

STRUCTURAL & FLUID ANALYSIS

ACMA6010

Samuel Thomas Melmoth (1809093)
1809093@student.uwtsd.ac.uk

Contents

Abstract.....	1
Introduction	1
Body	2
1	2
2	3
3	7
4	11
5	12
Conclusion	13
Works Cited.....	14

Abstract

FEA analysis is a crucial tool that can be used to improve the design process. To fault find and test designs before the prototyping phase. It is used in all areas of engineering design since its first use at NASA for the design of their space craft back in the 60's. It allows engineers to identify and eliminate waste from products and improve safety by predicting what a component will do under load.

Introduction

The purpose of this project is to demon straight and apply the skills that I have learned over the past weeks in this module. During this module we have learned about the origins of FEA and different types of analysis. We also learnt how to use FEA software such as Design studio. This application allowed me to evaluate a plate for a butterfly valve in two different materials. Steel and carbon fibber reinforced plastic. When the PCOMP is applied to the plate it allows us to refine our design with certain objectives such as minimum mass, volume, deflection etc. It does this by running multiple cycles and varying the number of plys in each orientation until it finds the best outcome, that satisfies the set objectives.

CFRP is a customisable material as it is made by layering multiple plys of carbon fibre sheets which have brilliant tensile strength, in different orientations. This is beneficial as it allows us to change the materials strength properties to suit our needs and applications such as applying forces in certain directions. Once the layers have been glued together, they are then cured in an oven under pressure and vacuum to form the desired solid shape.

After the computational work has been done there needs to be physical testing to reinforce and verify the results. This will be done under lab conditions in a small-scale controlled test.

The design of the component must meet the standards and best practices that are set for the area which the component will be used. In this case the oil and gas sector within the UK.

Body

1

FEA is a computer aided design software that analyses objects by placing nodes around the surface of the objects and makes hundreds of calculations to predict the behaviour between these nodes that allows the software to predict how the object will react under load and offers suggestions on how best to optimise the part. (Sergio Corbera, 2016) The example that we have been shown in class is how the software can be used to inform the user how best to layer up composite layers in order to create the strongest part possible to withstand forces in certain directions.

Three well known optimisation techniques are square, polynomial and discrete approximation. These names refer to how the elements are shaped; nodes are placed at each corner of the mesh around the object. These get incrementally more complex and therefore more accurate. These nodes tend to group around key points of an object such as boundaries and changes in geometry. (Ho-Le, 1988) The calculations needed to solve the differential equations between nodes were developed and formally written about in the 1850's by Schellbach, but FEA as we know it today was developed by the age of digital computers which allowed these complex calculations to be made economically. This was in the 1950's and 60's for NASA to do analysis for the space missions.

FEA allows designers to make more efficient, strong and safe products. It does this by reducing the waste from a design as well as evaluating problem areas that could cause issues. For example, bends in material that could cause weakness and failure after repeated use.

Prototyping is expensive and time consuming. By implementing FEA into the design process at multiple stages it allows engineers to reduce the prototyping stage of the development process. It does this by reducing the number of models that need to be made and tested as this can now be done computationally, negating material and labour costs significantly as computational testing can be run automatically without supervision over night for example, unlike real life testing. This does not completely replace prototype testing as the final design will still go through this key stage, but it will scale down on trial and error. (Swansea University, 2020)

In 1992 design studio was released alongside Genesis by Dr. Garret Vanderplaats. This software set out to improve the design process by allowing engineers to import CAD data and run pre and post processing features to simulate and therefore optimise their designs. This software has developed and been upgraded over the years but some of the developments moving forward are to enable the program to handle new manufacturing techniques such as additive manufacturing. In additive manufacturing overhanging material is common as the part is formed layer by layer on top of each other. Therefore, Genesis can now analyse how these overhangs will react as the material cools and forms. Continues improvement within the application is the display features and menu options. This application is limited in its layout and useability so increases in options like basic colours in the viewport which help distinguish between layers and materials. A filled-in shape domain display style that makes holes in the mesh appear more dominant and shaded feature styles to make parts look more realistic. These features help to improve the latest version. A technical improvement is the pre-processing enhancement to auto-fix elements that have been generated but are disconnected from other elements and do not form a grid. These can now be removed automatically before running a simulation. (VR&D, 2019)

2

A NASTRAN deck model was provided to us. This file included the dimensions of the disk, material set as steel, an element mesh made, and an example force applied. Before running the single analysis, the force was changed to 0.94N per element to represent the hydrostatic head on the valve plate. The model is also pre constrained down the centre of the Z axis as this is where the shaft would be located and cannot move.

```

simple_valve 151021c_dsg.out - Genesis File Editor
File Edit Search View Mode Buffers Help
1 simple_valve 151021c_dsg.out

THE MODEL FOR BOUNDARY CONDITION 1 CONTAINS
DISCONNECTED STRUCTURES.
NUMBER OF DISCONNECTED STRUCTURES = 2

1GENESIS VERSION 18.0 DATE 10-28-2021 TIME 18:10 PAGE
FOR EDUCATIONAL USE ONLY

*****
* DESIGN CYCLE 0 (ANALYSIS SUMMARY)*
*****

1GENESIS VERSION 18.0 DATE 10-28-2021 TIME 18:10 PAGE
FOR EDUCATIONAL USE ONLY

MASS / VOLUME SUMMARY

SYSTEM MASS 2.523485E-01
SYSTEM VOLUME 3.277254E-05
SYSTEM MASS/VOLUME 7.700000E+03

1GENESIS VERSION 18.0 DATE 10-28-2021 TIME 18:10 PAGE
FOR EDUCATIONAL USE ONLY

ENCLOSING BOX SUMMARY

LENGTH X LENGTH Y LENGTH Z AREA XY AREA YZ AREA XZ VOLUME XYZ
2.049E-01 1.256E-17 2.050E-01 2.573E-18 2.574E-18 4.201E-02 5.275E-19

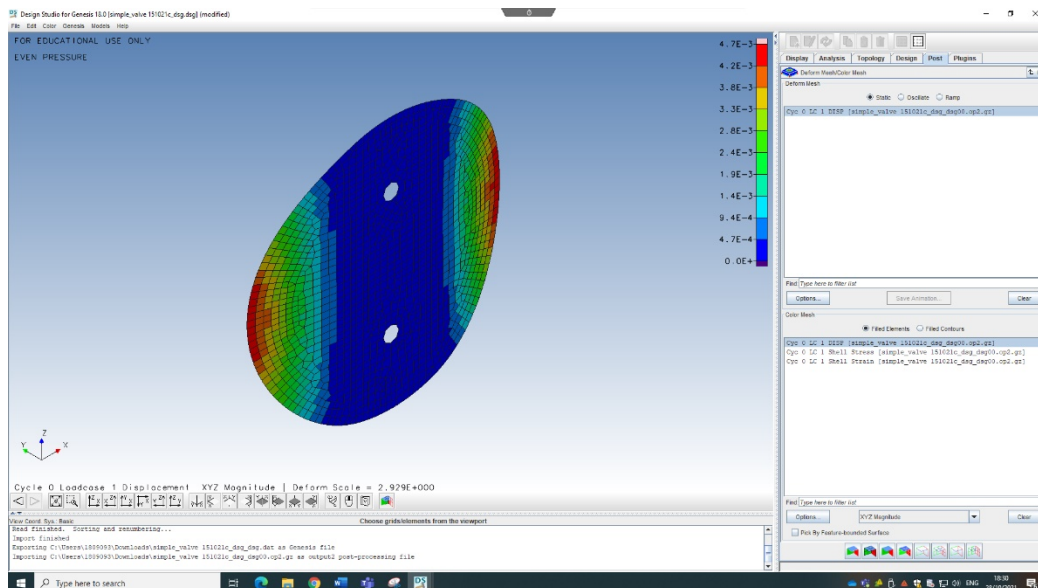
1GENESIS VERSION 18.0 DATE 10-28-2021 TIME 18:10 PAGE
FOR EDUCATIONAL USE ONLY

SOLUTION RESIDUALS

LOADCASE 1 ; RESIDUAL : 1.515486E-14 ; STRAIN ENERGY : 5.204371E-01

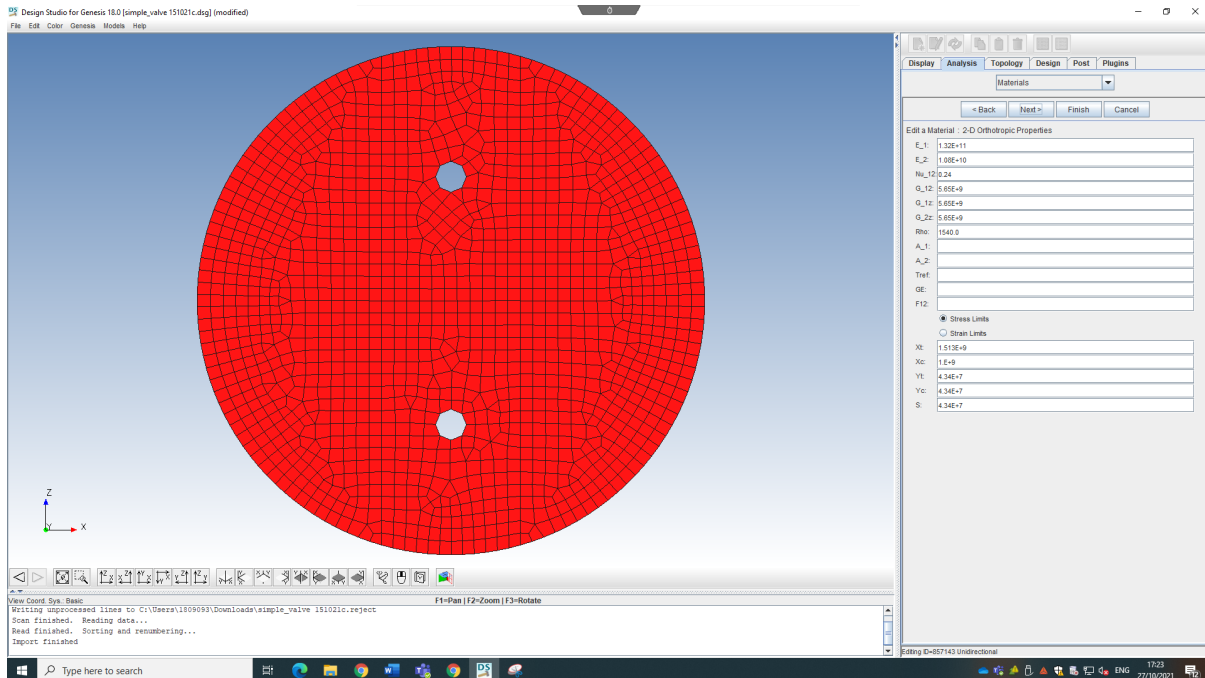
1GENESIS VERSION 18.0 DATE 10-28-2021 TIME 18:10 PAGE
FOR EDUCATIONAL USE ONLY

```

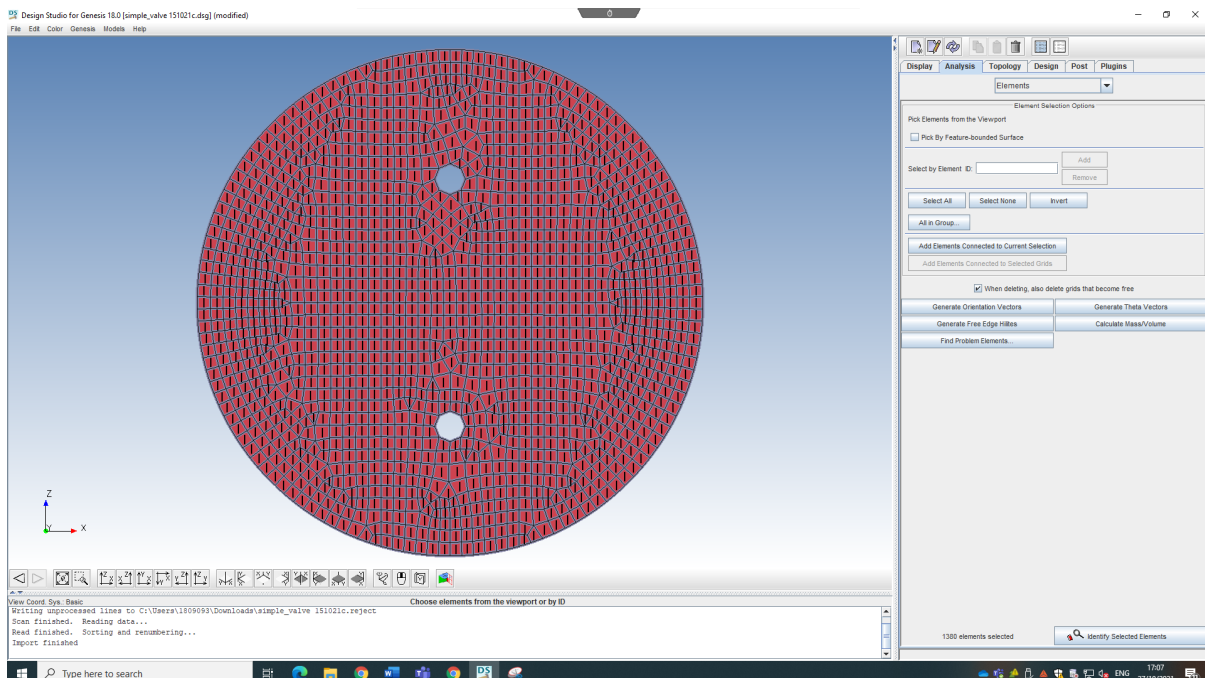


After running a single analysis, the mass of the steel baseline plate is 0.252kg and it deformed by 4.7mm. This is out of tolerance for the valve.

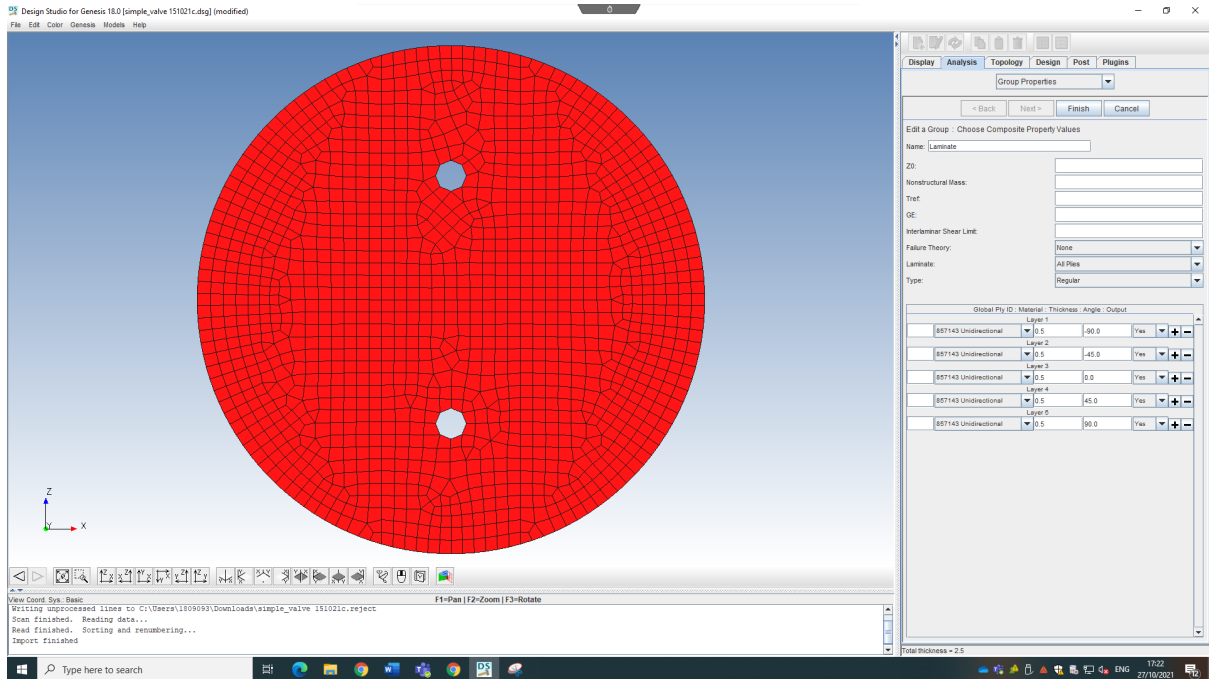
To create the model for the composite plate from this baseline the parameters were changed to simulate a composite disk made from 25 plies of CFRP in 5 layers in different orientations.



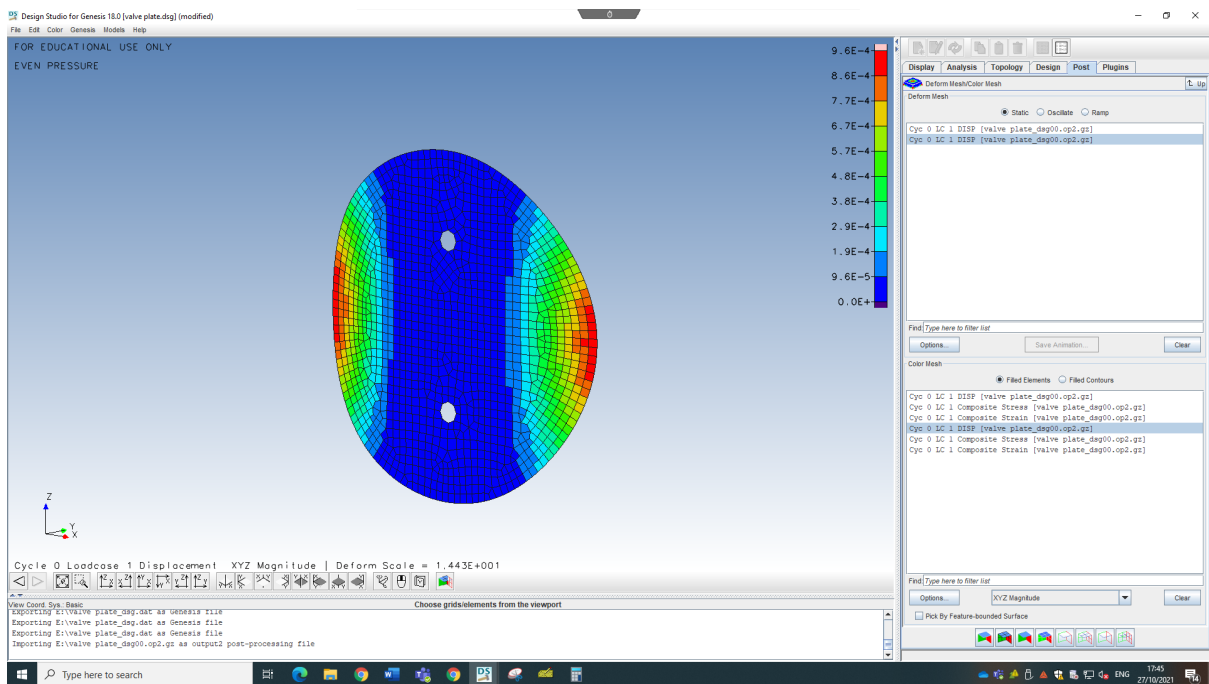
The material properties have to be changed to those of the CFRP plies specified. This is so the material can be save and aplied to the layers later on.

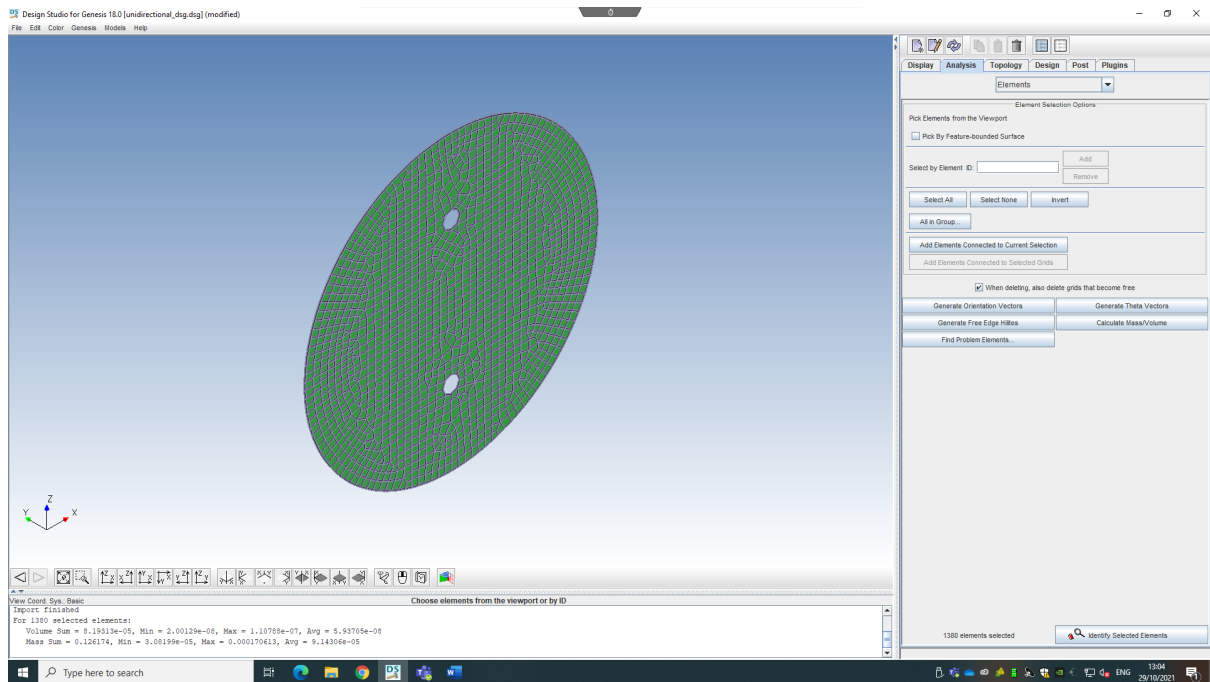


Next the element axis vectors must all be aligned with the +Z axis.



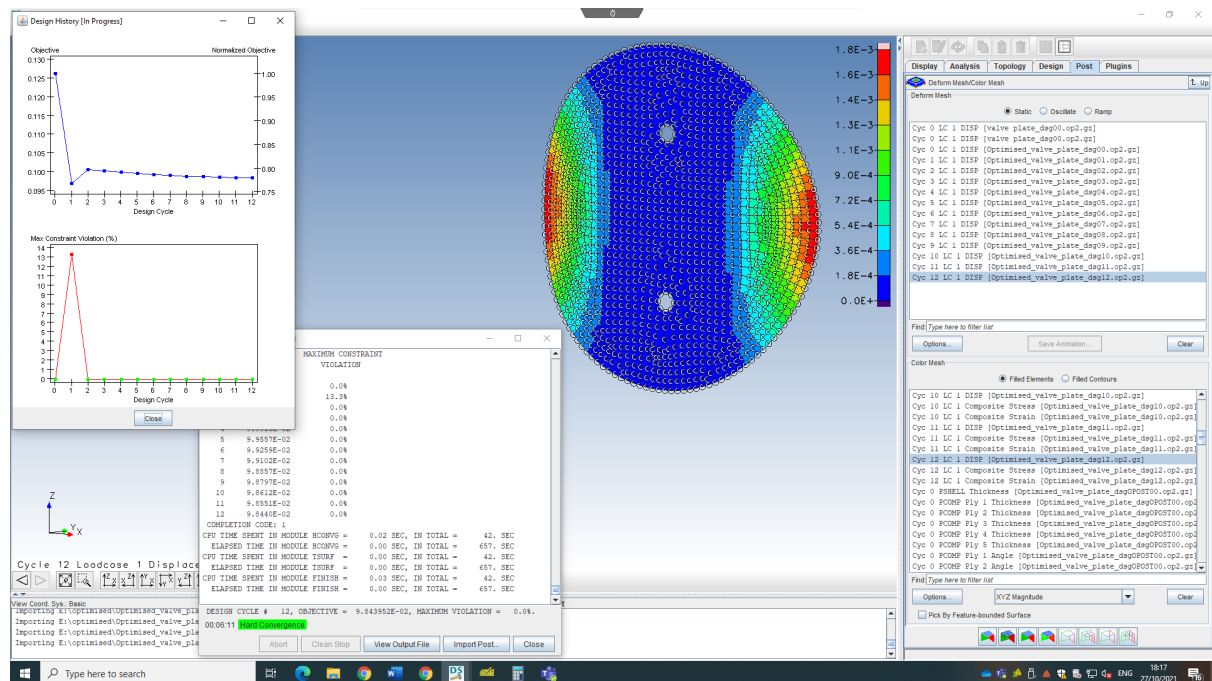
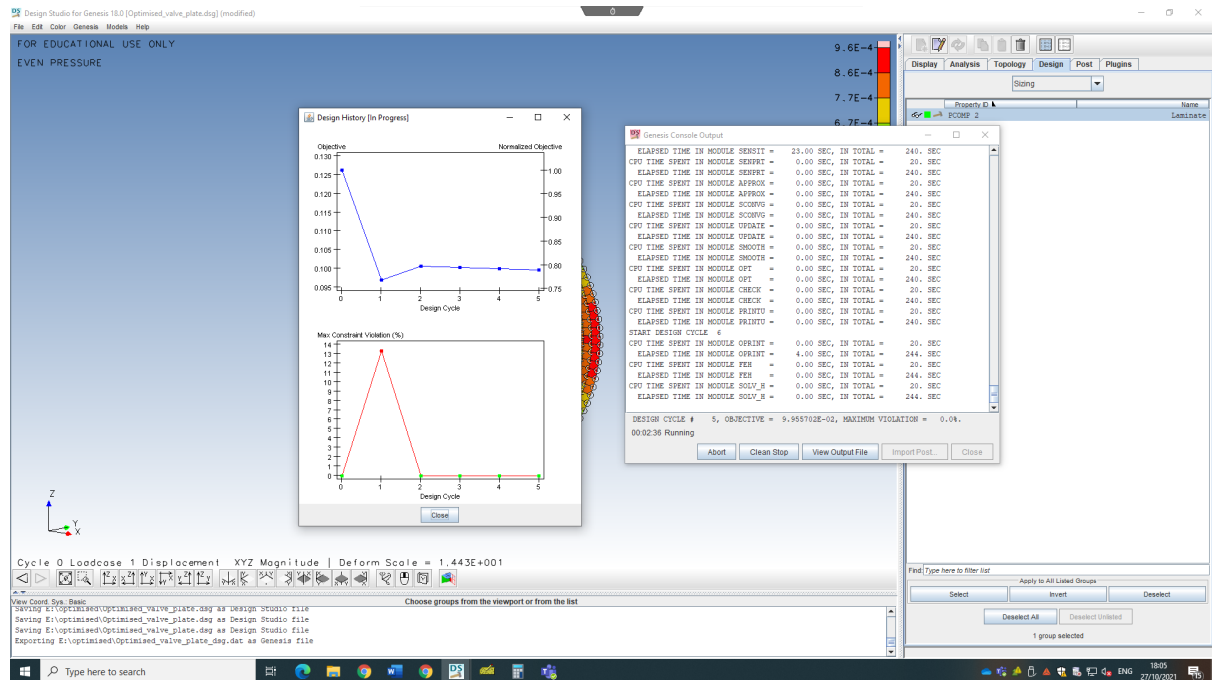
The model can then be changed to a PCOMP/PCOMPG. This allows the model to be broken down into different layers. The layer thickness is then specified as 0.5mm (5 ply) each and the layers are given a grain direction.





Single analysis of the unoptimized plate with 5 plies of material in each orientation and at weight 126g shows that the displacement is out of range at 9.6mm. This shows that although the CFRP is lighter it is also more flexible and cannot withstand the pressure exerted on it. And therefore, offers no advantage. Due to this, design studio is required to optimise the number of plies in each layer so that the material is stiffer in a specific direction to resist the forces better.

In order to optimise the plate, design studio must know what its objectives are. The main objective is to make the plate as light as possible whilst remaining within the deflection tolerance under the 5m hydrostatic head.



```

E:\optimised\Optimised_valve_plate_dsg.HIS - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
Optimised_valve_plate_dsg.HIS
46 DESIGN VARIABLE # 3 = 2.908977E-04 L3THK
47 DESIGN VARIABLE # 4 = 2.789054E-04 L4THK
48 DESIGN VARIABLE # 5 = 5.581301E-04 L5THK
49 DESIGN CYCLE # 8, OBJECTIVE = 9.885661E-02, MAXIMUM VIOLATION = 0.0%.
50 DESIGN VARIABLE # 1 = 5.702319E-04 L1THK
51 DESIGN VARIABLE # 2 = 2.679071E-04 L2THK
52 DESIGN VARIABLE # 3 = 2.809926E-04 L3THK
53 DESIGN VARIABLE # 4 = 2.682800E-04 L4THK
54 DESIGN VARIABLE # 5 = 5.713200E-04 L5THK
55 DESIGN CYCLE # 9, OBJECTIVE = 9.879653E-02, MAXIMUM VIOLATION = 0.0%.
56 DESIGN VARIABLE # 1 = 5.816836E-04 L1THK
57 DESIGN VARIABLE # 2 = 2.594968E-04 L2THK
58 DESIGN VARIABLE # 3 = 2.735876E-04 L3THK
59 DESIGN VARIABLE # 4 = 2.598946E-04 L4THK
60 DESIGN VARIABLE # 5 = 5.828785E-04 L5THK
61 DESIGN CYCLE # 10, OBJECTIVE = 9.861227E-02, MAXIMUM VIOLATION = 0.0%.
62 DESIGN VARIABLE # 1 = 5.918627E-04 L1THK
63 DESIGN VARIABLE # 2 = 2.515442E-04 L2THK
64 DESIGN VARIABLE # 3 = 2.654289E-04 L3THK
65 DESIGN VARIABLE # 4 = 2.519267E-04 L4THK
66 DESIGN VARIABLE # 5 = 5.931278E-04 L5THK
67 DESIGN CYCLE # 11, OBJECTIVE = 9.855098E-02, MAXIMUM VIOLATION = 0.0%.
68 DESIGN VARIABLE # 1 = 6.059398E-04 L1THK
69 DESIGN VARIABLE # 2 = 2.434623E-04 L2THK
70 DESIGN VARIABLE # 3 = 2.519764E-04 L3THK
71 DESIGN VARIABLE # 4 = 2.438706E-04 L4THK
72 DESIGN VARIABLE # 5 = 6.074266E-04 L5THK
73 DESIGN CYCLE # 12, OBJECTIVE = 9.843952E-02, MAXIMUM VIOLATION = 0.0%.
74 DESIGN VARIABLE # 1 = 6.132013E-04 L1THK
75 DESIGN VARIABLE # 2 = 2.373576E-04 L2THK
76 DESIGN VARIABLE # 3 = 2.473681E-04 L3THK
77 DESIGN VARIABLE # 4 = 2.378030E-04 L4THK
78 DESIGN VARIABLE # 5 = 6.147374E-04 L5THK
79 COMPLETION CODE: 1 1496 0
80
Normal text file length: 4,756 lines: 80 Ln: 75 Col: 41 Sel: 9 | 1 Windows (CR LF) UTF-8 INS

```

In order to allow the software more time to solve the model, the maximum number of cycles was increased from 10 to 30. This increases the likelihood of a hard convergence. There has to be a maximum number of cycles otherwise the program could run endlessly if there was no possible solution.

The optimisation process went through 12 cycles before reaching a hard convergence. The optimised composite with independent variables has a maximum deformation of 1.8mm this is below the target threshold. But due to the minimum ply thickness not being taken into account, this means that the model isn't manufacturable. For instance layer 2 is 2.373576mm thick. The lowest weight achieved was 98g.

DESIGN VARIABLE # 1 = 6.132013E-04 L1THK

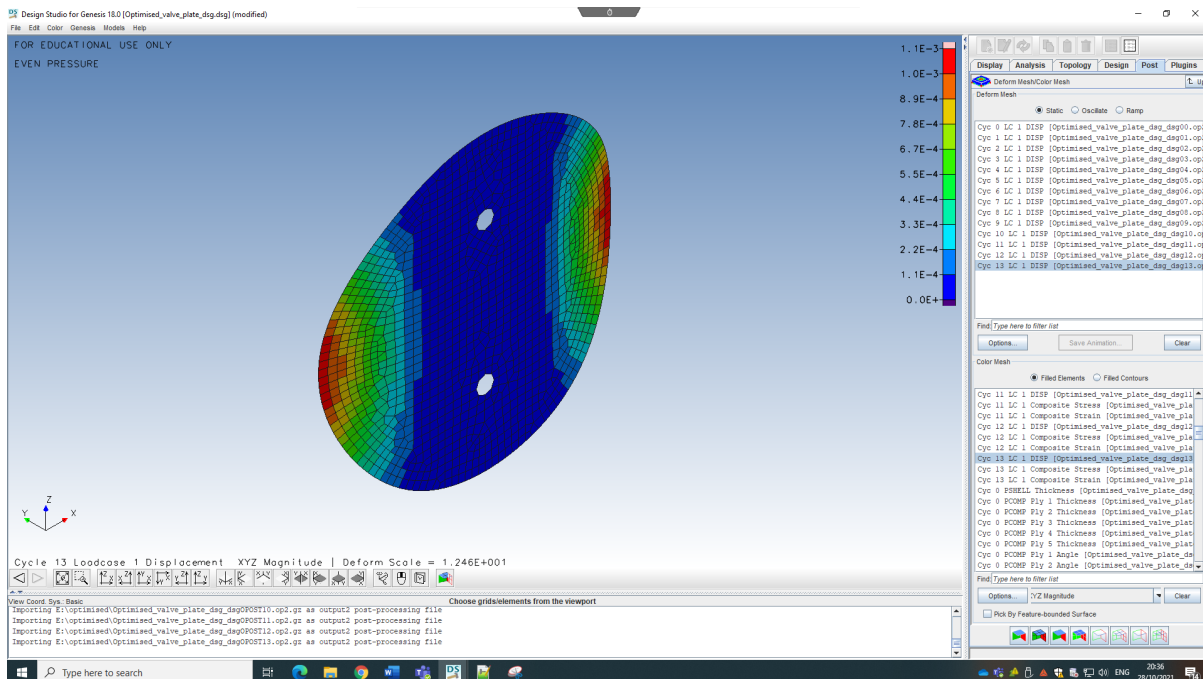
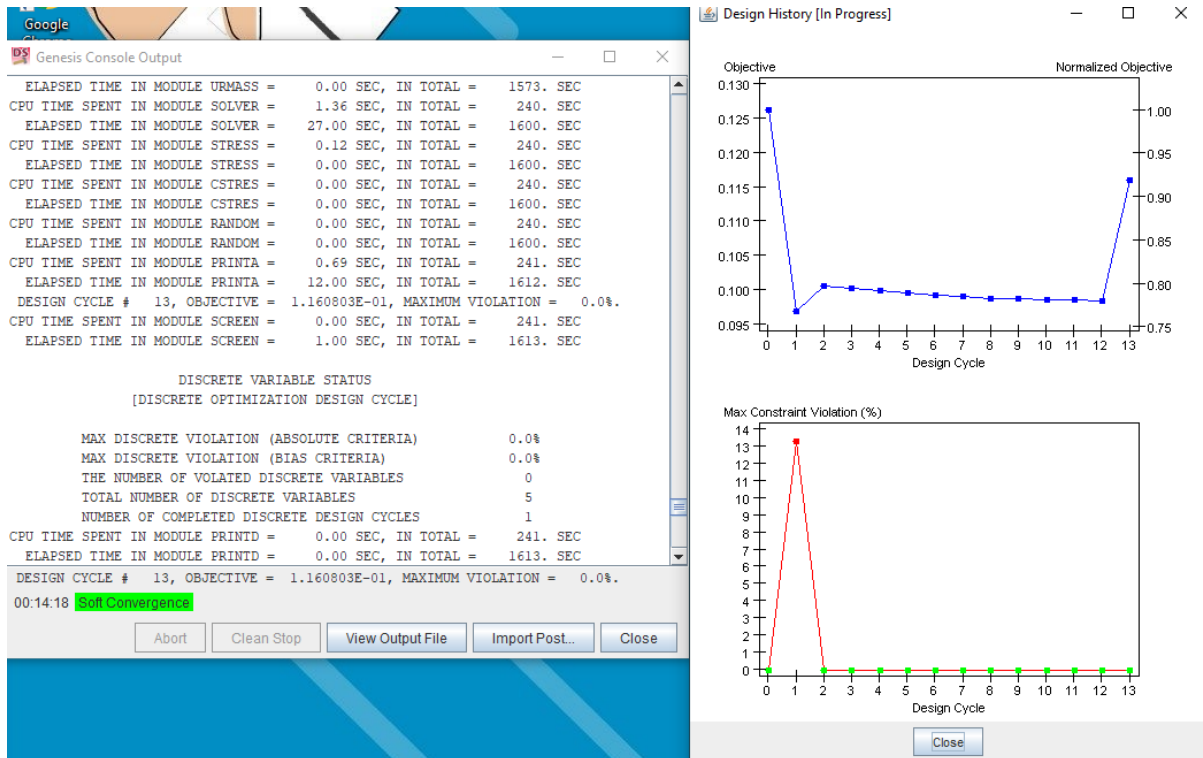
DESIGN VARIABLE # 2 = 2.373576E-04 L2THK

DESIGN VARIABLE # 3 = 2.473681E-04 L3THK

DESIGN VARIABLE # 4 = 2.378030E-04 L4THK

DESIGN VARIABLE # 5 = 6.147374E-04 L5THK

To fix this the optimization must be run again, but with discrete variables instead of independent variables. This is so design studio uses thicknesses that are the same as the carbon fiber cloth used in the ply. And is manufacturable.



The discrete variable simulation ended in a soft convergence after 13 cycles with 0% violation and a deflection of 1.1mm. After running the optimization with discrete variables, the simulation returned the mass as 116g and the following layer structure:

DESIGN CYCLE # 13, OBJECTIVE = 1.160803E-01, MAXIMUM VIOLATION = 0.0%.

DESIGN VARIABLE # 1 = 7.000000E-04 L1THK -90 degrees = 0.7mm = 7 plys

DESIGN VARIABLE # 2 = 3.000000E-04 L2THK -45 degrees = 0.3mm = 3 plys

DESIGN VARIABLE # 3 = 3.000000E-04 L3THK 0 degrees = 0.3mm = 3 plys

DESIGN VARIABLE # 4 = 3.000000E-04 L4THK 45 degrees = 0.3mm = 3 plys

DESIGN VARIABLE # 5 = 7.000000E-04 L5THK 90 degrees = 0.7mm = 7 plys

This final simulation shows that this layering isn't as efficient as the previous. But due to real world constraints it is the best possible. It is also has a far better stiffness to weight ratio compared to the steel plate.

In order to verify that the computational figures are accurate it is necessary to test the component in a lab. The figures can then be compared to each other.

When conducting experiments, the first consideration should always be health and safety. In order to test safely adequate PPE such as safety glasses, gloves and lab coat are required. A Risk assessment should also be filled in.

In order to simulate the pressure on the valve plate by the hydrostatic head, a compressor can be used to force a smaller amount of liquid through the housing, by doing this it allows the test to be done in a lab and without the large tank. It also allows the pressure to be varied easily. The reason for the liquid is because the valve needs to be watertight not airtight and replicates the circumstances of the valve more realistically. The equipment needed to conduct this experiment are:

- Air compressor, to simulate the hydrostatic head by compressing the fluid against the valve.
- Pig, to separate the air from the liquid so they do not mix under compression.
- Prototype valve.
- Bracket to hold the valve in situ.
- Strain gauges, to record the strain on the CFRP and allow for calculations on the deformation.

As the direction of the of the principal stress is known then using a more expensive Rosette gauge is not necessary and one of the other forms of gauge can be used, such as a linear strain gauge. I was able to come to this conclusion by answering questions in a workflow supplied by HBM a global sensor and measurement company with 65 years of experience in measurement technology, and one of the first companies globally to be certified to ISO9001. (HBM, 2021) This allowed me to select a linear strain gauge as the best option for this application. The specifications of the gauge needed was a type DY strain gauge with a 6mm measuring grid. The type DY gauge consists of two parallel measuring grids. There would need to be a gauge either side of the central axil to measure the two areas of flexion.

The first step is to apply the strain gauges to the underside of the valve plate. To do this the surface must be clean so that the adhesive can adhere firmly and measure an accurate reading.

Next fill one side of the valve with petroleum and place the pig on top

Next thread an end cap with a coupling for an air hose to seal the one side.

Once the valve is secured connect the compressor and increase the pressure to an equivalent of $39,240N/m^2$

Hold the pressure for 24 hours in order to capture any fatigue after an extended period and log the data on a computer.

A second test should be conducted for the same duration but with the pressure cycled so that the material will undergo cyclic loading to analyse any effect this might have.

After the test has concluded the highest strain value from both tests will be taken to calculate the maximum deformation.

The maximum allowable deformation of the plate is 1.9mm. Measuring this will be done by applying strain gauges to the underside of the valve gate. By measuring the strain on the material this will allow the equation $\varepsilon = \frac{\Delta L}{L_0}$ to be used to calculate the deformation on the edges of the plate.

This test must be conducted as the simulations carried out on FEA aren't 100% accurate due to assumptions that have to be made in order to allow the calculations to take place. Such as the density of the mesh and an even distribution of force across the surface of the plate.

5

ISO 27509:2020 is an international standard that is used within the UK to detail the requirements for flanged connections that are used within the oil and gas sector which our valve under the petroleum tank would fall within. It controls factors such as:

- Flange dimensions and material requirements
- Seal ring dimensions and material requirements
- Bolting dimensions and material requirements
- Requirements to tolerances and surface finish
- Requirements for designations and marking of finished products.

(ISO, 2020)

The butterfly valve is used more in the oil and gas industry as it is typically lighter than gate valves, this is desirable for use on ships and on offshore oil platforms. It also closes faster, making it better for precise measurements and emergency applications where a fast action is required such as Atex rated systems to contain explosions within pipelines.

ISO 5211 states that the butterfly valves must be able to operate from temperatures of -40°C to 210°C (Albion, 2020) (bsi, 2012)

Due to Brexit the UK no longer requires a CE mark on its products the new marking is UKCA which stands for UK Conformity Assessed. Change from CE to UKCA (ISO, 2020)

API standards are an American standard that is used internationally. API 609 stipulates what pressures valves used for petrochemical use must withstand, ASME B16.34 is the standard for temperature rating. API 598 is the standard for testing, ASME B16.5 and ASME B16.47 covers the flanged ends and connections and ASME B16.25 specifications cover the dimensions and the tolerances of butterfly valves for the petrochemical industry and piping applications.

Due to the valves proximity and use with flammable materials such as petroleum it should be Atex rated as not to cause ignition of the flammable material.

Conclusion

To conclude

After the simulations were complete steel and CFRP have very similar stiffness to weight ratios when the CFRP isn't optimised and equal numbers of plies are put in each direction. But when the optimisation is done. The carbon fibres advantage of tensile strength is utilised more efficiently by aligning the fibres with the direction of the force that needs to be resisted. This allows less material to be used and a reduction in weight.

Completing this assignment has given me practical experience in using Design Studio, and a much more in-depth understanding and appreciation for the features that it can bring to improve the design process. I have also gained knowledge from researching the many different standards that cover the use of butterfly valves in the petrochemical industry.

Works Cited

Albion, 2020. *tglynnes*. [Online]

Available at: <http://www.tglynnes.co.uk/downloads/tgl-coshh-albion-butterfly-valves.pdf>

[Accessed 28 october 2021].

bsi, 2012. *BS EN ISO 27509:2012 Petroleum and natural gas industries - compact flanged connections with IX seal ring*. London: British Standards Institution.

English, T., 2019. *Interesting Engineering*. [Online]

Available at: <https://interestingengineering.com/what-is-finite-element-analysis-and-how-does-it-work>

[Accessed 27 10 2021].

Harish, A., 2020. *Simscale*. [Online]

Available at: <https://www.simscale.com/blog/2016/10/what-is-finite-element-method/>

[Accessed 27 10 2021].

HBM, 2021. *HBM/About*. [Online]

Available at: <https://www.hbm.com/en/7164/how-to-find-the-right-strain-gauge/>

Ho-Le, K., 1988. Finite element mesh generation methods: a review and classification. *Computer-Aided Design*, 20(1), pp. 27-38.

ISO, 2020. *Petroleum and natural gas industries — Compact flanged connections with IX seal ring*. 2 ed. s.l.:ISO.

Sergio Corbera, J. L. O. J. A. L., 2016. Multi-objective global optimization of a butterfly valve using genetic algorithms. *ISA Transactions*, 63(1), pp. 401-412.

Swansea University, 2020. *Swansea University*. [Online]

Available at: <https://www.swansea.ac.uk/engineering/research/impact/benefits-of-finite-element-analysis/>

[Accessed 28 10 2021].

VR&D, 2019. *VR&D*. [Online]

Available at: http://www.vrand.com/wp-content/uploads/2019/06/DSG_18_newFeatures.pdf

[Accessed 28 October 2021].