industry best practice to ensure that all flanged joints are made with the correct gasket, with the correct torque on the assembly components, and that the flanges are aligned both around the periphery and across the faces. Special attention needs to be paid to the choice of gasket material to ensure that the seal is hard enough to create a good seal, but soft enough as to not cause any damage to the CFRP valve body.

Bibliography

- Akbar Javadi, A. F.-A. (2009). An Artificial Intelligence Based Finite Element Method. *ISAST Transactions on Computers and Intelligent Systems*, No. 2, Vol. 1,, 1.
- BSi. (2021). *Pressure vessels*. Retrieved from BSi Group:
<https://shop.bsigroup.com/categories/pressure-vessels-in-energy-and-utilities>
- Crahmaliuc, R. (2020, November 10). *75 Years of the Finite Element Method (FEM)*. Retrieved from Simscale: <https://www.simscale.com/blog/2015/11/75-years-of-the-finite-element-method-fem/>
- Cyprien. (2017, April 17). *Why is there different types of fea analysis?* Retrieved from FAE For All: <https://feaforall.com/type-fea-analysis/>
- Engineering Toolbox. (2021). *ANSI B16.5- Steel pipe flanges*. Retrieved from Engineering toolbox: https://www.engineeringtoolbox.com/ansi-flanges-pressure-temperature-d_342.html
- Genesis Key Features*. (n.d.). Retrieved from GRM Optimized Engineering Design: <https://www.grm-consulting.co.uk/our-software/genesis>
- notes, L. (2009). *Introduction to the Finite Element Method for Structural Analysis*. Dordrecht: Springer.
- notes, L. (2021). *Types of Analysis*. Swansea: UWTSD.
- Roade, G. (2018, August 06). *THE FUTURE OF FINITE ELEMENT ANALYSIS: MESHFREE*. Retrieved from Midas: <https://www.midasoft.com/mechanical/blog/the-future-of-finite-element-analysis-meshfree>