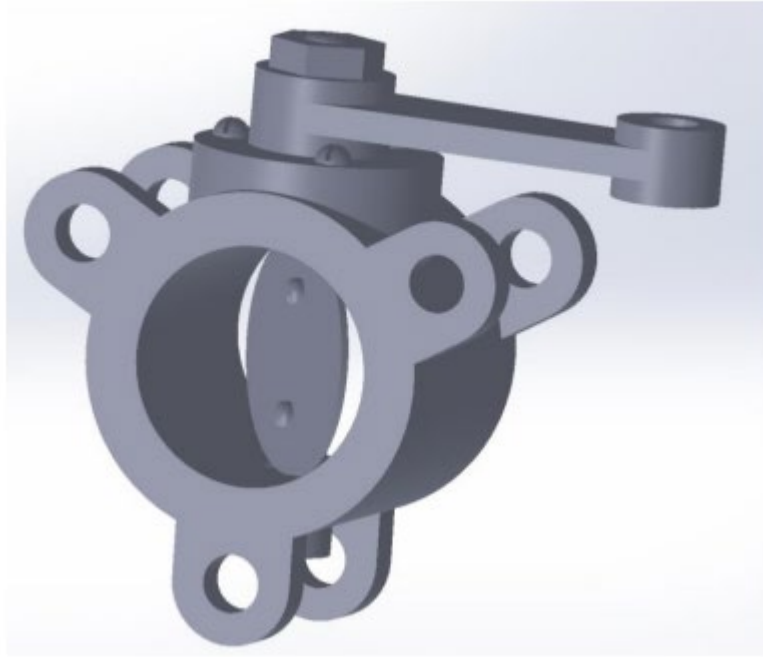


Structural and Fluid Analysis Assignment 2



By Daniel Lloyd Tubb

Contents

Introduction	...3
FEA Analysis	..4-5
Design Studio	...6
How FEA improves the design	...7
Baseline Composite	..8-10
Optimised Composite	..11
Optimised Composite (discrete)	..12-13
Prototype	...14
Conclusion	...15
Bibliography	...16

Introduction

The task was to design the lay-up of a carbon fibre reinforced plastic composite material so that when the hydrostatic pressure/force seen from assignment 1 were applied, a maximum deformation of 0.1mm would occur. This is to ensure a good seal between the plate and the outer housing of the valve. The plate needed to use as little material as possible to reduce the cost of manufacture of the part.

To get an understanding of how to do this I wanted to research different Finite Element Analysis (FEA) techniques and learn a bit more about the optimization of them. This would help me understand how to implement them when it came to using software to design the material layup of the plate.

1)

Finite element analysis is a numerical method that uses mathematical techniques to solve engineering problems. The name comes from the technology itself, because it divides a problem into smaller parts called finite elements. The equations that are gained are combined into a larger equation system again to model the entire problem. This FEA involves complex mathematics and requires advanced computers and processors to perform this complex mathematical problem. FEA has developed tremendously in the past few decades. The advantage of this analysis is that the solution to our problem will be obtained in a cost-effective and faster way. For many industries such as automotive, medical and construction, FEA has become an important tool for calculating the stress caused in bridges for example.

In order to analyse any structure with load conditions, this FEA gives approximate results compared to any other practical technique. This method divides the structure into small elements and calculates each part of the structure, and then summarizes all the content that needs to be output.

Evolution of FEA over years:

It is said that FEA evolved from the work of Euler in the 16th century. But the actual work of this analysis can be found in the works of Schellbach [1851] and Courant [1943]. Later, this was developed by engineers in different fields such as civil engineering and aerospace. They began to use it to solve real-life problems in the mid-1950s. It led to the future evolution of the work of Zienkiewicz [1971] and Strang & Fix [1973]. Later, these resources were used to develop FEA software.

(Understanding FEA: These Are the Different Types of Finite Element Analysis, 2021)

Nastran

NASA Structural Analysis or (NASTRAN) is a finite element analysis program developed by Computer Science Corporation for NASA in the late 1960s. NASTRAN was released to the public by NASA's Office of Technology 1971. The MacNeal-Schwendler Corporation was one of the original developers of the publicly available NASTRAN code. NASTRAN source code is integrated in many different software packages that we see in today's engineering world.

(Editors, 2021)

About FEA Software:

Finite element analysis has brought about big changes in the technology used by industry to perform these operations. This is used in the engineering sector and has a wide range of applications in aerospace engineering.

Three types of analysis are:

Static:

Here static and linear structures like are analysed using this technique. Also, can be calculated when a static load is applied to a linear structure.

Dynamic:

Dynamic loads can be analysed using this technique. Calculating the impact are uses of this dynamic analysis.

Modal:

All objects vibrate at a certain frequency. You can't see the vibration, but everything is vibrating. When you add more objects or external factors (such as speed or wind), the vibration may increase or conflict. Modal analysis applies different forced vibrations to the structure. These destructive vibrations affect your structure in different ways. This analysis allows you to adjust the vibration during the design phase.

(Understanding FEA: These Are the Different Types of Finite Element Analysis, 2021)

Future Of FEA:

Over the years, finite element analysis is being carried out in a better way. In the future, it will be regarded as one of the important tools for performing such operations. It will give more accurate results through perfect calculations. It will be more effective for analysis in the medical field, which is the main requirement. In addition, it will be implemented more effectively in space research.

Optimisation techniques/software

There are many types of FEA software that is widely available and used in today's engineering community. They play a vital role in the engineering world in terms of simulation and analysis and these software/techniques include:

- Design Studio
- Sim scale
- Ansys
- Open foam
- Abaqus
- Auto desk

For the task that is set out, the technique and software that will be used to do so is design studio.

What is design studio

Design studio is a pre-processor and post-processor graphical interface for design-oriented structural analysis and optimisation software. Design Studio allows users to display finite element models and easily create design data. It allows users to display analysis and optimisation results.

Advantages of Design Studio

Easy and fast to create data

- Built-in paths can easily create goals and constraints
- Built-in trajectories make modelling the parameters easy

Easy and fast to post process

- Shows stress, displacement, thickness, or other contour plots and animations
- Animations and plots with deformed shapes
- Displays both deformation and contours at the same time.
- Densities for topology
- Frequency response plots

Can create presentations and reports

- Can create images (png format)
- Can create animations (gif format)
- Catalogues of views to create pictures with same orientations
- Back and forward buttons to return to previous view
- Colour control for printings

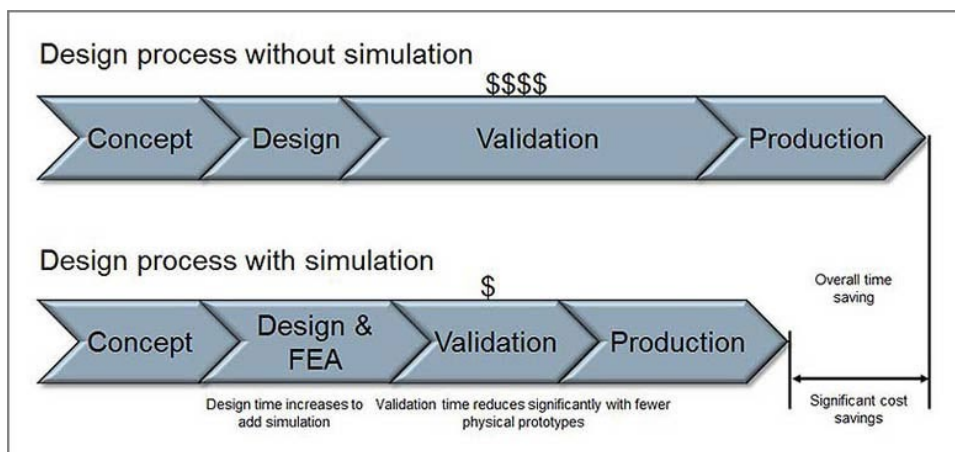
(Design Studio for GENESIS - Vanderplaats, 2021)

How FEA improves the design process

Using traditional methods to design the best quality products requires a lot of money and time to be invested in the process of prototyping, testing, iterating, and continuing until the desired result is achieved. FEA is the perfect tool to test and analyse numerous factors of product design, such as smaller footprint, less carbon emissions, ergonomic/user-friendly design, higher performance, and better manufacturability. By applying FEA tools, the design can be optimized. In addition, for better performance; it can also translate into improved manufacturability and help save a lot of operating costs.

By introducing FEA simulation in your design process, the design cycle will be longer, but the time to enter the design verification is reduced, so in general, in addition to saving money, engineers also save a lot of time. It is beneficial to introduce FEA early in the design process, but it can also be used in later stages, especially failure analysis. Failure analysis is performed when the product is on the market but must be recalled because it is not performed in accordance with the required standards.

(FEA and the design process, 2021)



(FEA and the design process,2021)

We have been tasked in this assignment to design the lay-up of the carbon fibre reinforced plastic composite material that when the hydrostatic pressure seen on the plate are applied that there is a maximum deflection of 0.1mm that can be seen. To make sure of a good seal between the plate and the outer housing it needs to use as little material as possible and reduce the cost of manufacturing of the part. With the aid of design studio this report will show and calculate the requirements mentioned above to meet the task asked.

First it is required to create a model of the plate and pin from the butterfly valve with the correct material properties to meet the demands of a unidirectional carbon fibre reinforced plastic composite material. The following properties are listed as follows:

- Young's Modulus in the longitudinal direction (E₁): 132 GPa
- Young's Modulus in the lateral direction (E₂): 10.8 GPa
- Poisson's Ratio (Nu₁₂): 0.24
- In plane Shear Modulus (G₁₂): 5.65 GPa
- Transverse Shear Modulus (1-Z plane)(G_{1z}): 5.65 GPa
- Transverse Shear Modulus (2-Z plane)(G_{2z}): 5.65 GPa
- Mass Density (Rho): 1540 kg/m³
- Allowable longitudinal stress in tension (Xt): 1513 MPa
- Allowable longitudinal stress in compression (Xc): 1000 MPa
- Allowable transverse stress in tension (Yt): 43.4 MPa
- Allowable transverse stress in compression (Yc): 43.4 MPa
- Allowable in plane shear stress (S): 43.4 MPa

(University Wales Trinity Saint David)

Baseline Composite

With the material properties set in design studio, the next thing to consider is the applied load or static load from hydrostatic pressure. This will impact the deflection upon the plate and the mass of the plate. The following equation is used to calculate the hydrostatic pressure:

$$\text{Pressure} = \text{Liquid height} \times \text{Density of liquid} \times \text{Gravity}$$

From Assignment number 1 we have values of the liquid height which is 5 meters and the density of liquid which is 800 kg/m³

$$\text{Pressure} = 5 \times 800 \times 9.81 = 39,240$$

$$\text{Force} = \text{Pressure} \times \text{Area}$$

However, Area is unknown so;

$$\text{Area} = \pi r^2$$

$$= \pi (0.097)^2$$

$$= 0.03 \text{ (2 d.p.)}$$

Now it is possible to calculate the force

$$\text{Force} = 39,240 \times 0.03 = 1177.2 \text{ N}$$

The final calculation to consider is that we need to determine the force per node required to enter into design studio. For this calculation we use:

Force / Number of nodes

$$1177.2 / 1438 = 0.82 \text{ N/node (2 d.p)}$$

Another thing to consider is the direction the vectors run. With the pin situated where it is the direction of the vectors are running vertically. With this information the structure of the vectors will be altered to run vertically in design studio.

The final thing to consider is the number of ply's and the orientation of those ply's within the composite material. Each orientation consists of 5 ply's that are 0.1 mm thick. These orientations are set at -90 degrees, -45 degrees, 0 degrees, 45 degrees and 90 degrees.

Layer 1, thickness 0.5mm, -90 degrees

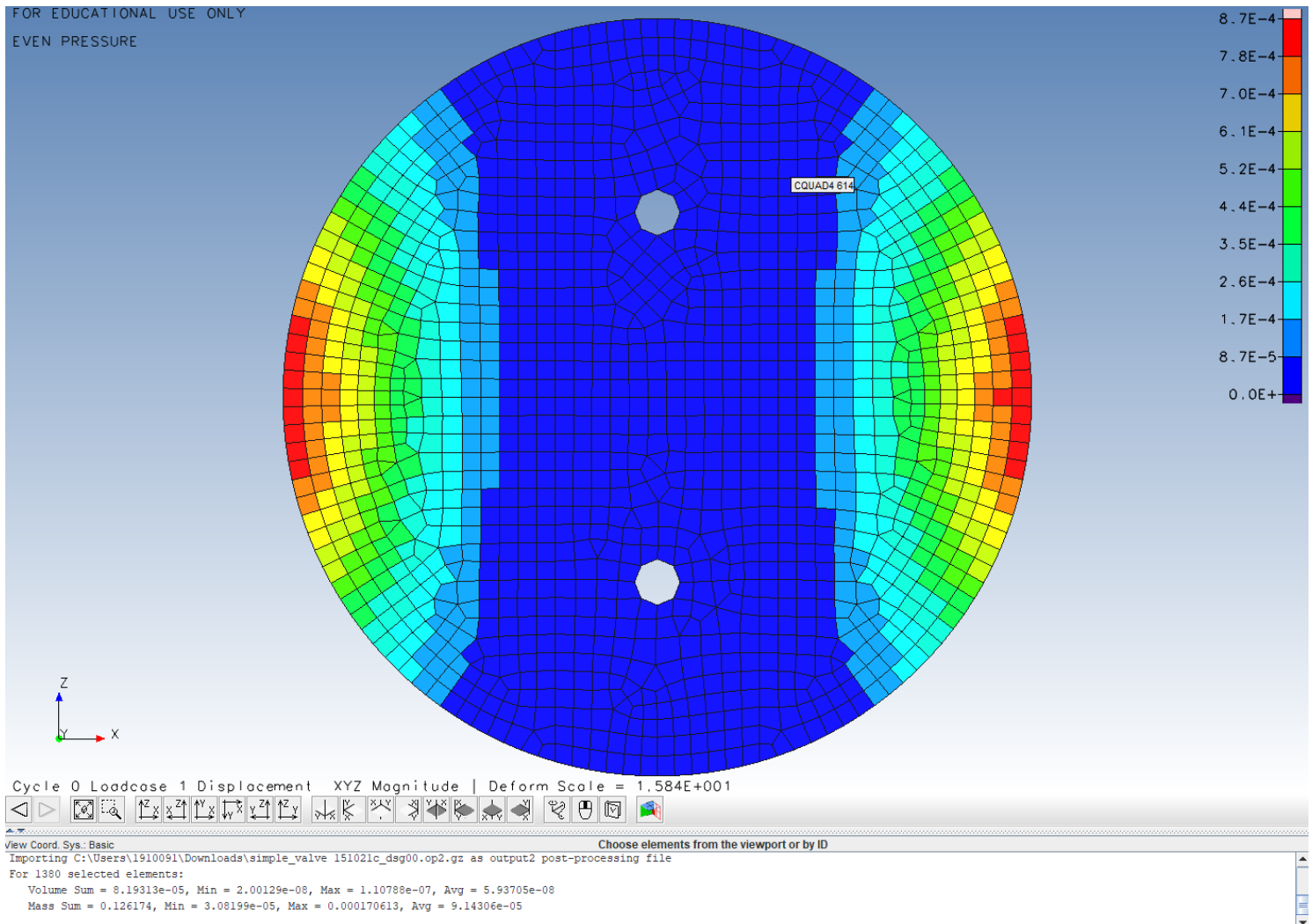
Layer 2, thickness 0.5mm, -45 degrees

Layer 3, thickness 0.5mm, 0 degrees

Layer 4, thickness 0.5mm, 45 degrees

Layer 5 thickness 0.5mm, 90 degrees

Our structure is made up of 25 ply's each 0.1mm thick. With all these factors we can create a baseline composite of the plate from the butterfly valve.



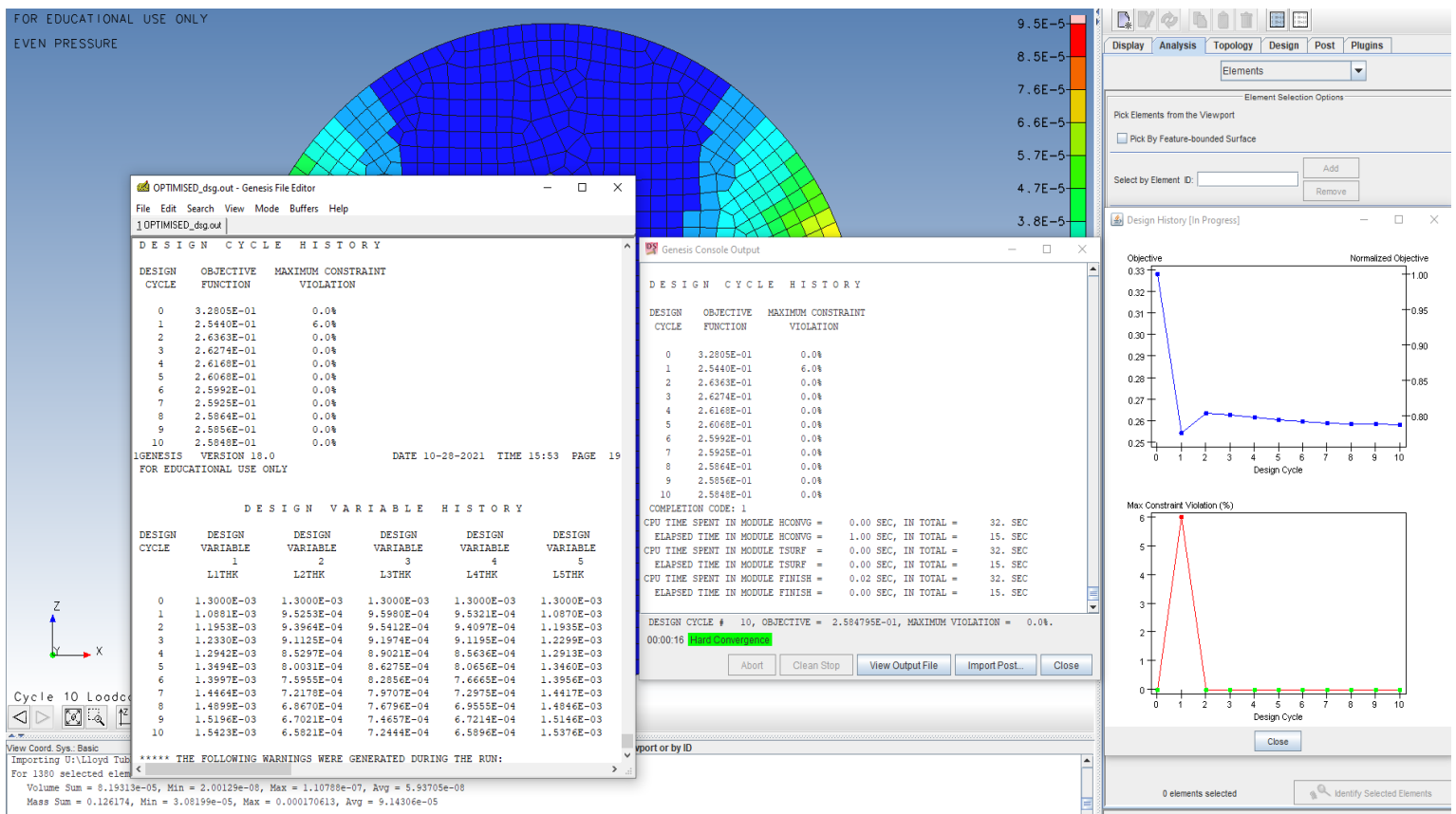
We can now assess the composite structure. As you can see the displacement is 0.87mm and the mass is 126 grams.

Run	Material thickness	Deflection	Mass	Load
Baseline Composite	2.5mm	0.87mm	126 grams	1177.2N

From the baseline composite the deflection/displacement of the plate is 0.87 mm. This does not meet the required deflection of 0.1mm that we require in order to have a good seal on the plate. However, the mass of the structure is significantly less than if it were to be made from steel.

Optimised composite

Now it is time to optimise the orientation of the layers and the thickness of these layers to achieve a deflection that is within the 0.1mm constraint.



Layer 1 = 1.542mm = 15.42 ply at -90 degrees

Layer 2 = 0.658mm = 6.58 ply at -45 degrees

Layer 3 = 0.724mm = 7.24 ply at 0 degrees

Layer 4 = 0.659mm = 6.59 ply at 45 degrees

Layer 5 = 1.534mm = 15.34 ply at 90 degrees

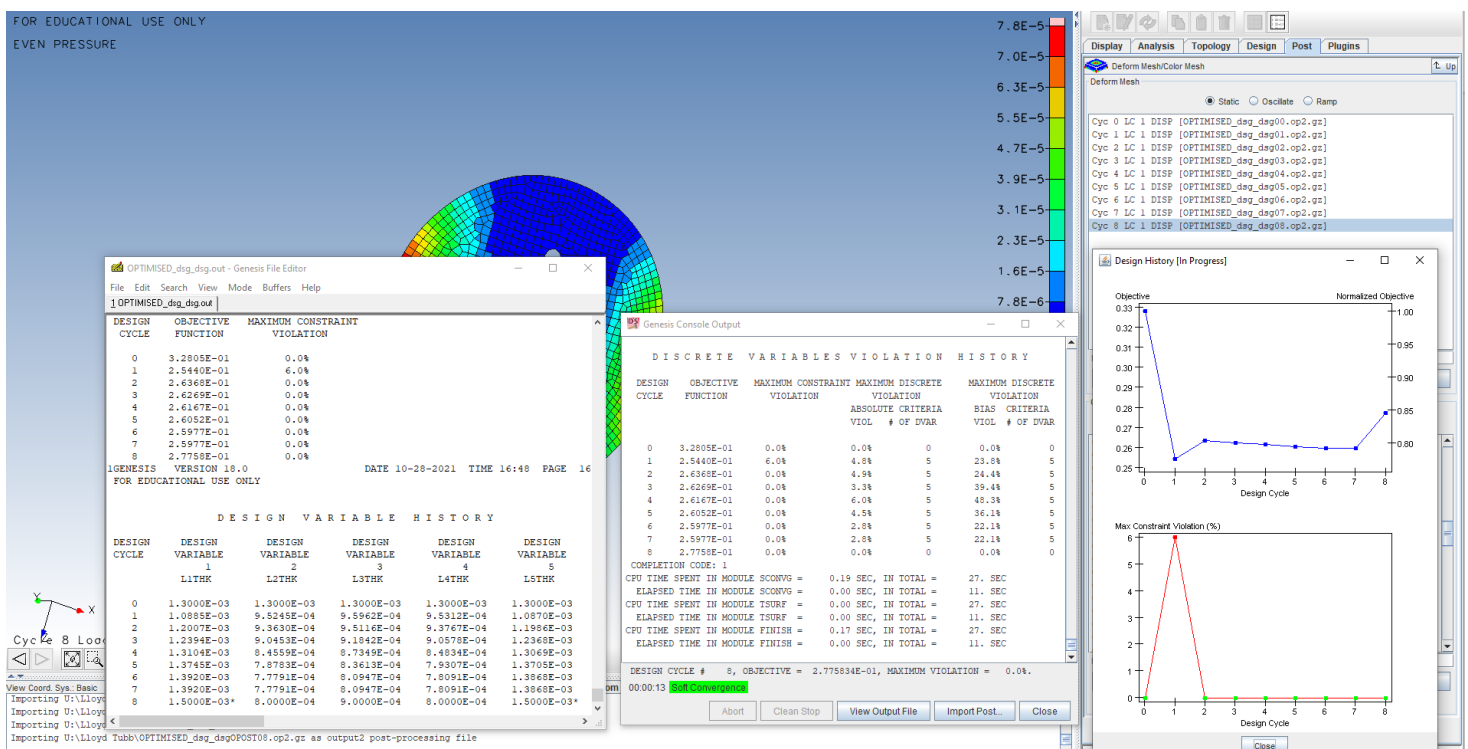
Total = 5.12mm = 51.17 ply

With the information above the results show that the deflection constraint had been met with a maximum deflection of 0.095mm. This meets the criteria also the mass and thickness of the plate has increased but the mass hasn't increased a significant amount that the cost of manufacturing the plate will be super expensive. The mass of the plate when optimised is 258.4 grams. However even though this plate is optimised the ply's generated by design studio would mean that the plate cannot be machinable. This is because it is impossible to have a ply for example .24 thick as you can only have whole increments and whole values to make the plate machinable. The next stage to do in design studio would be to make a model with discrete values so that the plate can be manufactured.

Run	Material thickness	Deflection	Mass	Load
Baseline Composite	2.5mm	0.87mm	126 grams	1177.2N
Optimised Composite	5.12mm	0.095mm	258.4 grams	1177.2N

Optimised composite (discrete)

Now it's time to optimise this plate structure with a set of discrete values entered in design studio so it can be manufactured.



Here is the final optimised discrete plate structure in design studio.

Layer 1 = 1.5mm	= 15 ply at -90 degrees
Layer 2 = 0.8mm	= 8 ply at -45 degrees
Layer 3 = 0.9mm	= 9 ply at 0 degrees
Layer 4 = 0.8mm	= 8 ply at 45 degrees
Layer 5 = 1.5mm	= 15 ply at 90 degrees
Total = 5.4mm	= 54 ply

With the information above the results show that ore deflection constraint has been met with a maximum deflection of 0.078mm. This meets the criteria also the mass and thickness of the plate has increased but the mass hasn't increased a significant amount that the cost of manufacturing the plate will be super expensive. The mass of the plate when optimised is 277.6 grams. This plate structure has now been fully optimised with ply's that are machinable so it can be manufactured. The mass is as light as possible to reduce manufacturing costs and the deflection is well within the 0.1mm limit that has been set. Also, each ply is 0.1mm thick with each orientation containing 5 plies.

Run	Material thickness	Deflection	Mass	Load
Baseline Composite	2.5mm	0.87mm	126 grams	1177.2N
Optimised Composite	5.12mm	0.095mm	258.4 grams	1177.2N
Optimised Composite(discrete)	5.4mm	0.078mm	277.6 grams	1177.2N

4+5)

As it is proven that the plate can be manufactured we can prepare the prototype of the butterfly valve, and then prepare for inspection. To check for leaks, the valve needs to be hydraulic tested. In the hydraulic test, the test medium is water or kerosene or other liquids. Hydraulic or pneumatic testing is to check the maximum pressure that the valve can withstand without failure. We must maintain the limits of pressure when doing tests, not work pressure. If the test is successfully completed and no leaks are found, the product passes successfully. If a leak is observed, we must redesign and remanufacture it again.

When designing the valve, all parameters such as the strength, bearing capacity, and pressure bearing capacity of the valve must also be checked. CFD simulations can be used to predict the value of stress applied to the valve. The coefficient of friction, the pressure valve and the stress acting on the valve can be predicted using simulation techniques.

Conclusion

In conclusion this report has summarised the tasks that were set out. It shows that a FEA analysis technique has been used with the aid of Design studio to create and optimise the plate of the butterfly valve from assignment 1. Using this technique has given us a deflection, mass and thickness the plate will be manufactured to so that it can withstand hydrostatic forces with a maximum deflection of 0.1mm. It also includes the history of FEA as well as an explanation for a prototype to be made and tested.

Bibliography

Meadows Analysis. 2021. *Understanding FEA: These Are the Different Types of Finite Element Analysis*. [online] Available at: <<https://meadowsanalysis.com/understanding-fea-these-are-the-different-types-of-finite-element-analysis/>> [Accessed 11 October 2021].

Editors, D., 2021. *What is NASTRAN? - Digital Engineering*. [online] Digital Engineering. Available at: <<https://www.digitalengineering247.com/glossary/nastran>> [Accessed 13 October 2021].

Vanderplaats. 2021. *Design Studio for GENESIS - Vanderplaats*. [online] Available at: <<http://www.vrand.com/products/design-studio-for-genesis/#1498230423359-4548517e-052ba23d-62fa>> [Accessed 16 October 2021].

Hitechcae.com. 2021. [online] Available at: <<https://www.hitechcae.com/blog/fea-and-its-impact-on-the-design-process-and-lifecycle-of-a-product/>> [Accessed 21 October 2021].

2021. [image] Available at: <<https://www.hitechcae.com/blog/fea-and-its-impact-on-the-design-process-and-lifecycle-of-a-product/>> [Accessed 22 October 2021].