

Course Lecturer: Kelvin Lake

Structural & Fluid Analysis ACMA6010

Designing a Butterfly Valve:
Structural Design

Kaine Angus (1603343)

Table of Contents

<i>Introduction</i>	<i>2</i>
<i>Task One</i>	<i>3</i>
<i>Task Two.....</i>	<i>6</i>
<i>Task Three.....</i>	<i>10</i>
<i>Task Four.....</i>	<i>12</i>
<i>Task Five</i>	<i>13</i>
<i>Conclusion:.....</i>	<i>13</i>
<i>References.....</i>	<i>14</i>

Introduction

In this report a detailed description of the skills and knowledge I have picked up over the last few months in this module. Over the period of this module, we have learned the different types of analysis of Finite Element analysis (FEA) and also the how to critically use a FEA software Design studio. From this software it allowed me to analysis a butterfly valve plate to ensure that the product does not leak past the valve when it is closed but use as little material as possible, it also allowed me to evaluate the valve in two different materials which are steel and unidirectional (UD) carbon fibre reinforced plastic (CFRP). It was proposed that the plate centre section is to be supported by a pin which is used to rotate the plate, which the edges are unsupported. When the layered composite element property (PCOMP) is applied to the plate it allows the user to select certain objectives such as deflection, minimum mass, volume etc. this is carried out by running multiple cycles with various different number of layers (plys) in a set orientation until the most effective outcome is achieved with the desired outcomes set.

Carbon fibre reinforced polymers are composite materials which ultimately rely on the carbon fibre to give the stiffness and strength whereas the polymer gives a cohesive matrix to protect and hold the fibres together and gives some of the toughness. The carbon fibres give highly directional properties much different than metals and mostly used in the automotive industry for there applications. They are often engineered to achieve mass reductions that are not achievable by metals, since as mentioned previously their properties and performance can be customised. (Direct, 2021)

Task One

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

The Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM). Engineers use FEA software to reduce the number of physical prototypes and experiments and optimise components in their design phase to develop better products, faster while saving on expenses.

Differential equations not only describe natural phenomena but also physical phenomena encountered in engineering mechanics. These partial differential equations (PDEs) are complicated equations that need to be solved in order to compute relevant quantities of a structure like stresses and strains, etc in order to estimate the structural behaviour under a given load. It is important to know that FEA only gives an approximate solution to the problem and is a numerical approach to get the real result of these partial differential equations. (Belytschko, 2007) FEA is a numerical method used for the prediction of how a part or assembly behaves under given conditions. It is used as the basis for modern simulation software and helps engineers to find weak spots, areas of tension, etc. in their designs. The results of a simulation-based on the FEA method are usually depicted via a colour scale that shows, for example, the pressure distribution over the object. (SimScale, 2021)

FEA can be said to have its origin in the work of Euler, as early as the 16th century. However, the earliest mathematical papers on Finite Element Analysis can be found in the works of Schellbach in the 1850's and Courant in the 1940's. FEA was independently developed by engineers in different industries to address structural mechanics problems related to aerospace and civil engineering. The development for real-life applications started around the mid-1950s as papers by Turner, Clough, Martin & Topp in the 1950's, Argyris in the 19570's, and Babuska & Aziz in the 1970's show. The books by Zienkiewicz in the 1970's and Strang & Fix in the 1970's also laid the foundations for future developments in FEA software. (Bi, 2017)

The mathematical principles of FEA are also useful in other areas, such as computational fluid dynamics or CFD. The key difference is that FEA focuses on structural analysis and CFD on fluid dynamics. FEA algorithms are integrated into simulation software like Autodesk Inventor Nastran or ANSYS's software. These programs are usually integrated into computer-aided design (CAD) software, making it much easier for engineers to go from design to running complex structural analysis. To run an FEA simulation, a mesh is first generated, containing millions of small elements that make up the overall shape. This is a way of transcribing a 3D object into a series of mathematical points that can then be analysed. The density of this mesh can be altered based upon how complex or simple a simulation is needed. Calculations are run for every single element or point of the mesh and then combined to make up the overall final result for the structure.

Since the calculations are done on a mesh, rather than the entirety of a physical object, it means that some interpolation needs to occur between the points. These approximations are usually within the bounds of what's needed. The points of the mesh where the data is known mathematically are referred to as nodal points and tend to be grouped around boundaries or other areas of change in an object's design.

FEA can also be applied to thermal analysis within a material or shape. Utilising FEA, an approximation can be made for these points using different modes of accuracy. There's a square approximation, a polynomial approximation, and a discrete approximation. Each of these techniques increases in accuracy and complexity. (Bi, 2017)

Static FEA:

A linear static FEA is performed when the forces applied to a part or assembly do not change over time. A static FEA is the simplest form of FEA, and is performed exclusively on linear elastic materials, where small displacements are anticipated.

Thermal Analysis:

Thermal analysis simulates heat transfer between solid bodies. A thermal analysis can be a useful standalone tool, for example when analysing the design of a heat-sink. It can also be used in combination with a static analysis to conduct thermal stress analyses on parts or assemblies.

Dynamic FEA:

A dynamic analysis is performed when the forces applied to a part or assembly change over time. A dynamic analysis can be performed on either linear elastic or non-linear elastic materials. Common examples of a dynamic study include drop tests and vibration analysis.

CFD – Computational Fluid Dynamics:

CFD simulates how gasses or liquids flow in, or around, a part or assembly. It is frequently used to analyse air flow through a design and optimise external forms to reduce drag or to provide directional force to the structure such as to increase downforce on high-speed vehicles.

A frequent issue with FEA is that the original load cases and loading scenarios are incorrect so they don't truly represent the actual load onto a component or system. This risk can be reduced by taking the time to truly consider what is happening and using Simulation to identify peak loads and understand loading profiles.

One of the biggest issues we find with FEA being outsourced, is that the iterative nature of design optimisation can be broken or come too late. It's typical for analysis to identify issues in a design, which subsequently require further investigation and design optimisation. Therefore, it is always an advantage to introduce FEA, as early as possible, into the design process. This means problems are identified more quickly and changes can be made to the basic design early on. Doing this well before extensive engineering is undertaken will significantly reduce time and therefore costs or delays.

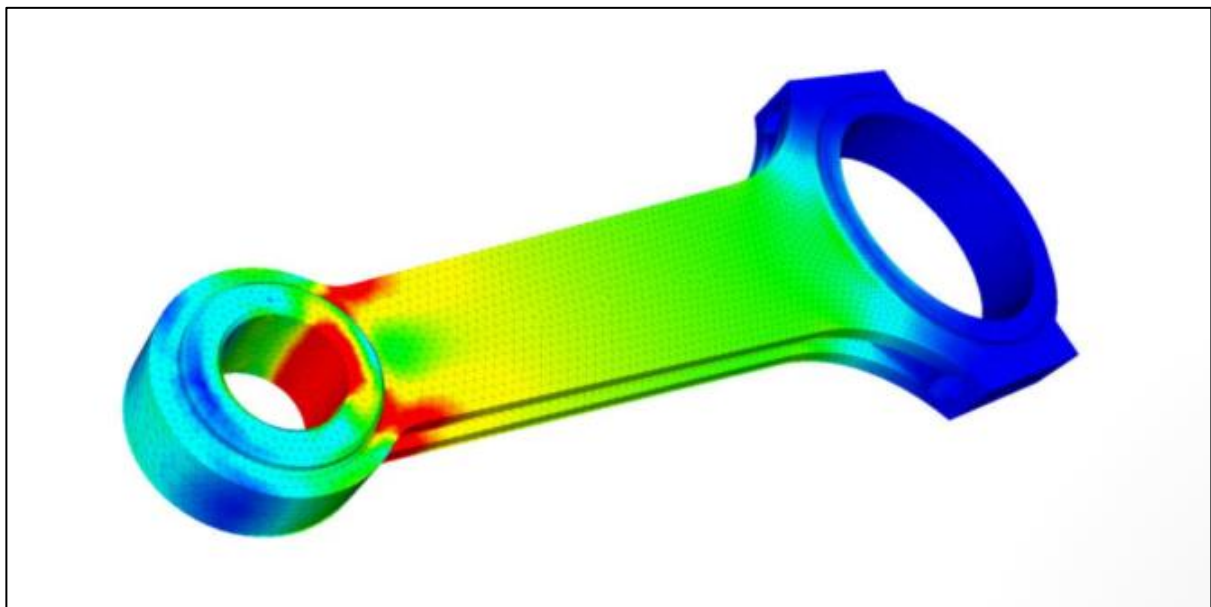
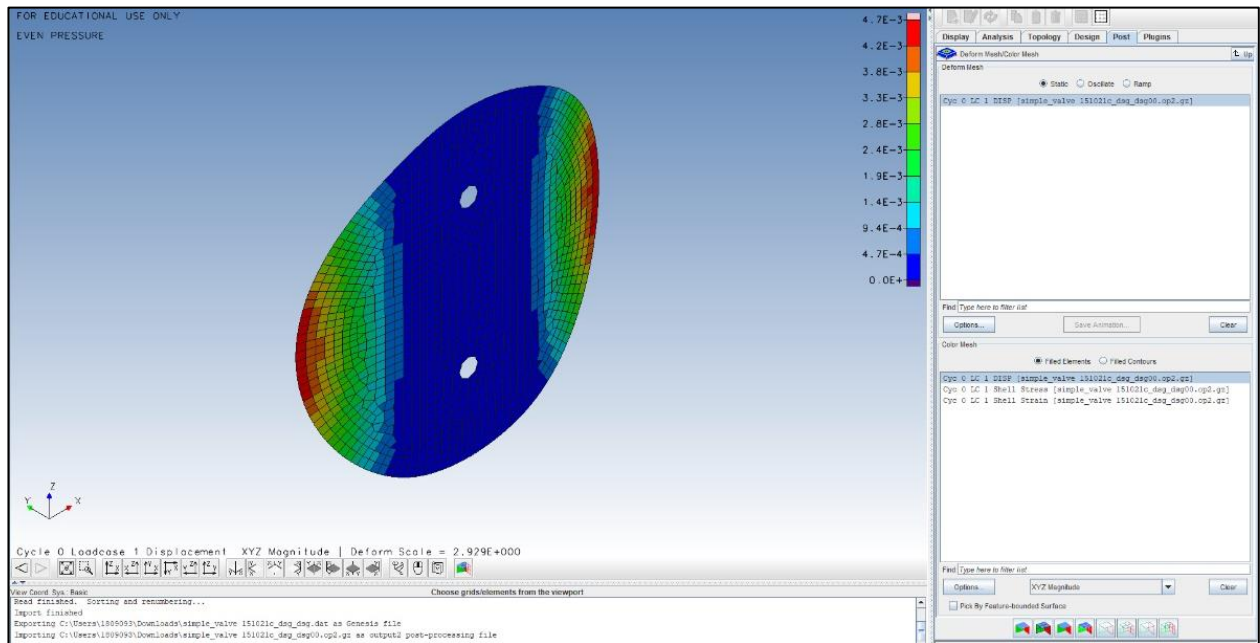


Figure 1: FEA Simulation of a piston rod. The different colours are indicators of variable values that help predict mechanical behaviour (SimScale, 2021)

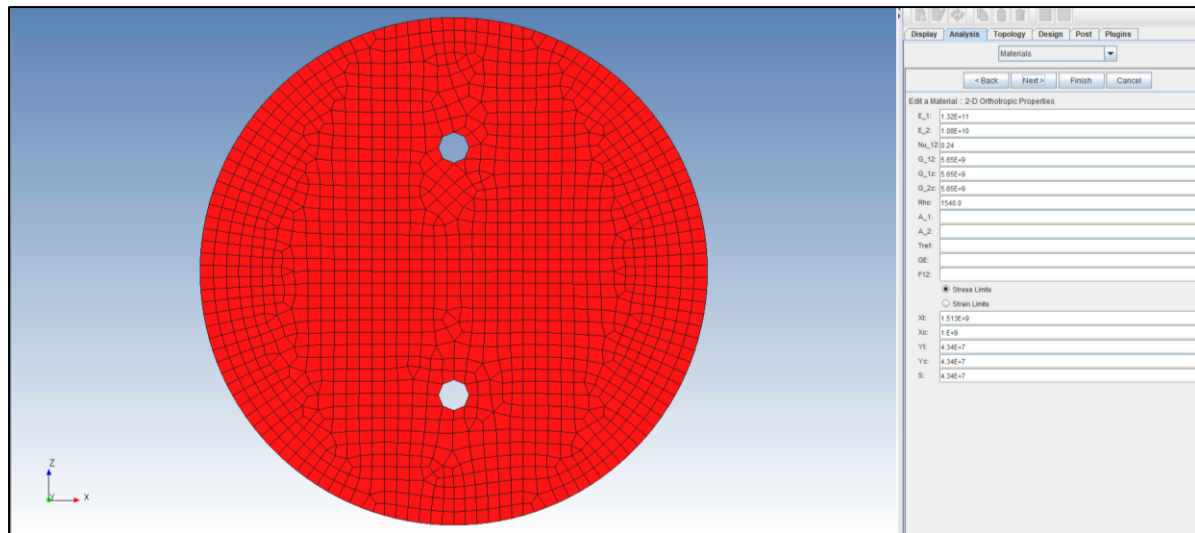
Task Two

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.

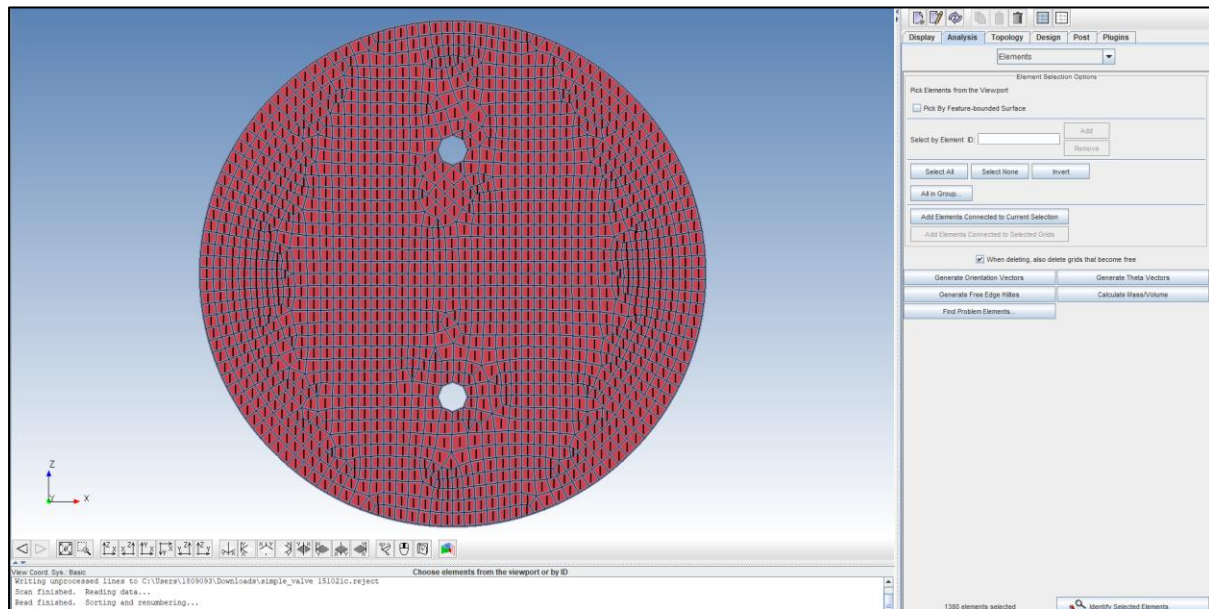
When carrying out this analysis in Design Studio, A NASTRAN desk model was provided to us. The file that was provided included overall dimensions, material in this case the material selected was steel, the mesh was also generated. The force applied was determined to be 0.94N per element to represent the hydrostatic head on the valve plate. The model was also constrained in the Z axis.

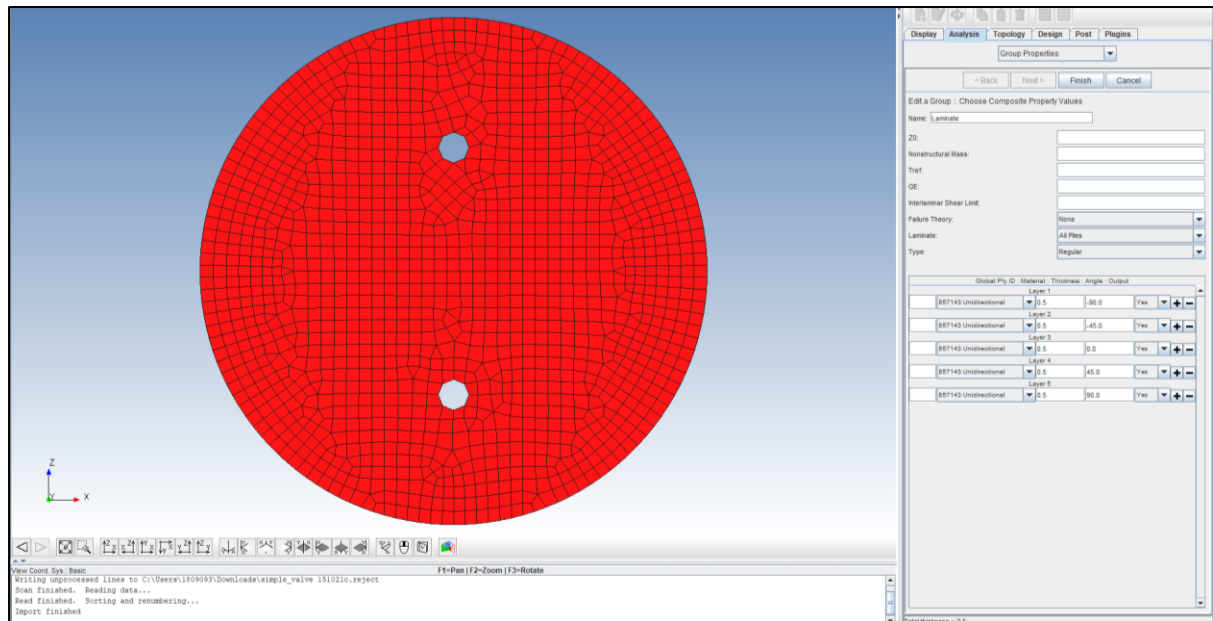


The initial analysis was run and the mass of the steel baseline plate is 0.25kg and results determined the deformation of 4.7mm. The tolerance for the valve was not met. The model was then created for the composite plate from this baseline and the parameters were altered to simulate a composite disk made from 25 plies of carbon fibre reinforced polymer (CFRP) in five layers in alternative orientations.

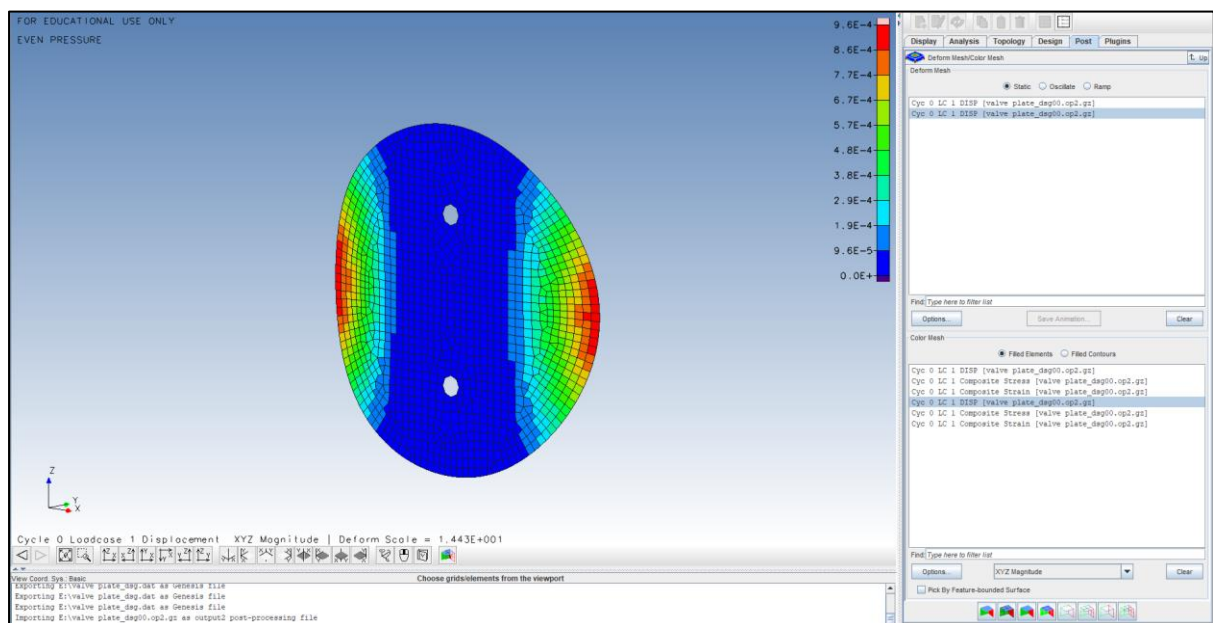


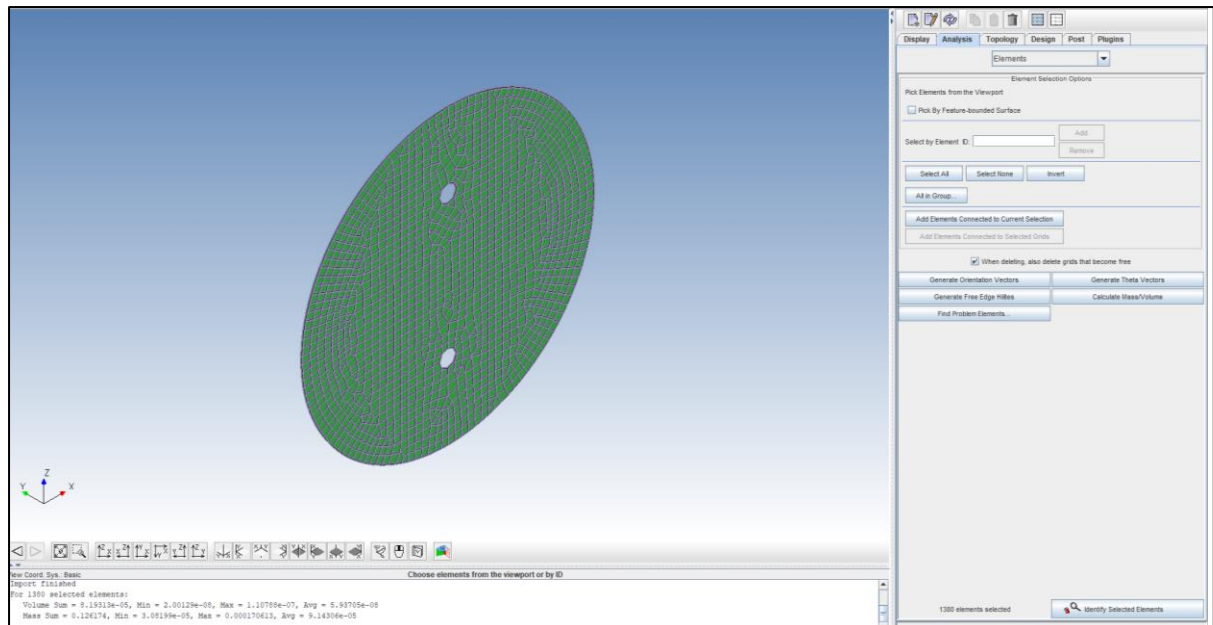
The material properties of the disk would also need to be altered to the chosen material (CFRP). And the vectors aligned on the +Z Axis





Next, the model can be altered to PCOMP/PCOMPG. This will in turn allow the model to be broken down in to layers for customisation. The layer thickness was specified as 0.5mm and each layer was given an orientation.



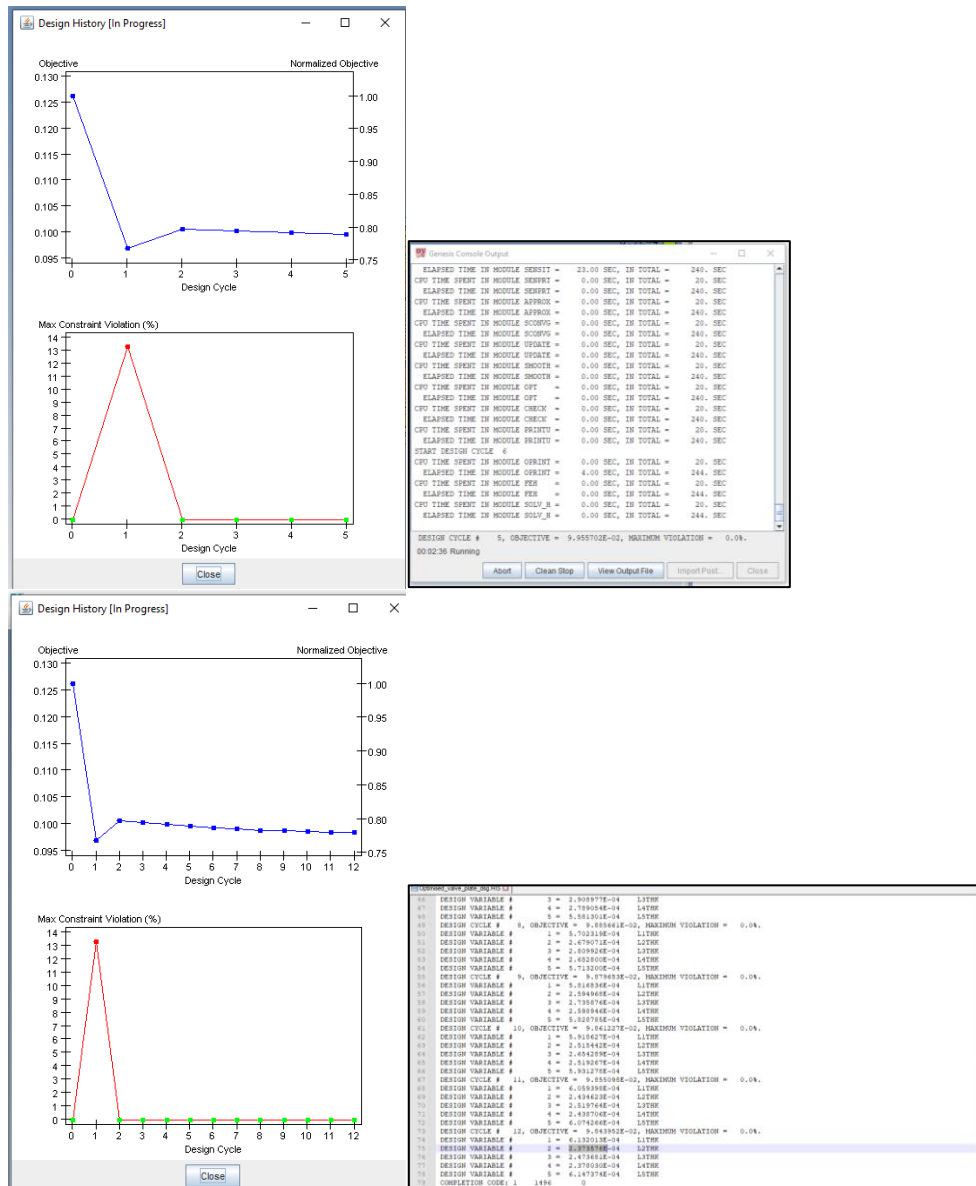


The results from the initial analysis showed a displacement of 9.6mm which is out of the range specified. Therefore this indicates that the CRFP can be more light and flexible but does not have much resistance to force that would be applied. Design studio is then utilised to modify the number/direction of plies in each layer to ultimately 'increase the stiffness of the material and therefore allow the material to resist the forces being applied far greater.

Task Three

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, zero degrees, 45 degrees and 90 degrees.

Design studio was used to optimise and minimise the mass of the plate while meeting a displacement constraint of 5mm. The number and orientation of plies were specified to be -90, -45, 0, 45 & 90°.



The maximum number of cycles was increased to 30, this is required due to the software needing an end point. If this was not set then the program could continue to run if there was no solution found.

The number of cycles reached 12 during the optimisation before the software had sufficient convergence, the maximum deformation showed was 1.8mm and was under the specified target valve. But when analysing the total weight it resulted in the plate being 98g. this is because some of the layers reached thicknesses of 2.37mm. Therefore this would make the model unrealistic to manufacture.

Further optimisation was carried out with more discrete variables rather than independent variables to try and achieved some results that would be acceptable to manufacture whilst also having properties that compare to standard steel plates for the valve. This is due to design studio using the same thicknesses as the caron fibre cloth used in the plys. The simulation showed that it reached 13 cycles with a deflection of 1.1mm at a mass of 116g

Simulation 1 layer structure:

DESIGN VARIABLE #	1 = 6.132013E-04	L1THK	-90 degrees =
DESIGN VARIABLE #	2 = 2.373576E-04	L2THK	-45 degrees =
DESIGN VARIABLE #	3 = 2.473681E-04	L3THK	0 degrees =
DESIGN VARIABLE #	4 = 2.378030E-04	L4THK	45 degrees =
DESIGN VARIABLE #	5 = 6.147374E-04	L5THK	90 degrees =

Simulation 2 layer structure:

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

Simulation 2 showed that this structure isn't as efficient and effective but due to the requirement of the simulation it is the most suitable to manufacture. Further simulation would be required to ultimately fine tune and determine the most effective structure for manufacture.

Task Four

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.

The prototype produced is to undergo analysis in the laboratory to understand the deformation of the valve plate when it is subjected to hydrostatic forces and compare these result against the simulations carried out.

Prior to conducting any experiment/analysis suitable personal protective equipment (PPE) should be worn at all times, these include the following: Safety glasses, overalls/lad coat and gloves. There should also be relevant safe working procedures and a risk assessment conducted prior to even starting.

The experiment will be conducted by using a compressor to force the liquid through a housing, the following equipment will be used: Air compressor, pigging system (to separate the water from the liquid), prototype valve, strain gauges (to monitor the strain applied to the prototype, clamp brackets to hold the valve securely.

Step 1 is to fit the strain gauges to the valve plate.

Step 2 Fill a section of the pigging system with petroleum and place the pig on top.

Step 3 Connect the air hose connection to seal the section.

Step 4 Set the pressure on the compressor to 5.69 PSI.

Step 5 Apply the pressure for a period of 24 hours for adequate data to be obtained

Step 6 Repeat experiment to ensure results are reliable.

Step 7 Obtain strain valve to calculate the maximum deformation using the following equation $\varepsilon = \frac{\Delta L}{L_0}$. The maximum deformation for the valve plate is 1.9mm.

Task Five

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

EN ISO 27509:2020: is the international standard for welding neck flanges, blind flanges, paddle spacers and spacer blinds (paddle blanks), valve/equipment integral flanges, orifice spacers, reducing threaded flanges and rigid interfaces for use in process piping for the petroleum, petrochemical and natural gas industries. (Anon., 2020)

Butterfly valves should normally be mounted with the spindle horizontal since this allows debris in the pipe invert to be swept clear as the valve is closed. Where the spindle is vertical solids can lodge under the disc at the spindle and cause damage to the seal. Disc position indicators are useful and strong disc stops integral with the body should be specified, so this gives certainty to the person when closing the valve.

Typically butterfly valves have been made to very large diameters when operating under very high pressures and water velocities. Many of these valves have proven successful in industry but when use to controlling the flow the maximum velocity should be limited. Use of such valves for isolation of air valves allows maintenance to be carried out on the air valve but since replacement of air valves is likely to be cheaper than in situ, flanged valves are the preferred option. (Toro, 2012)

Conclusion:

To Conclude, the experiments and simulations that have been carried out to compare standard CFRP with steel have stiffness to weight ratios that are very similar, but when the CFRP is optimised with the number of plies and the direction this allows less material to be used making it a more effective model to produce and also has other major benefits such as weight reductions. It is also evident that CFRP has a high fatigue resistance and the flexible carbon fibres crack far less frequently than traditional alternatives like steel and even concrete. Also when looking at the carbon fibre atoms they are bonded in the microscopic crystals that are aligned parallel to the long axis of the fibre making it incredibly strong for its size.

References

- Anon., 2020. *ISO*. [Online]
Available at: <https://www.iso.org/standard/78201.html>
[Accessed Jan 2021].
- Belytschko, J. F. a. T., 2007. *A First Course in Finite Elements*. [Online].
- Bi, Z., 2017. *Finite Element Analysis Applications (A Systematic And Practical Approach)*. 1st ed. s.l.:s.n.
- Direct, S., 2021. *Carbon Fibre Reinforced Polymer*. [Online]
Available at: <https://www.sciencedirect.com/topics/engineering/carbon-fibre-reinforced-polymer>
- Koutromanos, I., 2017. *Fundamentals of Finite Element Analysis: Linear Finite Element Analysis*. 1st ed. s.l.:John Wiley & Sons.
- SimScale, 2021. *Finite Element Analysis*. [Online].
- Toro, A. D., 2012. *Computational Fluid Dynamics Analysis of Butterfly Valve*, s.l.: s.n.